

**SIEMENS**

**SINUMERIK 810/840D  
ShopTurn**

Training Manual

Edition 2008.01

Training Material

**SIEMENS**

**SINUMERIK 810/840D**

**Shopturn**

Valid for

Control	Software
SINUMERIK 810/840D	6.4

## **Module content**

### **Operating and Programming**

<b>CNC Basic Principles Turning Machine Tool</b>	<b>A100</b>
<b>Operator components</b>	<b>A101</b>
<b>Switching the machine On</b>	<b>A102</b>
<b>Manual operating area ShopTurn</b>	<b>A103</b>
<b>Manual machine</b>	<b>A104</b>
<b>Program manager</b>	<b>A107</b>
<b>Tool Management</b>	<b>A108</b>
<b>Zero Offsets</b>	<b>A109</b>
<b>Spindles</b>	<b>A110</b>
<b>General program structure</b>	<b>A111</b>
<b>Program Header</b>	<b>A112</b>
<b>Programming Straight - Circle</b>	<b>A113</b>
<b>Programming Drill and Positioning</b>	<b>A115</b>
<b>Turning standard cycles</b>	<b>A117</b>
<b>Turning contour turning</b>	<b>A119</b>
<b>Milling standard cycles</b>	<b>A121</b>
<b>Milling contour milling</b>	<b>A123</b>
<b>Various - Set mark - Subprogram</b>	<b>A127</b>
<b>Various - Sub spindle</b>	<b>A131</b>
<b>Various settings</b>	<b>A133</b>
<b>Various transformation</b>	<b>A134</b>
<b>G - code editor</b>	<b>A138</b>
<b>Drawings for the examples</b>	<b>A141</b>

## 1 Brief description

**Objective of the module:**

This module explains the basic principles of operating and programming of a turning machine.

**Description of the module:**

In this module the following basic principles of CNC-turning machine will be explained:

- Which machine axes and machining planes are available on a turning machine?
- Which co-ordinate systems and important points which must be observed on a turning machine?
- What are zero-point-offsets?
- Which tool offsets will be taken into account?
- What possibilities are there for inputs for the programming and for co-ordinates?

**Content of the module:**

- Machine axes and machining planes
- Co-ordinate systems and important points on the machine
- Zero-point-offsets on a turning machine
- Tool offsets
- Co-ordinate programming (polar co-ordinates, absolute dimensions, incremental dimensions)

## Basic principles Turning Machine tools

Machine axes and machining planes

 **Section 2**

Co-ordinate systems and important points on the machine

 **Section 3**

Zero-point-offsets on a turning machine

 **Section 4**

Tool offsets

 **Section 5**

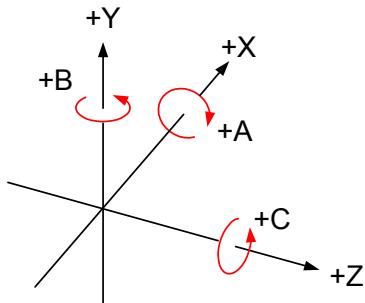
Co-ordinate programming (polar co-ordinates, absolute dimensions, incremental dimensions)

 **Section 6**

## Section 2

### Machine axes and machining planes

#### 2.1 Machine axes



Axes of numerically controlled horizontal lathes (turning machines)

The axes of numerically controlled machines are specified as per DIN 66217 or ISO 841 in a rectangular co-ordinate system. This system refers to the clamped work piece.

The axes of the machine are split up into linear and rotary axes.

Notes

##### Linear axes:

The three main linear axes are denominated by X, Y and Z.

##### Rotary axes:

A rotary axis is attributed to each of the linear axes:

- A (rotary axis to the linear axis X)
- B (rotary axis to the linear axis Y)
- C (rotary axis to the linear axis Z)

A simplified method of describing the co-ordinate system is by means of the „Right Hand Rule“:

The axes of an NC-machine are defined according to this rule, where the Z-axis is perpendicular to the chuck axis and where the fingers are pointing away from the work piece in the positive direction.

- The thumb points in direction +X
- The first finger in direction +Y
- The middle finger in direction +Z

For representation of the co-ordinate system on a turning machine the hand must be rotated to suit.

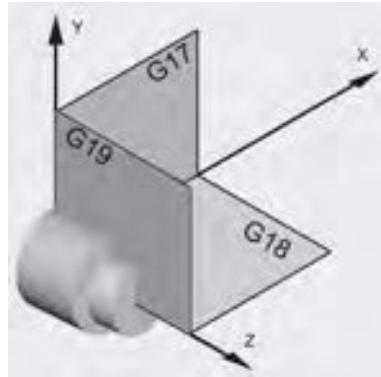
#### 2.2 Machining planes

Two of the co-ordinate axes form one working plane. The third co-ordinate axis (tool axis) is always perpendicular to this plane and thus specifies the infeed axis for the tool.

## Section 3

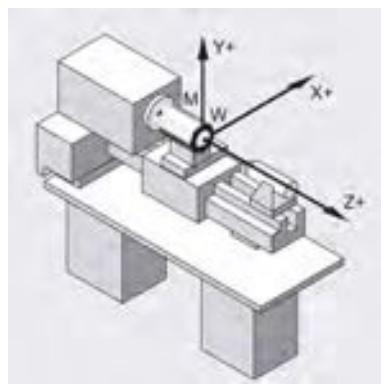
Notes

### Co-ordinate systems and important points on the machine



When programming it is important to inform the control unit regarding the plane in which machining is to take place in order to make sure that tool correction values for instance are evaluated correctly.

Plane	Tool axis	G-Code	Face
X/Y	Z	G 17	End face
Z/X	Y	G 18	Rotation plane
Y/Z	X	G 19	Peripheral plane



Co-ordinate systems of a machine

A differentiation is made on a machine between the machine and the work piece co-ordinate system.

The machine co-ordinate system (MCS) refers to points and axes on the machine.

The work piece co-ordinate system (WCS) refers to points and axes on the work piece.

The following points within the machining space therefore result from these co-ordinate systems.

#### Machine zero point:



The machine zero point **M** is a fixed point on the machine that must be seen as the reference for all (derived) measuring systems. It is thus the reference for all other co-ordinate systems. See also the documentation by the machine manufacturer.

#### Work piece zero point:



The work piece zero point **W** is the origin of the work piece co-ordinate system. It is defined by specified distances to the machine zero point. It can be selected at will and should be chosen as being that point on the drawing from which most of the dimensioning originates

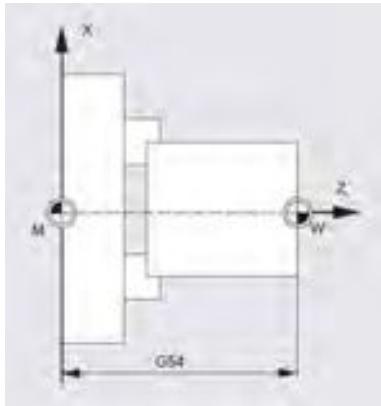
#### Machine reference point:



The machine reference point **R** is a point on the machine to which the measuring system of the machine refers. The position of the reference point is determined by the machine manufacturer. When the reference point is approached the position readout of the control unit is matched to the machine co-ordinate system

## Section 4

### Zero-point-offsets on a turning machine



A zero-point-offset (ZPO) is an offset of the zero point of the absolute co-ordinate system by a certain amount in the axes X, Y and/or Z.

Hence the zero-point-offset is a specification of a new reference point for a co-ordinate system with reference to an existing zero point.

A differentiation must be made between the following zero-point-offsets:

1. Basic zero point offset

The activation of the basic zero point offset is achieved by means of a machine datum (MD). This MD can be set by the user.

It is shown as a ZPO, which can be selected.

2. Selectable (G54 - G57)

There are at least four selectable zero point offsets for each CNC axis. These offsets are alternatively operative.

Hint: The documentation by the machine manufacturer must be observed.

3. External:

In addition to all offsets, which determine the work piece zero point, an external zero-point-offset for instance by means of a hand wheel adjustment (DRF-offset) can be overlaid.

4. Programmable:

The programmable ZPO provides the means for programming additional ZPOs for geometrical and additional axes.

These ZPOs will be cleared at the end of the program or by „Reset“.

Notes

If on the drawing several reference points are used for dimensioning, these reference points can be simplified for programming by means of a zero point offset.

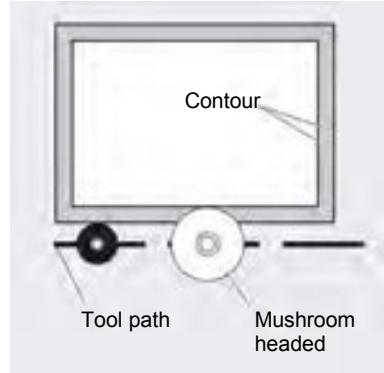
Hints. After „scratching“ it is usual to carry on with the basic zero point.

## Section 5

Notes

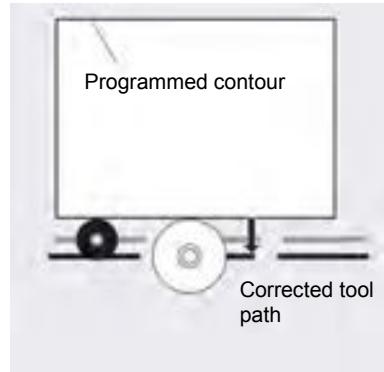
### Tool corrections

#### What are tool offsets?



During programming the tool diameters and tool lengths are not taken into account. The work piece dimensions are programmed directly, e.g. on the basis of the machining drawing. During machining the tool paths are controlled depending on tool geometry such that the correct contour will result independently of which tool is being used.

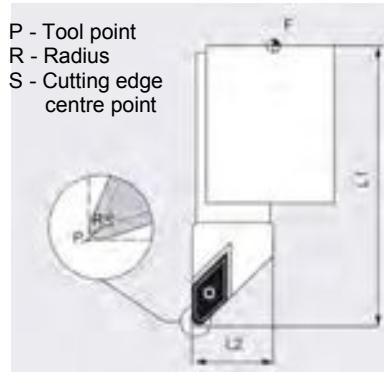
#### The control unit corrects the tool path



The tool data are entered separately into the tables "Tool list" and "Tool wear" (See *Module A108 - Menu Tool management*). In the program only the required tool will be called up. During the program execution the control unit fetches the correction data from the tool tables and corrects the tool path accordingly for the individual tools.

#### Which tool offsets are there?

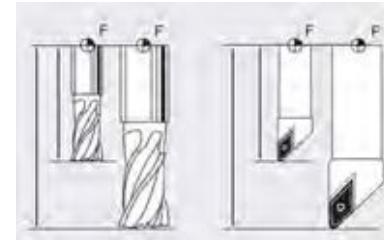
?



The offset memory for a tool contains:

The tool type: The type determines which tool data are required and how they are evaluated (e.g. roughing tool, finishing tool, grooving tool).

The dimensions: Tool length, cutting edge radius, tip angle. These consist of several components (geometry, wear). These components are utilized by the control unit to evaluate a resulting value. (e.g. Total length, total radius). The respective basic dimension is used when the offset memory is activated.

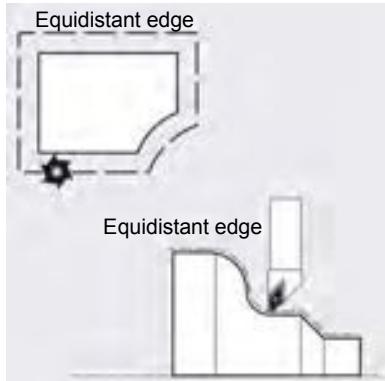


#### Tool length offset

These values are used to take into account the differing lengths of the tools used. The tool length is defined as the distance between the toll carrier reference point and the tool tip. This measured length is entered into the tool list. From this and the wear values the control unit evaluates the traversing distances in direction of the infeed.

Notes

#### Tool radius offset



To permit the direct programming of a contour, the control unit must traverse the tool along an equidistant path parallel to the programmed contour taking into account the radius of the tool used. To this end the programmed path of the tool centre is shifted depending on the tool radius and the direction of motion such that the cutting edge traverses exactly along the edge of the finished contour. The tool radius correction is activated by G41 / G42, where the selection depends on the machining and the co-ordinate on the machine

Example:

Lathe, machining behind the work piece centre:

G41 Internal machining  
G42 External machining

## Section 6

Notes

Co-ordinate programming  
(polar co-ordinates, absolute dimensions, incremental dimensions)

### Polar co-ordinates

The right angled co-ordinate system is used if the machining drawing is dimensioned for right angles. In case of work pieces that are dimensioned with radial arcs and angular dimensions it is more sensible to specify positions with polar co-ordinates. This is possible if a straight line or a circle is being programmed.

Polar co-ordinates have their zero point at the "Pole".

Example:

The points P1 and P2 could then – with reference to the **Pole** – be described as follows:

P1: Radius = 100 plus angle = 30°  
P2: Radius = 60 plus angle = 75°

### Cartesian co-ordinates

#### Absolute dimensions (G90)

In case of absolute dimensions the position statements are always referred to the then valid zero point. With respect to the tool motion, the absolute dimension statement describes the position to which the tool is to traverse.

Example:

The position statements for the points P1 to P3 in absolute dimensions are **with reference to the zero point**:

P1: X20 Y35  
P2: X50 Y60  
P3: X70 Y20

#### Incremental dimension (G91)

In case of machining drawings on which the dimensions are not referred to the work piece zero point, but to another point on the work piece, there is the possibility of incremental dimension input (chain dimensions).

In case of incremental dimension input the position statement refers in each case to the previously programmed point.

Example:

The position statements for the points P1 to P3 as chain dimensions are:

P1: X20 Y35; (with reference to the zero point)  
P2: X30 Y20; (referred to P1)  
P3: X20 Y-40; (referred to P2)

## 1 Brief description

### Objective of the module:

This module aims to explain, recognition and understanding the usual operating components of the Sinumerik 840D with ShopTurn.

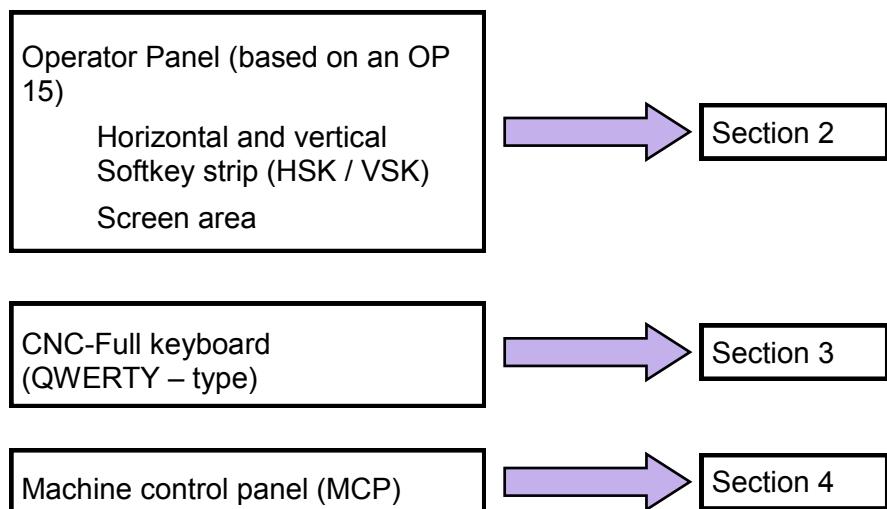
### Description of the module:

The general operation of a Siemens Control unit 840D with ShopTurn will be described.  
Depending on the machine manufacturer the following operating elements can be used:

Operator Panel (OP)  
CNC-full keyboard  
Machine control panel (MCP)

### Content of the module:

Operator Panel (based on an OP 15)  
- Horizontal and vertical Softkey strip (HSK / VSK)  
- Screen area  
CNC-full keyboard (QWERTY – type)  
Machine control panel (MCP)



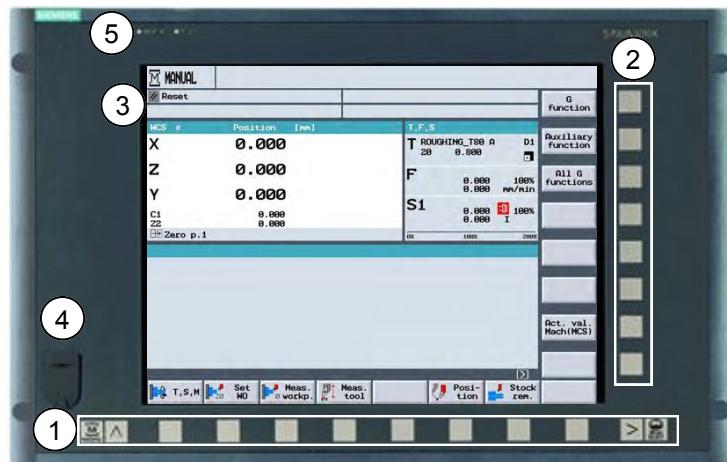
## Section 2

Notes

### Operator panel (based on an OP 15)

The operator panel (OP) consists of the following operating elements:

- Membrane keyboard with 8 + 4 horizontal and 8 vertical Softkeys
- 15“-TFT-colour display
- Front-USB-connection



- ① Horizontal Softkey strip with 4 screen keys (2 each located on the left and right side) (HSK)
- ② Vertical Softkey strip (VSK)
- ③ 15“ TFT-colour display
- ④ Front-USB-connection, e.g. for connection of external memory media
- ⑤ Status-LED: Power  
Status-LED: Temp (if ON, the temperature of the OP has increased)

#### 2.1. Horizontal and vertical Softkey strip (HSK / VSK)

Softkeys are keys, the attributed function of which in each case is shown by text on a menu strip either to the left or above along the edge of the screen.

The horizontal Softkeys are used to access the individual operation area's including further menu layers. There is an associated vertical menu strip/Softkey strip for each of the horizontal menu points.

The vertical Softkeys are functions associated with the presently selected horizontal Softkey.

The function will be called up when the vertical Softkey is pressed.

The content of the vertical Softkey strip can therefore change once again if a sub-function to the selected function is chosen.

## Section 2

### Operator Panel (based on an OP 15)

The horizontal Softkey strip consists of :

8 Softkeys for the individual operation sectors of ShopTurn (see module A103 - „Manual operation sector ShopTurn“) and

4 screen keys



Call-up of the operating area (Machine Manually, MDA or Machine Auto)

Revert: No function under ShopTurn



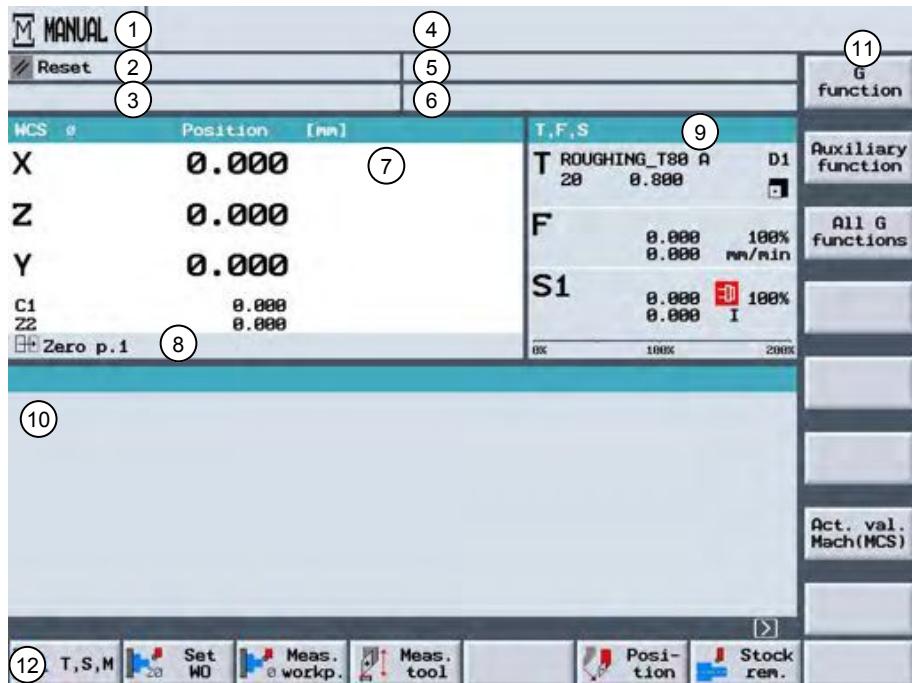
Extension: Selection of further functions that are included on the horizontal Softkey strip.



Selection of the five basic area's of ShopTurn.

## 2.2 Screen area

The ShopTurn screen is laid out as follows. For further variants see module 103 - „Manual operation sector ShopTurn“:



- |                                                   |                                                                                                       |
|---------------------------------------------------|-------------------------------------------------------------------------------------------------------|
| (1) Operation area                                | (9) Display of:<br>Active tool T<br>Present feedrate F<br>Spindle S<br>Spindle load factor in percent |
| (2) Status and program influence                  | (10) Working window                                                                                   |
| (3) Operating messages                            | (11) Vertical Softkey strip (VSK)                                                                     |
| (4) Alarm and message line                        | (12) Horizontal Softkey strip (HSK)                                                                   |
| (5) Program path                                  |                                                                                                       |
| (6) Program name                                  |                                                                                                       |
| (7) Position readout for the axes                 |                                                                                                       |
| (8) Display of the active zero point and rotation |                                                                                                       |

Notes

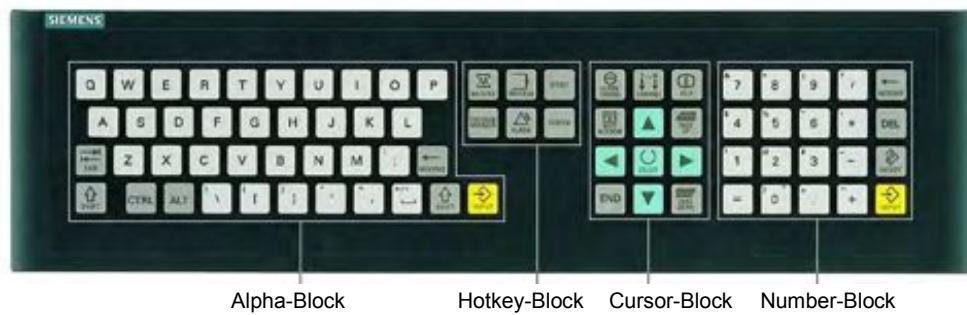
## Section 3

Notes

### CNC-Full keyboard (QWERTY – type)

A full CNC-keyboard for operating and programming can be integrated on operating panel of the machine depending on the type of OP used. The keys that are described here can also be located directly on the operator panel. The layout of the operating panel is described in the documentation of the machine manufacturer. Below follows a description of the basic keys of the CNC-Full keyboard.

#### CNC-Full keyboard KB 483



Alpha-Block:

The Alpha-Block features the letters A, ..., Z, the space key and the special character for the input of text.

Hotkey-Block:

The Hotkey-Block serves the direct selection of operation sectors.

Cursor-Block:

The Cursor-Block is used for navigation around the screen display.

Number-Block:

The Number-Block features the numbers 0 ... 9, the decimal point and special characters for the input of numerical characters and operators.

#### Keys in the Alpha-Block



##### Backspace

Clearing a value in the input field.  
If in the insertion mode, the character in front of the cursor will be cleared.

##### Tab

Not used for ShopTurn.



##### Shift

If the Shift-key is held depressed, the upper character on keys with double usage will be entered.

##### Ctrl

With the following key combinations navigation in the work plan and the G-Code-Editor is carried out:

Ctrl + Next Window: Jump to the beginning.  
Ctrl + End: Jump to the end.

##### Alt

Not used for ShopTurn.

## Section 3

CNC-Full keyboard (QWERTY – type)



### Input

- Acceptance of an edited value
- Opening/closing a directory
- Opening a file

Notes

## Keys in the Hotkey-Block



### Machine

Call-up of the operating area (Machine Manually, MDA or Machine Auto)

### Program

Call-up of the operating area "Program".  
The key corresponds to the Softkey "Prog. edit".

### Offset

Call-up of the operating area "Tools/Zero point offsets".  
The key corresponds to the Softkey "Tool zero pt.".



### Program Manager

Call-up of the operating area "Program manager".  
The key corresponds to the Softkey "Program".

### Alarm

Operating area "Messages/Alarms"..  
The key corresponds to the Softkey "Alarm list".

### Custom

This key is customized by the machine manufacturer.  
See documentation by the machine manufacturer.

## Keys in the Cursor-Block



### Alarm Cancel

Clearing an alarm that is identified with this symbol.

### Channel

Not used for ShopTurn.



### Help

Toggling between the work plan and the programming graphics as well as between the parameter mask and the help picture. In case of the G-Code editor the online documentation is called up.



### Next Window

By using the key combination „Ctrl + Next Window“ the cursor can be located at the start of the work plan and the G-Code-editor screen displays.



### Page Up or Page Down

Paging up or down in a directory or the work plan.

## Section 3

Notes

CNC-Full keyboard (QWERTY – type)

### Further keys in the Cursor-Block



#### End

To locate the cursor in the last input field of a parameter mask.



#### Cursor

Is used to select the various fields or lines on the screen.

While in the program listing, cursor ,right' opens a directory or a program. To change to the next level above the present level press cursor ,left'.



#### Select

With this it is possible so select amongst several given alternatives. The key corresponds to the Softkey "Alternativ".

### Keys in the Number-Block



#### Backspace

Clearing the value in the input field.

While in the insertion mode, just the character in front of the cursor will be cleared.



#### Del

Clearing the value in the parameter field.

While in the insertion mode, just the character in front of the cursor will be cleared.



#### Insert

Activation of the insertion mode or the pocket calculator.

#### Input

Acceptance of an edited value

Opens/closes a directory

Opens a file

## Section 4

### Machine control panel (MCP)

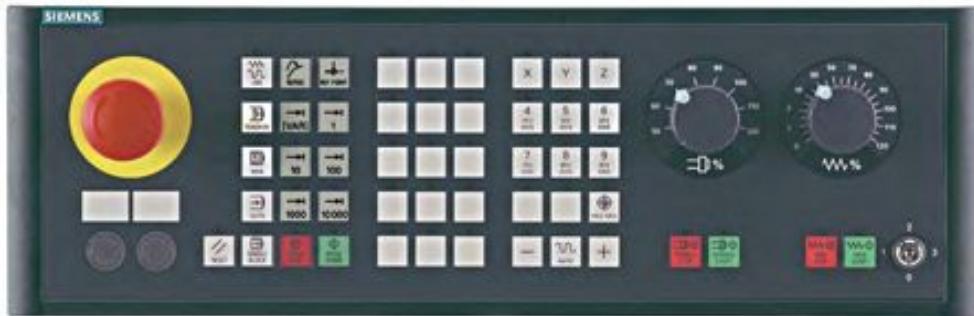
Depending on the type of operating panel the machine manufacturer may be using either a Siemens or his own machine control panel for the operation of the machine.

This section describes the Standard-keys of the Siemens machine control panel.

Depending on the machine further keys may be used; such information should be taken from the documentation by the machine manufacturer.

Notes

#### Machine control panel MCP 483



Below follows a description of the keys of the machine control panel and their function:



#### EMERGENCY-OFF key

Press this key in cases of emergency, i.e. if human life is endangered or if the machine or work piece could be damaged.

All drives will be braked to a standstill with the greatest possible braking torque.

**For further reactions that may be caused by pressing the EMERGENCY-OFF key please refer to the documentation by the machine manufacturer.**



#### Reset

Stops the machining from the presently running program. The NC-control unit remains synchronized with the machine. It is now in the basic condition ready to commence a new program run.

Clearing an alarm



#### Jog

Selection of the Mode of operation "Machine Manual".



#### Teach In

Not used for ShopTurn.



#### MDA

Selection of the Mode of operation "MDA".



#### Auto

Selection of the Mode of operation "Machine Auto".

## Section 4

Notes

Machine control panel (MCP)

### Single-Block

Running a program block-by-block (single block).



### Repos

Re-positioning, re-approach of a contour.



### Ref Point

Approach reference point.



### Inc Var (incremental feed variable)

Traverse through an incremental dimension with variable step lengths.

### Inc (incremental feed)

Traverse through an incremental dimension with a given step size of 1, ..., 10000 increments.

The actual length of an incremental step depends on a machine datum.

**Please observe the documentation by the machine manufacturer.**

### Cycle Start



### Cycle Stop

Stopping a program run.



### Axis keys

Axis selection.



### Direction keys

For traversing an axis either in the positive or negative direction.



### Rapid

For traversing an axis at rapid traverse rate (fastest speed).

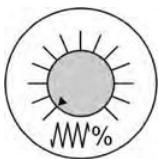


### WCS MCS

Toggling between the work piece co-ordinate system (WCS) and the machine co-ordinate system (MCS).

## Section 4

### Machine control panel (MCP)



#### Feed/Rapid traverse override

For increasing or reducing the programmed feedrate. The programmed feedrate is represented by 100% and can be varied within the range of 0% to 120%, in rapid traverse only up to 100%. The new value thus selected appears as an absolute value and as a percentage in the feed status display on the screen.

Notes



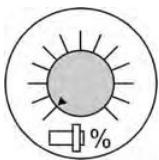
#### Feed Stop

To stop the machining from the running program in order to stop the axes.



#### Feed Start

Continuation of the program as from the present block and to increase the feedrate to the programmed value.



#### Spindle override

For increasing or reducing the programmed speed. The programmed speed corresponds to 100% and can be varied within the range of 0% to 120%. The new value thus selected appears as an absolute value and as a percentage in the speed status display on the screen.



#### Spindle Stop

To Stop the spindle.



#### Spindle Start

To Start the spindle.

#### Key-operated switch



Position 0

No key

Access stage 7

Lowest access stage



Position 1

Key 1 black

Access stage 6

Increasing access stages



Position 2

Key 1 green

Access stage 5

Highest access stage  
(Key switch)



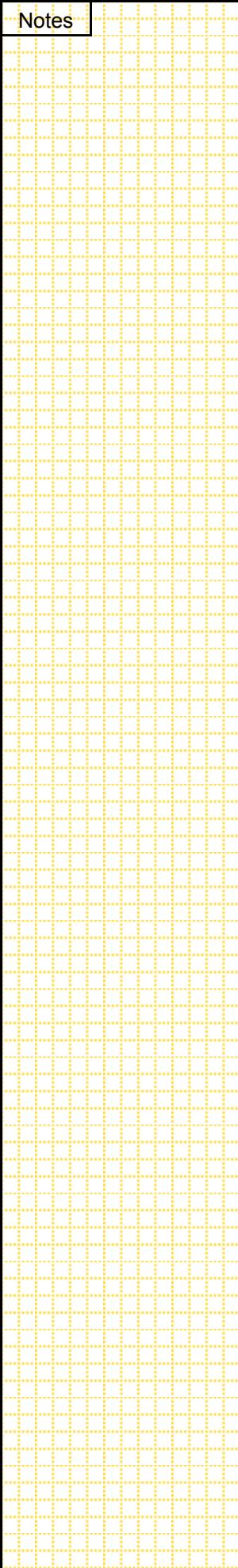
Position 3

Key 1 red

Access stage 4

Further access stages (Access stage 3 - 0) are possible by means of passwords.

Notes



# Switching the machine ON / Control unit - Referencing the axes

## 1 Brief description

### Objective of the module:

This module explains how to switch ON the machine and/or the control unit and how to approach the reference point of the axes.

### Description of the module:

Both the machine and the control unit must be switched ON before any work can be done on the machine. Following this all axes with incremental measuring systems must be referenced in order to enable the control unit to establish the position of the axes within the machine co-ordinate system. There is no need to reference any axes equipped with absolute measuring system. In this case the control unit recognizes the position of the axes automatically.

### Content of the module:

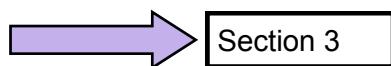
Switching ON the machine and the control unit

Approaching the reference point of the axes

Switching ON the machine and the control unit



Approaching the reference point of the axes



## Section 2

Notes

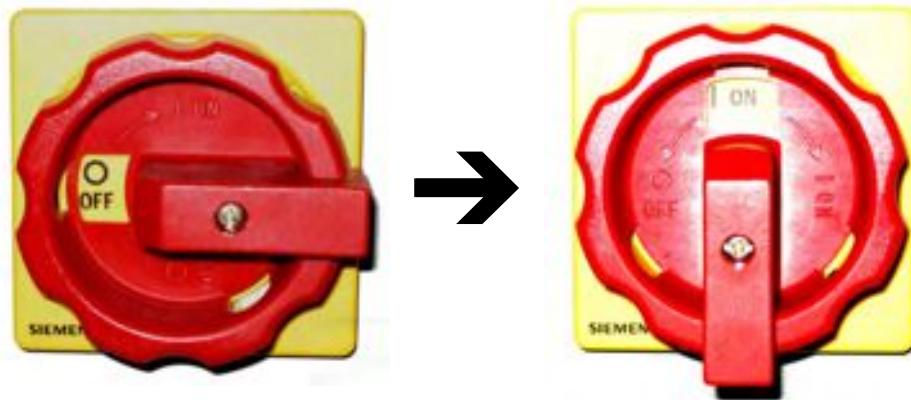
Switching the machine ON and the control unit



Please follow the precise switching-ON sequence as stated by the machine manufacturer.

### Switching-ON sequence:

1. Turn the main switch of the machine.  
(Normally the main switch will be found on the electrical cabinet.)



2. All EMERGENCY-OFF keys of the machine must be released  
(operating panels, electrical cabinet etc.).
3. The locations of the EMERGENCY-OFF keys are shown in the documentation by the machine manufacturer.

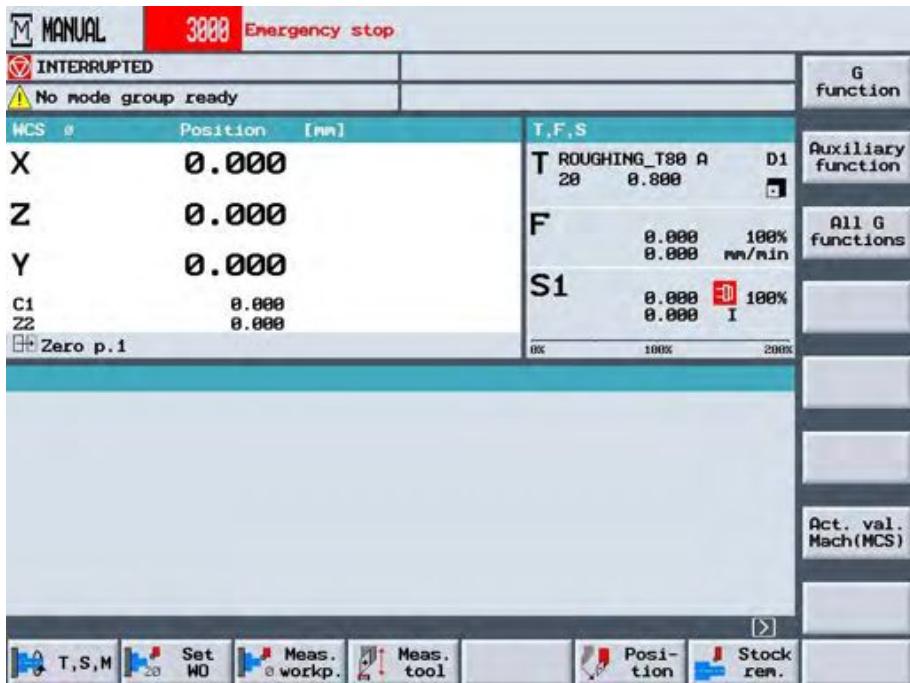


## Section 2

### Switching-ON the machine and the control unit

3. Switch ON the control unit. Depending on the individual machine this switch can be found on the machine control panel or the electrical cabinet of the machine or else the control unit is switched ON automatically when turning the main switch. Further to this see the documentation by the machine manufacturer.

The control unit boots. While booting a welcome image is displayed, followed by the basic ShopTurn screen:



4. Any fault messages that might be displayed can be cleared by means of the key „Reset“ on the machine control panel.



Notes

## Section 3

Notes

Approaching the reference point of the axes



**Before referencing the axes a check must be carried out to ensure that there is no danger of collisions during the approach.**

Machines with incremental measuring systems must be referenced after switching ON in order to synchronize the measuring system with the machine co-ordinate system.

**Activation of referencing on the machine control panel:**

press. press.

The respective LEDs above the keys are illuminated.

- ① Axes that need referencing are shown with an empty circle preceding the axis denomination.

MCS	Position [mm]
○ X	0.000
○ Z	0.000
○ Y	0.000
○ C1	0.000
○ Z2	0.000

### Approach of reference point

Depending on the commissioning of the machine there are various ways of referencing. Further to this see the documentation by the machine manufacturer.

1. Automatic referencing:

press.

The axes will be referenced one after the other.

2. Manual referencing



Press "Feed Start".

1. Select axis.

+ or - press.

Select all other axes one after the other and start the referencing motion by pressing either "+" or "-".

## Section 3

### Approaching the reference point of the axes



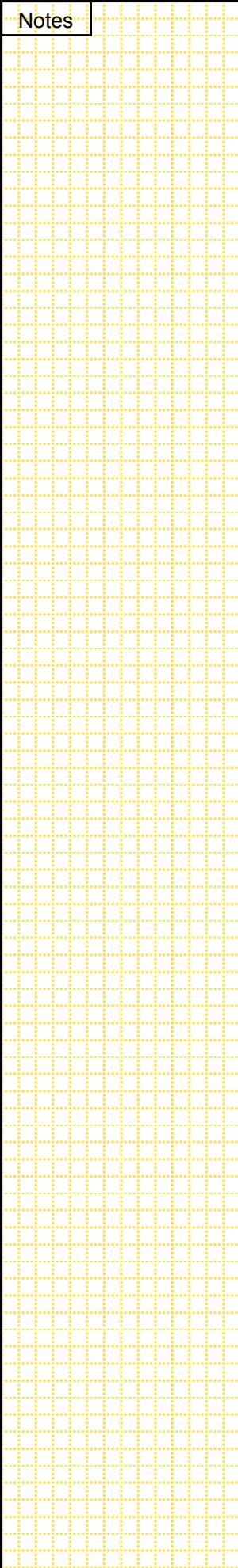
When referencing, keep a close eye on the feed override setting, since the axes are traversed at the feed rate that has been pre-set by means of a machine datum.

Notes

- ① Axes that have been referenced are shown on the display with a "Reference point symbol" preceding the axis name.

MCS	Position [mm]
① X1	0.000
Y1	0.000
Z1	0.000
C1	0.000
B1	0.000

Notes



## 1 Brief description

**Objective of the module:**

This module explains the operating and programming in the manual operating area under ShopTurn.

**Description of the module:**

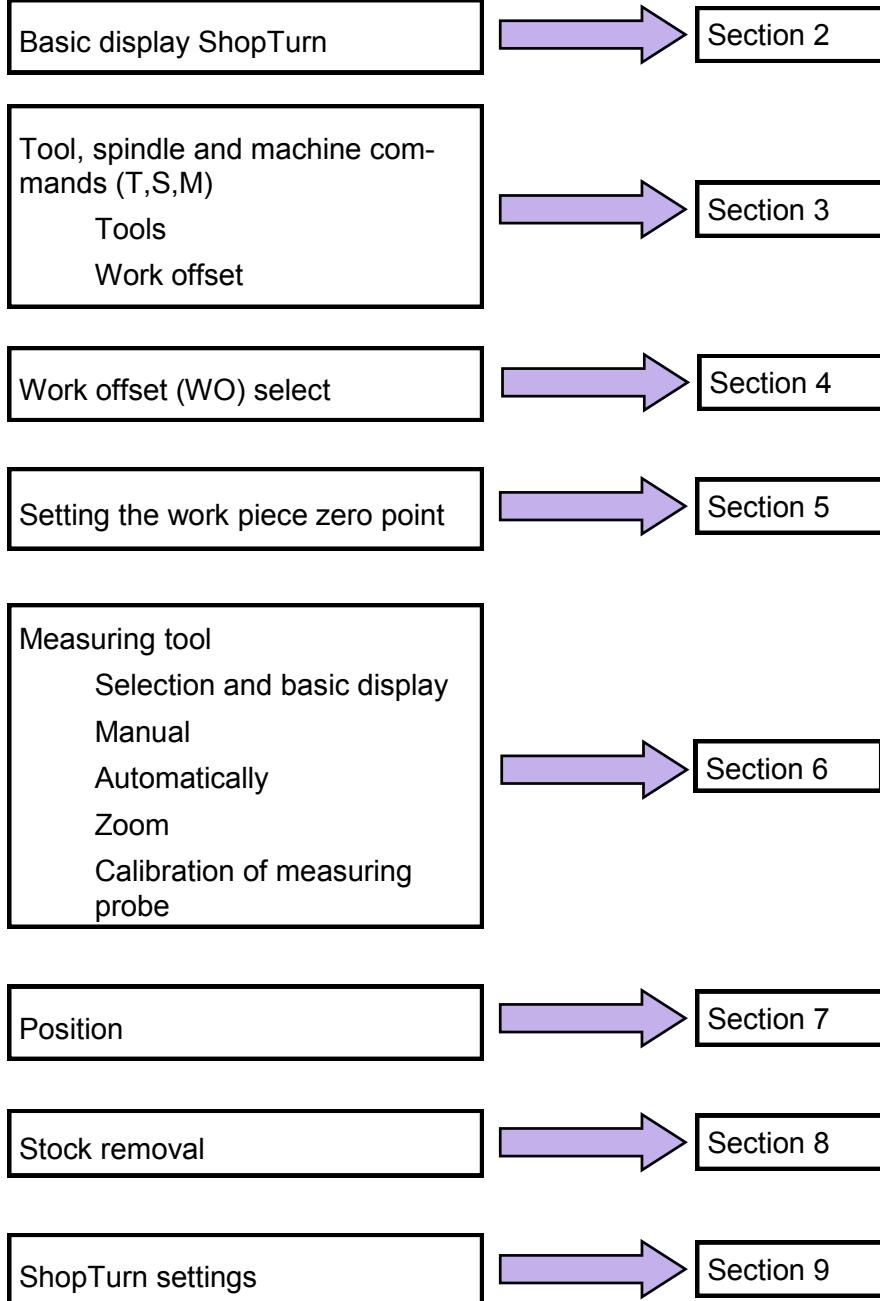
The individual sections of the screen will be described.

The Softkeys available in the mode-of-operation "Manual" for the setting-up and machining functions (set WO, Set Tool Offsets) will be described in this module.

**Content of the module:**

- Basic display ShopTurn
- Tool, spindle and machine commands (T,S,M)
- Tools
- Work offset
- Work offset (WO) select
- Setting the work piece zero point
- Selection and basic display
- Measuring work piece - Edge
- Measuring tool
- Manually
- Automatically
- Zoom
- Calibration of measuring probe
- Position
- Stock removal
- ShopTurn settings

# Manual Operating Area ShopTurn

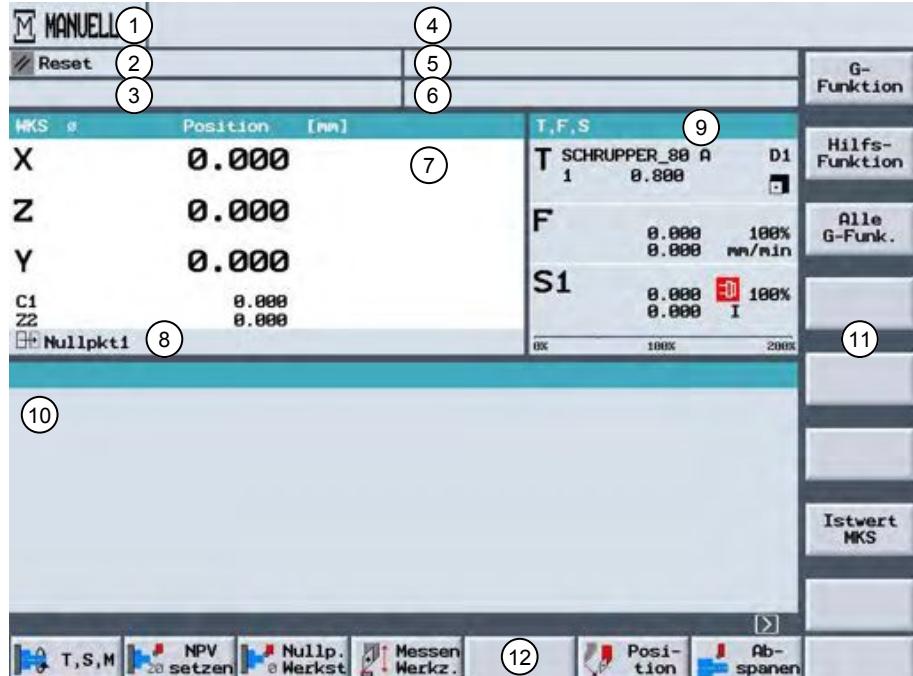


## Section 2

### Basic display ShopTurn

In this section the individual screen area's will be explained.

Notes



- |     |                                                      |      |                                                                                                |
|-----|------------------------------------------------------|------|------------------------------------------------------------------------------------------------|
| (1) | Operating area                                       | (9)  | Display of:<br>Active tool T<br>Current feed rate F<br>Spindle speed S<br>Spindle loading in % |
| (2) | Status and program influence                         | (10) | Work window                                                                                    |
| (3) | Operation messages                                   | (11) | Vertical Softkey-strip                                                                         |
| (4) | Alarm and message line                               | (12) | Horizontal Softkey-strip                                                                       |
| (5) | Program path                                         |      |                                                                                                |
| (6) | Program name                                         |      |                                                                                                |
| (7) | Position readout for the axes                        |      |                                                                                                |
| (8) | Display of the active zero point offset and rotation |      |                                                                                                |

- (1) **Display area,**  
depends on the selected mode-of-operation on the machine control panel (MCP)

MCP	Display area	Description
		<p>The mode-of-operation "MANUAL" (Jogging mode) is selected by pressing the key "JOG". The parameters set under "T, S, M..." (see also Section 3) are operative for all motions in the manual mode.</p> <p>Further functions in the mode-of-operation "JOG" are the approach of the reference point (Ref) as well as Preset (setting the actual value).</p>

## Section 2

### Basic display ShopTurn

Notes

MCP	Display area	Description
	 MDI	The mode-of-operation "Manual Data Input" is selected by means of the key "MDI". In the mode-of-operation "MDA" G-Code-commands can be entered and executed block-by-block to set up the machine or to carry out simple actions.
	 AUTO	In the mode-of-operation "MDA" individual program blocks or block sequences can be entered and started immediately with the key NC -Start without any reference to a main or sub-program.  The mode-of-operation "Automatic" is selected with the key "AUTO". In the mode-of-operation "AUTO" (continuous operation according to DIN) In the automatic mode a program is executed completely or just parts thereof. Additionally the progress of the machining can be followed graphically on the screen.

#### ② Status and program influence

Display section	Description
	Reset
	Interrupted
	Activated

Notes

#### ③ Operation messages

Display of operation messages by symbols

Display section      Description



**In case of conditions with this symbol a manual operation is required.**

Operation in case of the message "Stop":  
After the fault remedy the machining program will be continued after pressing "NC-Start".

Operation in case of the message "Wait":  
After the fault remedy the machining program will be continued automatically.

Stop: No NC-Ready  
Stop: BAG-Ready (mode-of-operation group)  
Stop: EMERGENCY-OFF activated  
Stop: Alarm with Stop activated  
Stop: M0/M1 activated  
Stop: Block completed in single block mode  
Stop: Cycle-Stop activated  
Wait: Read-in release missing  
Wait: Feed release missing  
Wait: Axis release missing  
Wait: Feed override > 0%  
Stop: NC-block erroneous  
Wait: for external NC-blocks  
Wait: Spindle release missing  
Wait: Axis feed rate is 0  
Stop: No channel ready  
Stop: SERUPRO has reached the search destination and the NCK has stopped. SERUPRO is the abbreviation for „Search Run by Program test“ and represents a new kind of block search.



**In case of conditions with this symbol a manual operation is not normally necessary..**

Wait: Remaining dwell in seconds or in spindle revolutions  
Wait: HiFu-acknowledgement missing  
Wait: Defined position not attained  
Wait: for positioning axis

## Section 2

### Basic display ShopTurn

Notes

Display section



Description

**In case of conditions with this symbol a manual operation is not normally required.**

Wait: for spindle  
Wait: for another channel  
Wait: due to a SYNACT-instruction  
Wait: Block advance activated  
Wait: for tool change acknowledgement  
Wait: for gear change  
Wait: for closed loop  
Wait: for tapping start  
Wait: for safe operation  
Halt: oscillation activated  
Wait: during access to a system variable

④

#### Alarm and message line

In case of a syntax error in the program or a machine fault (e.g. Emergency-OFF) the alarm-/message number including a clear text is displayed in the alarm line.

MCP



Display section



Description

After remedy of the fault (correction of the machine error) the fault display can be reset.

CNC-Keyboard



After remedy of the fault (correction of the syntax error) the fault display can be reset.

⑤

#### Program path

Display section

**/\_N\_WKS\_DIR/\_N\_DOKU\_WPD**

Description

Display of the currently selected program directory.

⑥

#### Program name

Display section

**HOLE\_PATTERN**

Description

Display of the currently selected program.

## Section 2

### Basic display ShopTurn

#### ⑦ Position display for the axes

MCP	Display section	Description
		Switching between the machine co-ordinate system and the tool co-ordinate system is possible either by means of the <b>vertical Softkey</b> or on the <b>MCP</b> .

HCS	Position [mm]
X	0.000
Z	0.000
Y	0.000
SP1 22	0.000 0.000

Display of the available axes wit axis denomination and position data in the machine co-ordinate system (MCS).

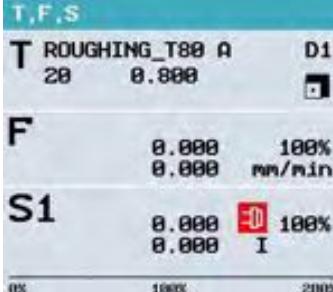
HCS	Position [mm]
X	0.000
Z	0.000
Y	0.000
C1 22	0.000 0.000

Display of the available axes wit axis denomination and position data in the work piece co-ordinate system (WCS), however, without programmable work offsets and system frames for cycles (ENS).

#### ⑧ Display of the work offset, rotation, mirroring and scaling

Display section	Description
   	Name of the currently active work offset, rotation, mirroring, rotation and scaling for the present machining sequence.

#### ⑨ Display of the feed rate and spindle values (TFS)

Display section	Description
	T: Name of the tool used for the current machining.  F: Display of the active feed rate for the current machining (top: actual feed rate, large digits during machining), as well as the display of the programmed feed rate (bottom) and the feed override in %.  S: Display of the active spindle speed for the current machining (top: actual speed, large digits during machining), as well as the display of the programmed spindle speed (bottom) and the speed override in %.

#### Hierarchy

- Main spindle before tool spindle and side spindle
- Tool spindle before side spindle

Notes

## Section 2

Notes

### Basic display ShopTurn

#### ⑩ Work window

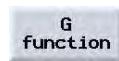
Display section      Description



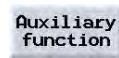
Depending on the pressed horizontal Softkey the associated parameter fields and Help pictures are displayed; in this example T,S,M - Mask.

#### ⑪ Vertical Softkey-strip (VSK)

Display section      Description



By pressing the VSK 1 "G-function" the most important G-functions will be displayed.



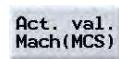
By pressing the VSK 2 "Auxiliary function" the available auxiliary functions at the time of issue will be displayed.



By pressing the VSK 3 "All G-functions" all G-functions will be displayed.



By pressing the VSK 4 "Basic block" the absolute axis positions, G-code commands and further programmed addresses as well as M-functions, which initiate a function of the machine, will be displayed.



By pressing the VSK 7 "Act. val. Mach(MCS)" the co-ordinate system can be switched over between the machine co-ordinate system (MCS) and the tool co-ordinate system (TCS).

## Section 2

### Basic display ShopTurn

#### ⑫ Horizontal Softkey-strip (HSK)

Display section      Description



By pressing the HSK 1 "T,S,M" the input mask "T,S,M" will be selected.



By pressing the HSK 2 "Set WO" the input mask "Set work offset" will be selected.



By pressing the HSK 3 "Meas. workp." the input mask "Measuring work piece" will be selected.



By pressing the HSK 4 "Meas. tool" the input mask "Measuring tool" will be selected.



By pressing the HSK 6 "Position" the input mask "Positioning" will be selected.



By pressing the HSK 7 "Stock rem." the input mask "Stock removal" will be selected.

**Action not possible: ref. pt. approach active**

Display of messages / Hints for erroneous operation / inputs in the line next to the ➤-Symbol above the horizontal Softkey-strip.



By pressing the key "Extension" on the Operator Panel (OP) the display of further Softkeys is possible.



By pressing the HSK 1 "ShopT. sett." settings for ShopTurn can be made.

Notes

## Section 3

Notes

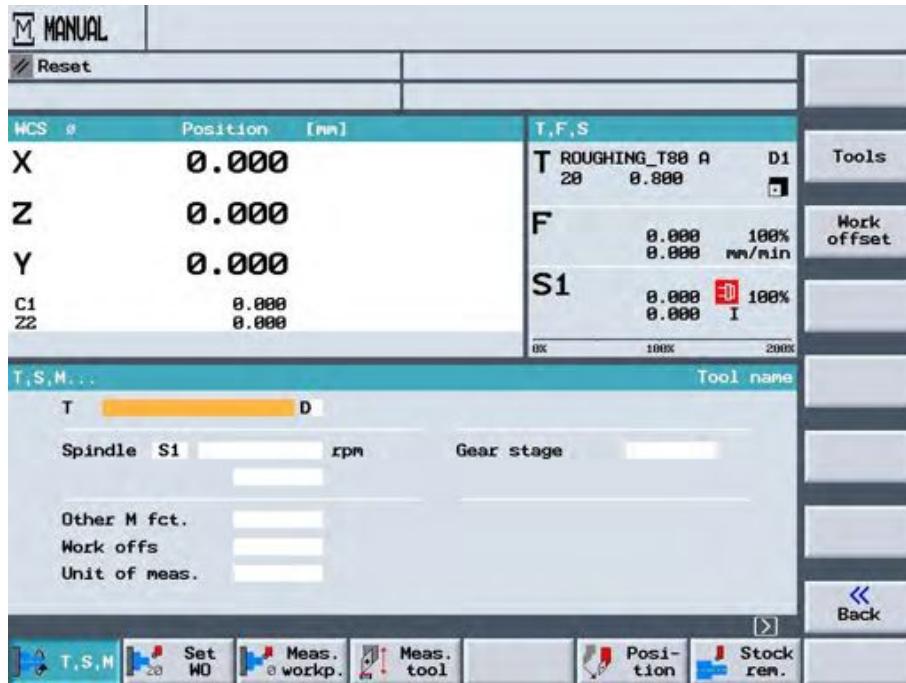
### Tool, spindle and machine commands (TSM)

#### 3.1 T,S,M

Input mask for tool, spindle and machine commands



By pressing the HSK 1 "T,S,M" the following input mask will be displayed on the screen.



#### T,S,M

#### Vertical Softkey-strip (VSK)

Display section



Description

By pressing the VSK 1 "Alternat.", active only depending on the selected input field/parameter, the selection/choice of alternative inputs is possible.

Tools

By pressing the VSK 2 "Tools" the tool list will be called up.

(See *Module A108 - Operating area - Tool management*).

Work offset

By pressing the VSK 3 "Work offset" the list with the work offsets will be called up.

Back

Pressing the VSK 8 "Back" initiates the return to the basic display ShopTurn.

## Section 3

### Tool, spindle and machine commands (TSM)

T,S,M

**Input mask, spindle and machine commands**

Notes



Display section      Description

- (1) Input of tool name "T".  
e.g.: T12 or Roughing Tool 80 (Alpha numerical).  
*(Operating area - Tool management see Module A108)*
- (2) Input of cutting edge number "D" (Numerical).
- (3) Selection of the machining spindle and selection with the Alternative-key; alternatively it is also possible to enter the turret location number.
- (4) Input spindle speed (Numerical).
- (5) Input of spindle M-function.  
(CW/CCW/positioning), Selection with the key Alternat.
- (6) Input of other M-functions. (Numerical)
- (7) Input of the work offset (*Operating area - Work offset, see Module A109*) (Numerical or with the key Alternat.)
- (8) Input of dimension unit (mm/in). (Selection with the key Alternat.)
- (9) Input of gear stage. (Selection with the key Alternat.)

## Section 3

Notes

### Tool, spindle and machine commands (TSM)

#### 3.2 Tools

See also *Module A108 - Operating area - Tool management*.

Tools

By pressing the VSK 2 "Tools" the tool list will be displayed.

OFFSET										
Tool list										
Loc	Typ	Tool name	DP 1st cutting edge				Insert length			Alternat.
			Lngth X	Length Z	Radius		Insert length	12		
20	■	ROUGHING_T80 A	1 55.840	39.124	0.800 ←	95.0 80	12.0			In manual
21	□	DRILL_32	1 0.000	185.124	32.000	180.0				Delete tool
22	▲	FINISHING_T35 A	1 125.726	57.370	0.400 ←	93.0 35	12.0			Unload
23	■	ROUGHING_T80 I	1 -8.950	122.457	0.800 ←	95.0 80	18.0			Cutting edges
24	□	PLUNGE_CUTTER_3 A	1 85.124	44.124	0.200	3.000				Sort
25	▲	FINISHING_T35 I	1 -12.658	121.887	0.400 ←	95.0 35	8.0			
26	■	THREADING_T1.5	1 66.326	33.333	0.050					
27	□	CUTTER_8	1 87.833	74.621	0.800		3			
28	▲	PLUNGE_CUTTER_3 I	1 -11.736	135.124	0.100	3.000		4.0		
29	□	DRILL_5	1 0.000	185.124	5.000	118.0				
30	■	BUTTON_TOOL_8	1 88.112	38.123	2.000					
31	□	THREADCUTTER_M6	1 0.000	145.132	3.000	1.000				
32										

In manual

By pressing the VSK 2 "In manual" the selected tool will be transferred.

Return to the basic display from which the selection was made.

(See *Module A108 - Operating area - Tool management*).

## Section 3

### Tool, spindle and machine commands (TSM)

#### 3.3 Work offset

See also Module A109 - Operating area - Work offset.

Work offset

By pressing the VSK 3 "Work offset" the list with the work offsets will be displayed.

OFFSET		HO 1 (G54)						In manual Further axes Clear Offset Position set Y Position set Z
WCS	MCS	X	55.840	Y	0.000	Z	39.124	
X	0.000							
Y	0.000							
Z	0.000							
X	Y	Z	X ↗	Y ↗	Z ↗	ΔZ		
Base	0.000	0.000						
HO 1	0.000	0.000	0.000					
	0.000	0.000	0.000					
HO 2	0.000	0.000	0.000					
	0.000	0.000	0.000					
HO 3	0.000	0.000	0.000					
	0.000	0.000	0.000					
Program	0.000	0.000	0.000	0.000	0.000	0.000		
Scale	1.000	1.000	1.000					
Mirror								
Total	0.000	0.000	0.000	0.000	0.000	0.000		
							[x]	
Tool list	Tool wear		Magazine	Work offset	R vari.			

In manual

By pressing the VSK 2 "In manual" the selected work offset will be transferred.

Return to the basic display from which the selection was made.

(See Module A109 - Operating area - Work offset).

Further axes

By pressing the VSK 3 "Further axes" further axes will be displayed on the screen.

Clear Offset

By pressing the VSK 4 "Clear offset" the values entered in the work offset will be deleted.

Position set Y

By pressing the VSK 6 "Position set Y" the values entered in the work offset will be transferred to the Y-Axis.

Position set Z

By pressing the VSK 7 "Position set Z" the values entered in the work offset will be transferred to the Z-Axis.

Notes

## Section 4

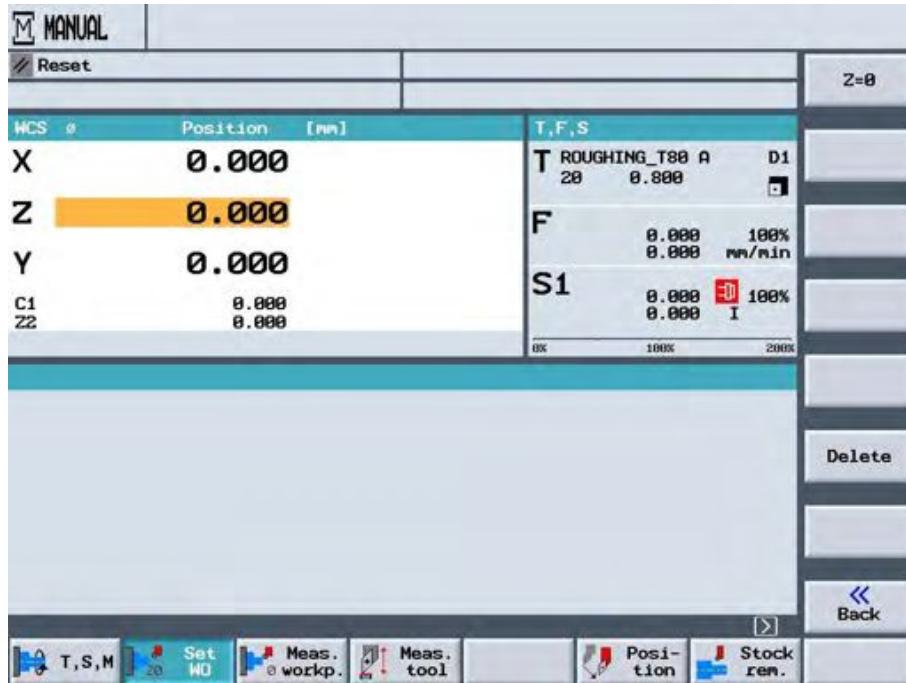
Notes

Set work offset

### 4.1 Selection and basic display screen



By pressing the HSK 2 "Set WO" the following input mask will be displayed on the screen.



Value input:

Enter the value of the work offset into the field highlighted by selection of the axis.

Hint:

The work offset value for the axes will be transferred and shown in the tool co-ordinate system (TCS).

A work offset (in this example work offset 1 / G54) must be activated. The difference between the original position to the new value entered will be written to the active work offset.

## Section 4

### Set work offset

WO

#### Vertical Softkey-strip (VSK)

Notes

Display section

Z=0

Description  
By pressing the VSK 1 "Z=0" the position readout of the Z - axis will be set to zero.

Delete

By pressing the VSK 6 "Delete" the work offset for all axes will be set to zero.

Back

Pressing the VSK 8 "Back" initiates the return to the basic display ShopTurn.

Hint:

After setting a position readout or deleting a work offset the display automatically changes back to the basic ShopTurn display.

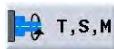
## Section 4

Notes

Set work offset

WO

**Work offset selection**



By pressing the HSK 1 "T,S,M" the mode-of-operation T,S,M will be selected.



In the T,S,M input mask select work offset .



By pressing the VSK 1 "Alternat." display the required work offset in the input field.



By pressing the key "Cycle Start" on the MCP the selected work offset will be activated.

Display area

Values for the work offsets (Example) assigned to the individual axes.



Display section      Work offset (example) selected.

HCS	Position [mm]
X	0.000
Z	0.000
Y	0.000
C1	0.000
Z2	0.000
Zero p.2	CC2

Display section      Values for work offsets (example) assigned to the individual axes.

WO

**Work offset unselect**



By pressing the HSK 1 "T,S,M" select the mode-of-operation T,S,M.



Select the input field Work offset in the input mask T,S,M.



By pressing the VSK 1 "Alternat." select to have **no** work offset displayed (empty input field)

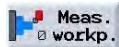


By pressing the key "Cycle Start" on the "MCP" the work offset will be de-activated.

## Section 5

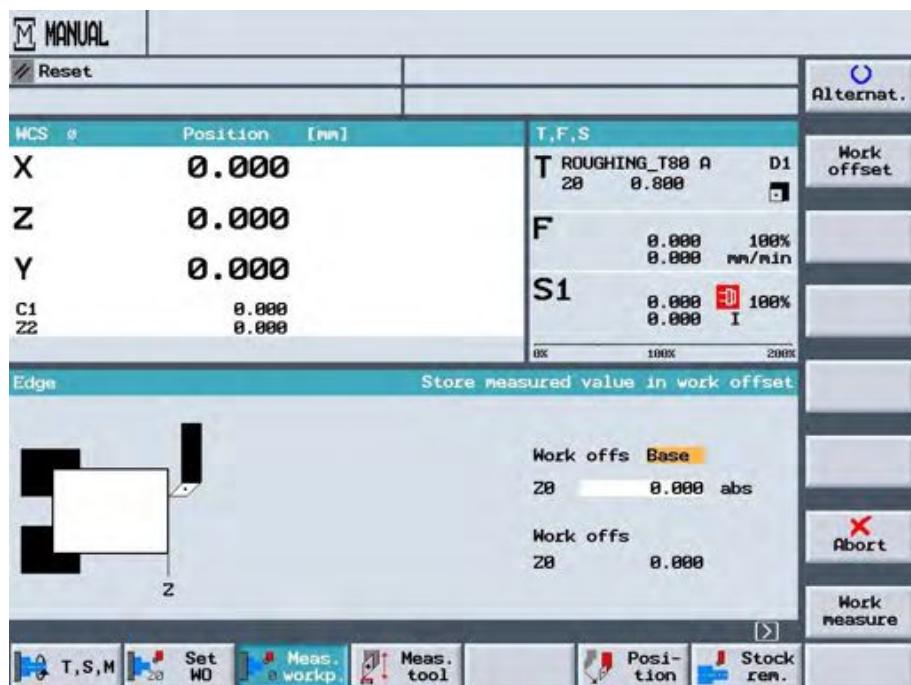
### Measuring work piece

#### 5.1 Selection and basic display



By pressing the HSK 3 "Meas. workp." the following input mask will be displayed on the screen.

Notes



The reference point when programming a work piece is always the work piece zero point. For the determination of the zero point the length of the work piece must be measured and the position of the cylinder end face in the Z-direction must be stored in a Work offset; i.e. The position is stored in the coarse offset and existing values in the fine offset will be deleted. The measuring of the work piece requires that a tool of known length is in the machining position.

#### Measuring work piece      Vertical Softkey-strip (VSK)

##### Display section



##### Description

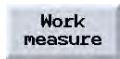
By pressing the VSK 1 "Alternat.", active only depending on the selected input field/ parameter, the selection/choice of alternative inputs is possible.



By pressing the VSK 2 "Work offset" the list with the Work offsets is called up (see section 3.3).



By pressing the VSK 7 "Abort" there will be no transfer of the entered values.  
Return to the basic display ShopTurn.

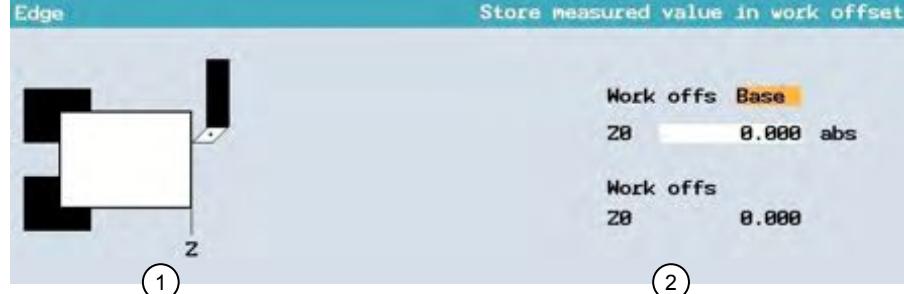


By pressing the VSK 8 "Work measure" the entered values will be transferred.

## Section 5

Notes

### Measuring work piece



- ① Help picture
- ② Input area for parameters

#### Parameters for the function "Zero point work piece"

Parameter	Help picture	Description and hints
Work offset		Work offset Work offset in which the measured value is to be stored.
Z0		Work piece edge

#### Measurement work piece zero point:

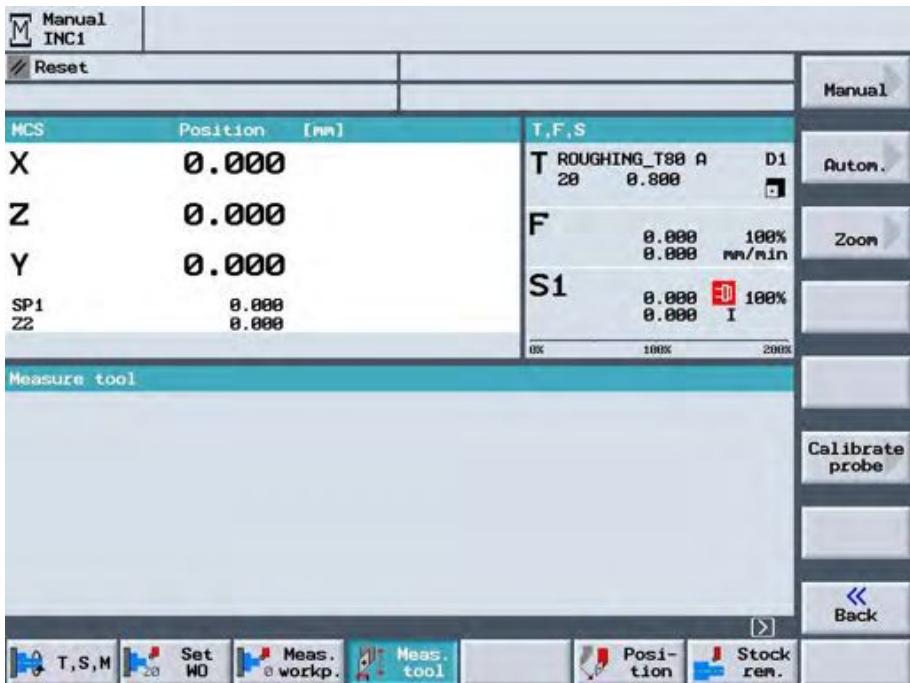
1. In the mode-of-operation "Manual" in the basic display Machine select the Softkey "Zero pt. workp.".
2. Select the required offset, in which the position of the cylinder end face is to be stored.
3. Press the Softkey "Work offset.".
4. Set the cursor on the required Work offset.
5. Press the Softkey "In manual".
6. 'Scratch' the work piece with the tool in the Z-direction.
7. Enter the command position of the work piece edge Z0.
8. Press the Softkey "Work measure".

The work piece zero point and with it the Work offset will be evaluated. For this the tool length will be taken into account automatically.

#### 6.1 Selection and basic display



By pressing the HSK 4 "Meas. tool" the following input mask will be shown on the screen.



When a program is being executed a great variety of tool geometries must be taken into account. These are stored as so-called tool offset data in the tool list. Each time a tool is called up the control unit uses these tool correction data. The tool length offset can be determined with the function "Tool measurement" either manually, automatically (using a measuring probe) or by means of Zoom.

#### Manual tool measurement

For manual measurement the tool is moved manually to a known reference point in order to establish the dimensions in the X and Z-direction. From the position of the tool carrier reference point and the reference point ShopTurn can evaluate the tool offset data. Either the work piece edge or, for measurement in the Z direction, the chuck of the main or the side spindle can be used. The position of the work piece edge is stated during the measurement. The position of the chuck, however, must be stated before the measurement.

Notes

## Section 6

Notes

### Measuring of tools

#### Automatic tool measurement

For automatic measurement the tool dimensions in the X and Z direction are established using a measuring probe. From the known position of the tool carrier reference point and that of the measuring probe ShopTurn can determine the tool offset data. If tools are to be measured using a probe, the machine manufacturer must provide a special measuring cycle. If there is a second measuring probe for the side spindle the machine manufacturer must provide details in a machine datum. Before the measuring process itself the tool must be entered into the tool list. Furthermore the probe must be calibrated beforehand.

#### Tool measurement using a zoom

For the determination of the tool dimensions it is also possible to use a Zoom, if one is fitted on the machine. ShopTurn evaluates the tool offset data from the known positions of the tool carrier reference point and the hair-line cross in the zoom.

#### Measuring tool

#### Vertical Softkey-strip (VSK)

Display section

Description

Manual

By pressing the VSK 1 "Manual" the operating section for the manual measuring-up of tools is called up.  
(See Section 6.2).

Autom.

By pressing the VSK 2 "Autom." the operating section for the automatic measuring-up of tools is called up.  
(See Section 6.3).

Zoom

By pressing the VSK 3 "Zoom" the operating section for the measuring-up of tools using the zoom function is called up .  
(See Section 6.4).

Calibrate probe

By pressing the VSK 6 "Calibrate probe" the operating section for the calibration of the probe is called up.  
(See Section 6.5).

< Back

Pressing the VSK 8 "Back" initiates the return to the basic display ShopTurn.

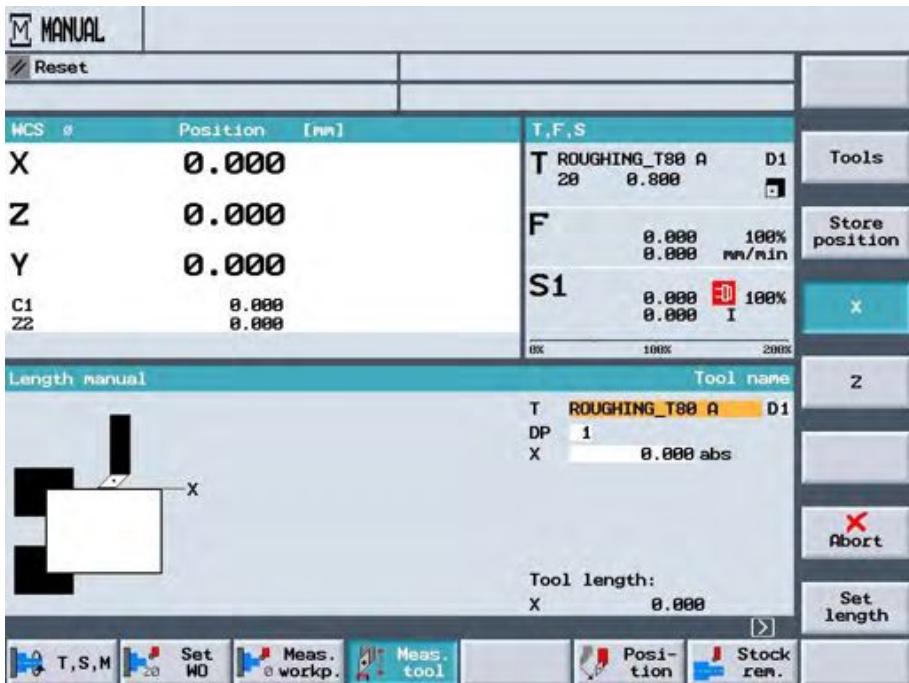
## Section 6

### Measuring of tools

#### 6.2 Measuring tools - Length manual -

Manual

By pressing the VSK 1 "Manual" the following input mask will be shown on the screen.



#### Length manual

Alternat.

Tools

Store position

x

z

Abort

Set length

#### Vertical Softkey-strip (VSK)

By pressing the VSK 1 "Alternat.", activated depending on the selected input field/ parameter, the selection/choice of alternative inputs is possible.

By pressing the VSK 2 "Tools" the tool list will be called up. (See Section 3.2) (See Module A109 - Operating area - Tool management)

By pressing the VSK 3 "Store position" the actual position will be displayed on the screen as "Stored position".  
(See Section 6.2.2)

By pressing the VSK 4 "x" the measured values in the X-axis will be determined.  
(See Section 6.2.3)

By pressing the VSK 5 "z" the measured values in the Z-axis will be determined.  
(See Section 6.2.4)

By pressing the VSK 7 "Abort" there will be no transfer of the entered values.  
Return to the basic display ShopTurn.

By pressing the VSK 8 "Set length" the inputs will be transferred.

Notes

## Section 6

Notes

### Measuring of tools

#### 6.2.1 Measuring of tools - Length manual - X direction



- ① Help picture
- ② Input range for parameters

#### Parameters for the function “Length manual“

Parameter	Help picture	Description and hints
T  T ROUGHING_T80 A D1 DP 1 X 0.000 abs  Tool length: X 0.000		Tool name (Alpha numerical). Alternative: Selection of a tool from the tool list (see Section 3.2)
D  T ROUGHING_T80 A D1 DP 1 X 0.000 abs  Tool length: X 0.000		Cutting edge number (1 to 9)
DP  T ROUGHING_T80 A D1 DP 1 X 0.000 abs  Tool length: X 0.000		Duplo-number (01-99)
X  T ROUGHING_T80 A D1 DP 1 X 0.000 abs  Tool length: X 0.000		Tool distance to work piece.

## Section 6

### Measuring of tools

#### 6.2.2 Measuring of tools - Length manual - Z direction

Notes



- ① Help picture
- ② Input range for parameters

Parameters for the function “Length manual“

Parameter	Help picture	Description and hints
T  T ROUGHING_T80 A D1 DP 1 Reference point Workpiece edge Z 0.000 abs  Tool length: Z 0.000		Tool name (Alpha numerical). Alternative: Selection of a tool from the tool list (see Section 3.2)
D  T ROUGHING_T80 A D1 DP 1 Reference point Workpiece edge Z 0.000 abs  Tool length: Z 0.000		Cutting edge number (1 to 9)
DP  T ROUGHING_T80 A D1 DP 1 Reference point Workpiece edge Z 0.000 abs  Tool length: Z 0.000		Duplo-number (01-99)
Reference point  T ROUGHING_T80 A D1 DP 1 Reference point Workpiece edge Z 0.000 abs  Tool length: Z 0.000		Scratching position  Work piece edge scratching Scratching on the main spindle chuck Scratching on side spindle chuck
Z  T ROUGHING_T80 A D1 DP 1 Reference point Workpiece edge Z 0.000 abs  Tool length: Z 0.000		Tool distance to the work piece.

## Section 6

Notes

### Measuring of tools

Manual tool measurement - Reference point work piece edge:

1. In the mode-of-operation "Manual" select the Softkey "Measure tool".
2. Press the Softkey "Manual".
3. Press the Softkey "X" or "Z", depending on which tool length is to be measured.
4. Press the Softkey "Tools".
5. Select the tool to be measured from the tool list. The cutting edge position and the radius or diameter of the tool must have been entered in the tool list beforehand.
6. Press the Softkey "In manual". The tool is transferred to the mask Measure tool.
7. Select the cutting edge number D and the Duplo-number DP of the tool.
8. Approach and scratch the work piece in the direction in which the measurement is to be carried out.
9. Enter the position of the work piece edge in X0 or Z0. If for X0 or Z0 no value is entered, the value from the actual value display will be used.
10. Press the Softkey "Set length".

The tool length will be evaluated automatically and entered into the tool list. Here the cutting edge position and the tool radius/diameter will be taken into account automatically. If the position of the tool after scratching the work piece is to be stored, press the Softkey "Store position". Following this it is possible to move the axes in order to ease the measurement of the work piece edge X0 manually.

Manual tool measurement - Reference point Chuck:

1. In the mode-of-operation "Manual" select the Softkey "Measure tool".
2. Press the Softkeys "Manual" and "Z".
3. Press the Softkey "Tools".
4. Select the tool to be measured from the tool list. The cutting edge position and the radius or diameter of the tool must have been entered in the tool list beforehand.
5. Press the Softkey "In manual". The tool is transferred to the mask Measure tool.
6. Select the cutting edge number D and the Duplo-number DP of the tool.
7. Approach and scratch the chuck.
8. Press the Softkey "Set length".

The tool length will be evaluated automatically and entered into the tool list. Here the cutting edge position and the tool radius/diameter will be taken into account automatically.

## Section 6

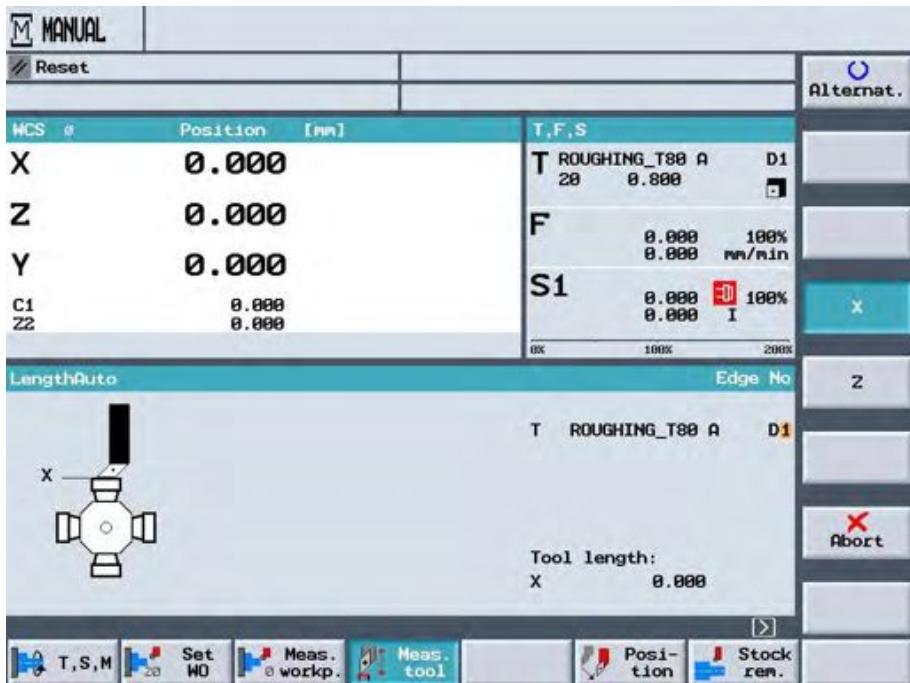
### Measuring of tools

#### 6.3 Measuring of tools - Length Autom. -

Autom.

By pressing the VSK 1 "Autom." the following input mask will be shown on the screen.

Notes



Length Auto

Alternat.

x

z

Abort

Vertical Softkey-strip (VSK)

By pressing the VSK 1 "Alternat.", active only depending on the selected input field/parameter, the selection/choice of alternative inputs is possible.

By pressing the VSK 4 "X" the direction X is pre-selected for the approach of the probe.

By pressing the VSK 5 "Z" the direction Z is pre-selected for the approach of the probe.

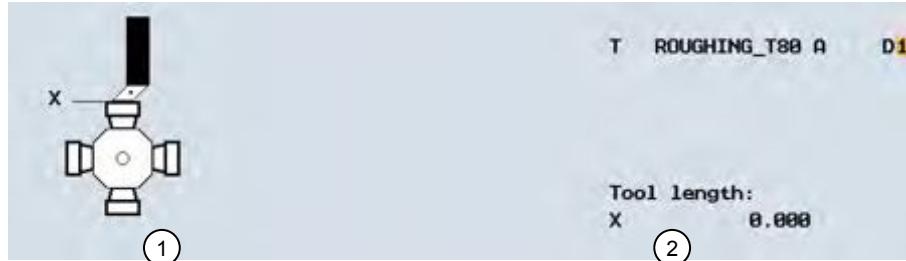
By pressing the VSK 7 "Abort" there will be no transfer of the entered values.  
Return to the basic display Measure tool.

## Section 6

Notes

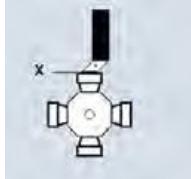
### Measuring of tools

#### 6.3.1 Measuring of tools - Length Autom. -



- ① Help picture
- ② Input range for parameters

Parameters for the function "Length Autom."

Parameter	Help picture	Description and hints
Cutting edge number  T ROUGHING_T8B A D1  Tool length: X 0.000		Cutting edge number (1 to 9)

Tool measurement using a probe:

1. Load the tool to be measured.
2. In the mode-of-operation "Manual" select the Softkey "Measure tool".
3. Press the Softkey "Autom.".
4. Press the Softkey "X" or "Z", depending on which tool length is to be measured.
5. Select the cutting edge number D of the tool.
6. Select, whether the measuring probe of the main or the side spindle is to be used if there are two probes on the machine,
7. Position the tool manually close to the measuring probe, such that the measuring probe can be approached in the selected direction without any danger of collision.
8. Press the key "Cycle Start".

The automatic measuring process commences; i.e. the tool is moved towards the probe stylus at measuring feed rate and retracted again. The tool length will be evaluated and entered into the tool list. The cutting edge position and the tool radius/diameter will be taken into account automatically.

## Section 6

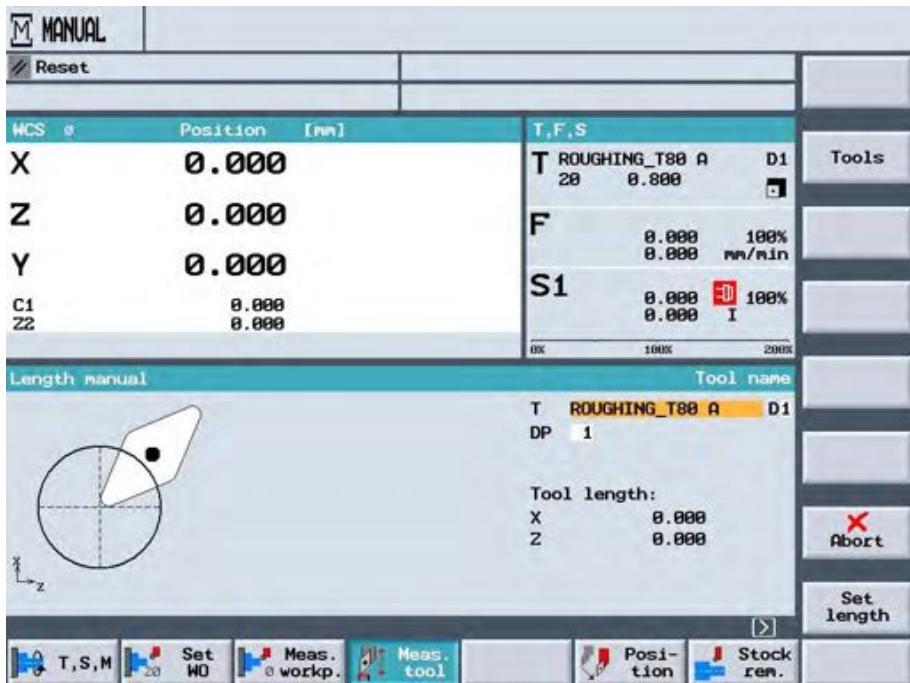
### Measuring of tools

#### 6.4 Measuring of tools - Length Zoom -

Zoom 

By pressing the VSK 3 "Zoom" the following input mask will be shown on the screen.

Notes



#### Length Zoom



#### Vertical Softkey-strip (VSK)

By pressing the VSK 1 "Alternat.", active only depending on the selected input field/parameter, the selection/choice of alternative inputs is possible. (See Section 6.4.1)

Tools

By pressing the VSK 2 "Tools" the tool list is called up. (See Section 6.2.1)

Abort

By pressing the VSK 7 "Abort" there will be no transfer of the entered values.  
Return to the basic display Measure tool.

Set length

By pressing the VSK 8 "Set length" the tool length will be automatically evaluated and entered into the tool list. The tool radius and diameter as well as the cutting edge position will be taken into account.

## Section 6

Notes

### Measuring of tools

#### 6.4.1 Measuring of tools - Length Zoom -



- ① Help picture
- ② Input range for parameters

#### Parameters for the function “Length Zoom“

Parameter	Help picture	Description and hints
T  T ROUGHING_T80 A D1 DP 1  Tool length: X 0.000 Z 0.000		Tool name (Alpha numerical). Alternative: Selection of a tool from the tool list (see Section 3.2)
D1  T ROUGHING_T80 A D1 DP 1  Tool length: X 0.000 Z 0.000		Cutting edge number (1 to 9)
DP  T ROUGHING_T80 A D1 DP 1  Tool length: X 0.000 Z 0.000		Duplo-number (01-99)

## Section 6

### Measuring of tools

Notes

Tool measurement with Zoom:

1. In the mode-of-operation "Manual" select the Softkey "Measure tool".
2. Press the Softkey "Zoom".
3. Press the Softkey "Tools".
4. Select the tool to be measured from the tool list. The cutting edge position and the radius or diameter of the tool must have been entered in the tool list beforehand.
5. Press the Softkey "In manual".
6. Approach the tool to the Zoom optics
7. Align the point of the tool P with the hair line cross of the zoom optics.
8. Press the Softkey "Set length".

The tool lengths are automatically evaluated and entered into the tool list. Here the cutting edge position and the tool radius/diameter will be taken into account automatically.

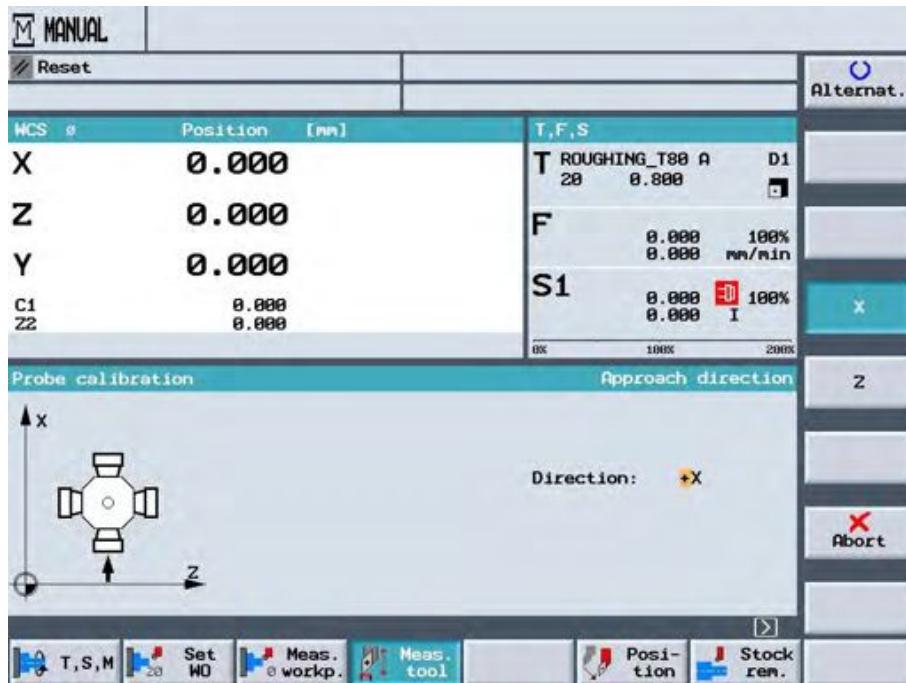
## Section 6

Notes

### Measuring of tools

Calibrate probe

By pressing the VSK 6 "Calibrate probe" the following input mask will be shown on the screen.



If tools are to measured up automatically, the position of the probe on machine table in relation to the machine zero point must be determined. The function "Measuring probe calibration" is available only, if the password for the security stage 1 has been entered. For calibration the measuring probe must be approached from 4 directions (+X, -X, +Z, -Z). For this purpose a tool of the Type Roughing tool or Finishing tool should be used. The cutting edge (position) must always point in the direction in -X and -Z direction. The length and the radius/Diameter of the calibration tool must be entered in the tool list.

Hint:

If there is a second probe for the side spindle, the machine manufacturer must state the details in a machine datum. The documentation by the machine manufacturer must be observed.

**Calibration of probe**

**Vertical Softkey-strip (VSK)**

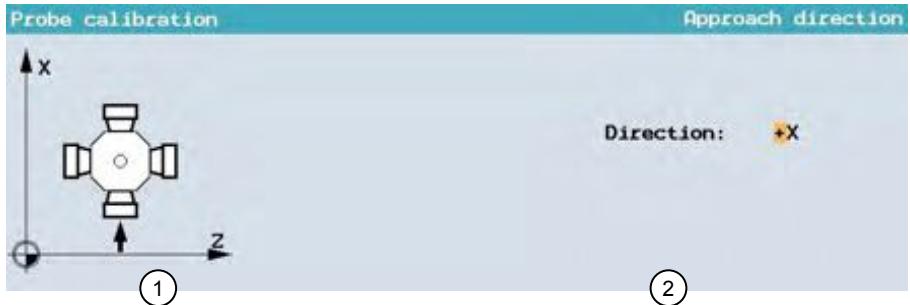
See page 25 - Measure tool  
- Length Autom. -

## Section 6

### Measuring of tools

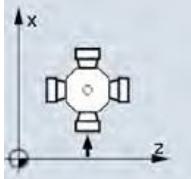
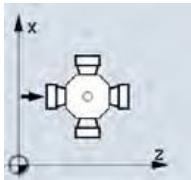
#### 6.5.1 Measuring of tools - Calibration measuring probe -

Notes



- ① Help picture
- ② Input range for parameters

Parameters for the function "Calibrate probe"

Parameter	Help picture	Description and hints
Direction		Approach direction (+X, -X, +Z, -Z)
		

## Section 6

Notes

### Measuring of tools

#### Measuring probe calibration:

1. Load the calibration tool.
2. In the mode-of-operation "Manual" select the Softkey "Measure tool".
3. Press the Softkey "Calibrate measuring probe".
4. Press the Softkey "X" or "Z", depending on which side of the probe is to be calibrated first of all.
5. Select whether the measuring probe for the main or side spindle is to be used if there are two probes on the machine.
6. Select the direction (+ or -) in which the measuring probe is to be approached.
7. Position the calibration tool close to the probe such that the first point of the measuring probe can be approached without danger of collision.
8. Press the key "Cycle Start".

The calibration process commences; i.e. the calibration tool will move automatically toward the probe stylus at the measuring feed rate and retracted again. The position of the measuring probe will be determined and stored in an internal memory. .

9. Repeat the procedure for the other three points of the probe stylus.

## Section 7

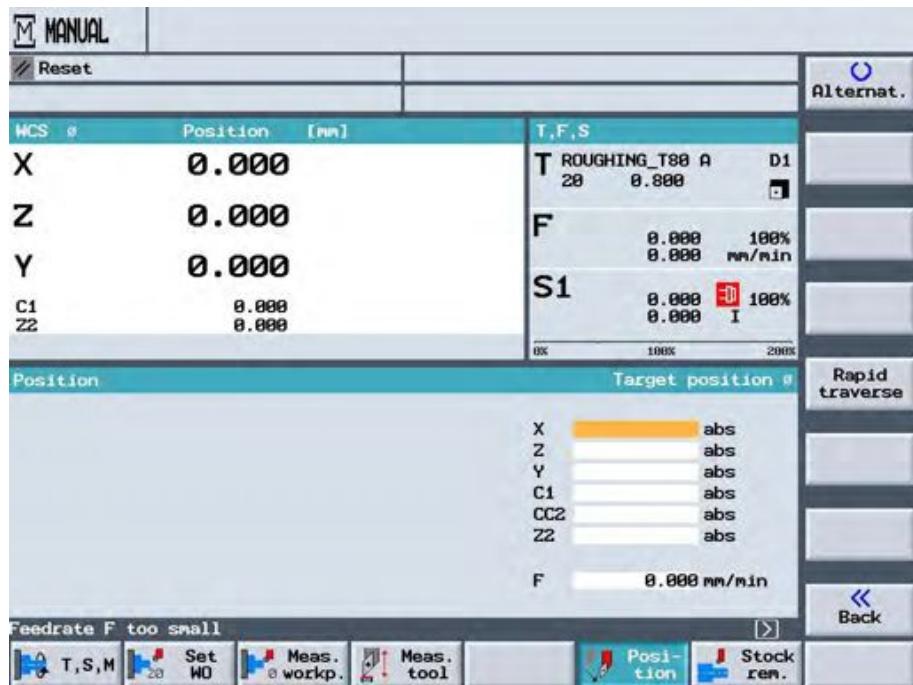
### Position

#### 7.1 Selection and basic display



By pressing the HSK 6 "Position" the following input mask will be shown on the screen.

Notes



In the manual mode the axes can be traversed to defined positions to carry out simple machining operations. The speed and feed overrides are operative for this.

#### Position

#### Vertical Softkey-strip (VSK)

##### Display area

##### Description



By pressing the VSK 1 "Alternat.", active only depending on the selected input field/parameter, the selection/choice of alternative inputs is possible.



By pressing the VSK 5 "Rapid traverse" the rapid traverse value will be transferred as feed rate.



Pressing the VSK 8 "Back" initiates the return to the basic display ShopTurn.

## Section 7

Notes

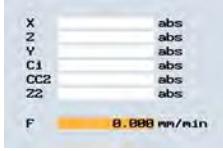
Position

### 7.1.1 Position - Positioning -



- (1) Help picture
- (2) Input range for parameters

Parameters for the function "Position"

Parameter	Help picture	Description and hints
X, Y, Z, C1, CC2, Z2		Destination position of the selected axis [abs, inc] Hint: several destination positions can be entered. B- and C-axis depend on the manufacturer. The documentation of the machine manufacturer must be observed.
F		Feed rate [mm/min, mm/rev] Hint: By pressing the VSK 5 "Rapid traverse" this value will be used.

Positioning the axes:

1. If necessary select a tool.
2. In the mode-of-operation "Manual" select the Softkey "Position".
3. Enter the destination position for the axis/axes.
4. Enter the required value for the feed rate F.
5. Press the Softkey "Rapid traverse".
6. Press the key "Cycle-Start".

The axis will move to the stated destination position. If more than one destination position was entered, the axes will move simultaneously.

## Section 8

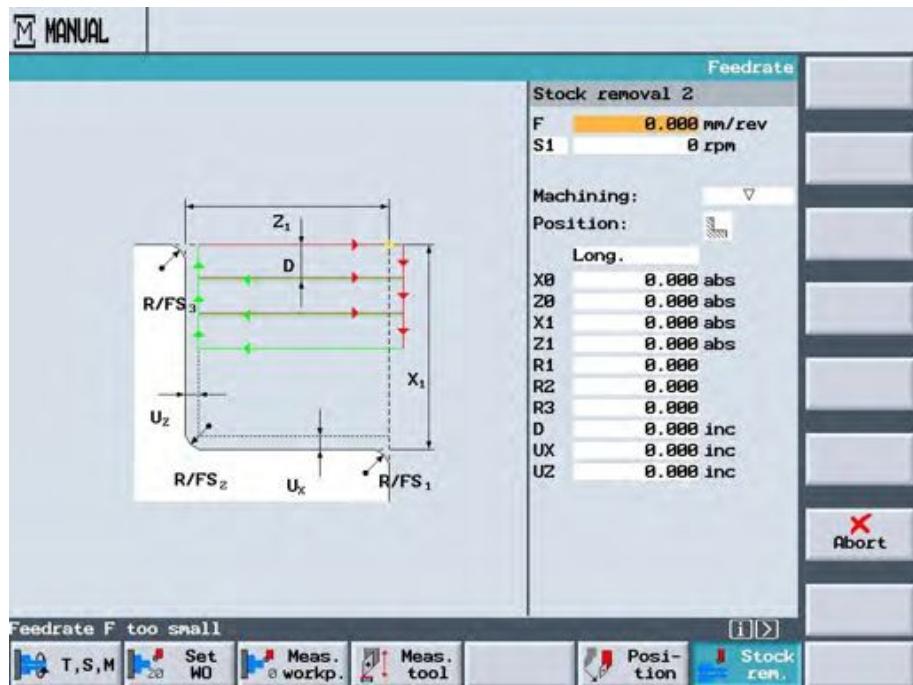
### Stock removal

#### 8.1 Selection and basic display



By pressing the HSK 7 "Stock rem." the following input mask will be shown on the screen.

Notes



#### Roughing

Display area



#### Vertical Softkey-strip (VSK)

Description

By pressing the VSK 1 "Alternat.", active only depending on the selected input field/parameter, the selection/choice of alternative inputs is possible.



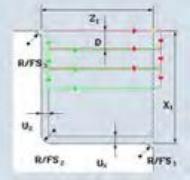
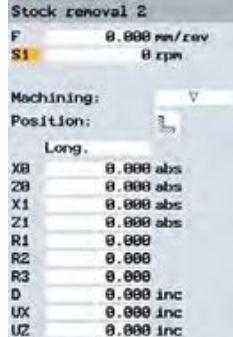
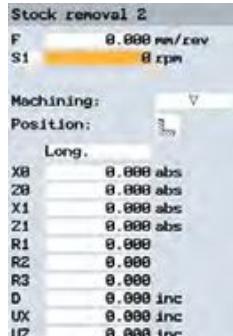
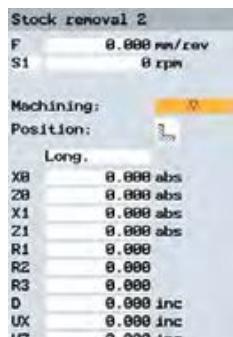
By pressing the VSK 7 "Abort" there will be no transfer of the entered values.  
Return to the basic display Measure tool.

## Section 8

Notes

### Stock removal

#### Parameters for the function Stock removal

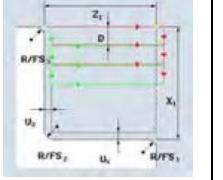
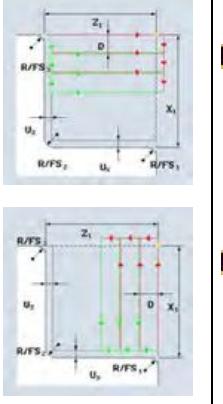
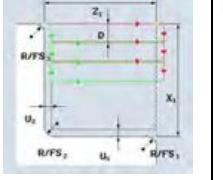
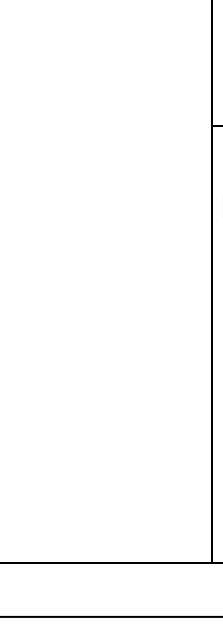
Parameter	Help picture	Description and hints
F		Feed rate [mm/rev]
Spindle		Selection between main spindle, tool spindle or side spindle
Spindle speed		Speed [rev/min] alternatively Cutting speed [mm/min]  Hint: When selecting the cutting speed the display of the spindle selection changes from S1 to V1.
Machining		Stock removal:  Finishing: 

## Section 8

### Stock removal

Parameters for the function Stock removal

Notes

Parameter	Help picture	Description and hints
<b>Position</b>  <b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  <b>Machining:</b> Position: 1 Long. X8 0.000 abs Z8 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 R2 0.000 R3 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		Position for stock removal:    
<b>Dir. of stock rem.</b>  <b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  <b>Machining:</b> Position: 2 Long. X8 0.000 abs Z8 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 R2 0.000 R3 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		Direction of stock removal Long:   Face: 
<b>X0, Z0</b>  <b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  <b>Machining:</b> Position: 3 Long. X8 0.000 abs Z8 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 R2 0.000 R3 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		Reference point [abs]
<b>X1, Z1</b>  <b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  <b>Machining:</b> Position: 4 Long. X8 0.000 abs Z8 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 R2 0.000 R3 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		End point [abs, inc]

## Section 8

Notes

### Stock removal

#### Parameters for the function Stock removal

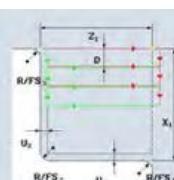
Parameter	Help picture	Description and hints
R1, R2, R3 FS1, FS2, FS3		Radius, alternative to FS
<b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  Machining: Position: Long. X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 abs <b>R1</b> 0.000 R2 0.000 R3 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		Chamfer, alternative to R
<b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  Machining: Position: Long. X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 abs <b>FS1</b> 0.000 <b>FS2</b> 0.000 <b>FS3</b> 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		
<b>XF2</b>  <b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  Machining: Position: Long. X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 <b>XU2</b> 0.000 R3 0.000 D 0.000 inc UX 0.000 inc UZ 0.000 inc		Relief cut, alternative to FS2 or R2
<b>D</b>  <b>Stock removal 2</b> F 0.000 mm/rev S1 0 rpm  Machining: Position: Long. X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 R2 0.000 R3 0.000 <b>D</b> 0.000 inc UX 0.000 inc UZ 0.000 inc		Depth of infeed – only for roughing [inc]

## Section 8

### Stock removal

Parameters for the function Stock removal

Notes

Parameter	Help picture	Description and hints
<p>UX / ZX</p> <p><b>Stock removal 2</b></p> <p>F 0.000 mm/zev S1 0 rpm</p> <p>Machining:</p> <p>Position: </p> <p>Long.</p> <p>X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 abs R1 0.000 R2 0.000 R3 0.000 D 0.000 inc <b>UX 0.000 inc</b> <b>UZ 0.000 inc</b></p>		<p>Finishing allowance in X or. Z direction– only for roughing</p> <p>[ink]</p>

## Section 8

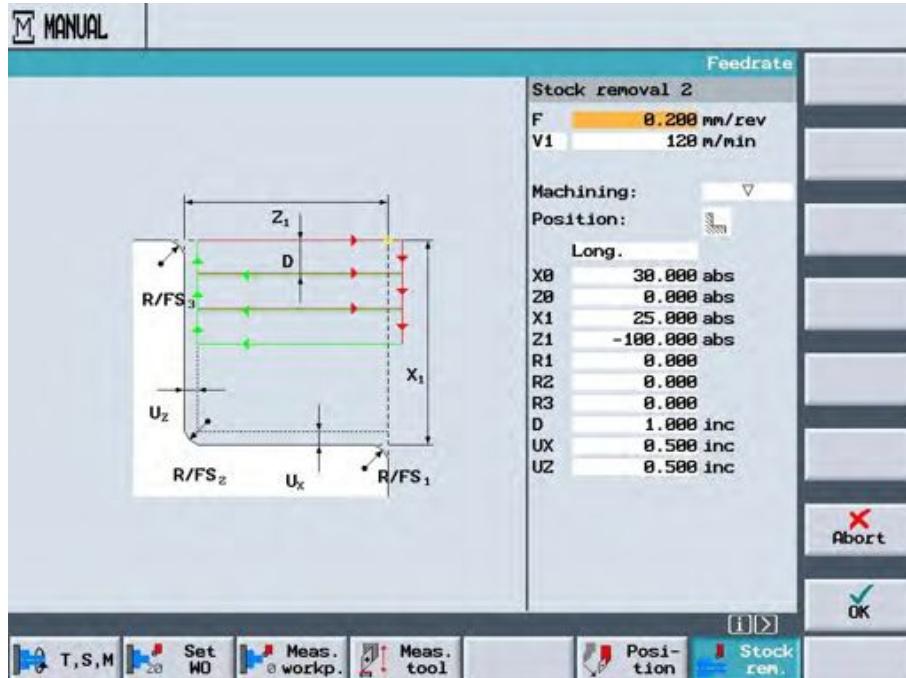
Notes

Stock removal

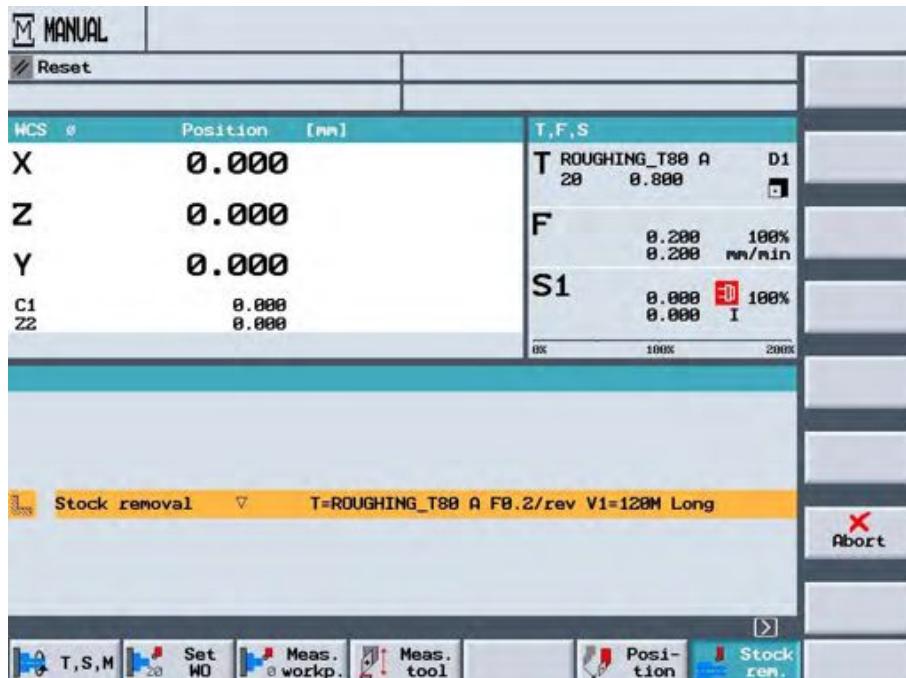
### 8.2 Example



By pressing the HSK 7 "Stock rem." the following input mask will be shown on the screen.



By pressing the VSK 8 "OK" the roughing will be transferred to the machining.



By pressing the key "Spindle Start" start the spindle and by pressing the key „Feed Start“ start the feed.



Start the machining by pressing the key "Cycle Start".

## Section 9

### ShopTurn settings

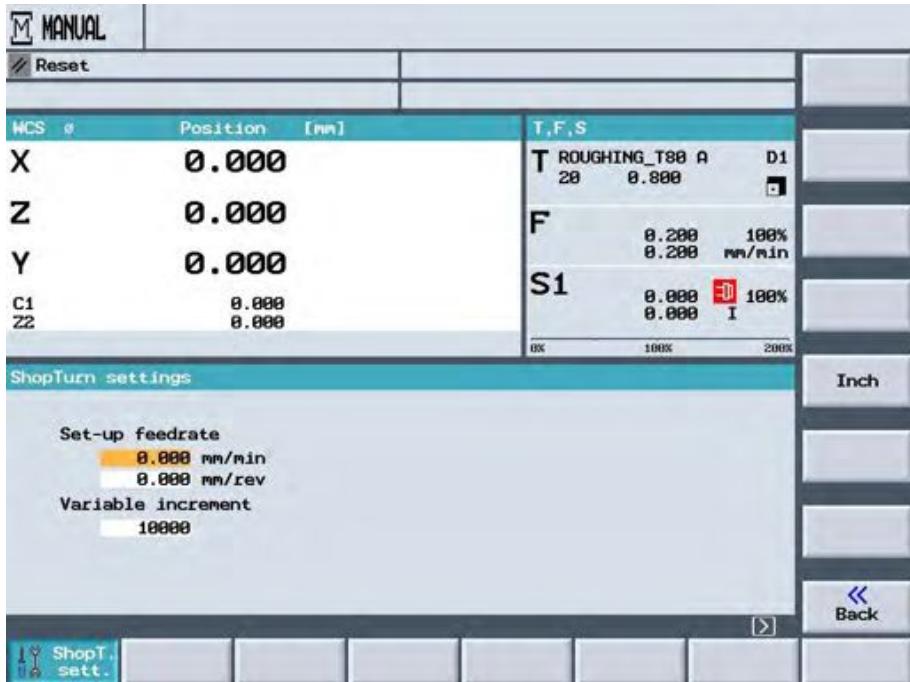
#### 9.1 Selection and basic display



ShopT.  
sett.

By pressing the key "Extension" on the OP followed by pressing the HSK 1 "ShopTurn Settings" the following input mask will be displayed.

Notes



#### ShopTurn settings

#### Vertical Softkey-strip (VSK)

Display area

Inch

Description  
By pressing the VSK 5 "Inch" the measuring unit system can be switched over (See Section 9.1.2)

« Back

Description  
Pressing the VSK 8 "Back" initiates the return to the basic display ShopTurn settings.

## Section 9

Notes

### ShopTurn settings

#### 9.2 ShopTurn settings



##### ① Input range for parameters

#### Parameters for the function “ShopTurn Settings“

Parameter	Help picture	Description and hints
Jogging feed rate	<p>The screenshot shows a software interface titled "ShopTurn settings". Under the "Set-up feedrate" section, the values are displayed as 0.000 mm/min and 0.000 mm/rev. Under the "Variable increment" section, the value is 10000.</p>	Value input for the pre-set jogging feed rate [mm/min]
Jogging feed rate	<p>The screenshot shows a software interface titled "ShopTurn settings". Under the "Set-up feedrate" section, the values are displayed as 0.000 mm/min and 0.000 mm/rev. Under the "Variable increment" section, the value is 10000.</p>	Value input for the pre-set jogging feed rate [mm/rev] Hint: If the jogging feed rate set by the machine manufacturer is less than the selected value, this will be used. The documentation by the machine manufacturer must be observed.
Variable incremental feed	<p>The screenshot shows a software interface titled "ShopTurn settings". Under the "Set-up feedrate" section, the values are displayed as 0.000 mm/min and 0.000 mm/rev. Under the "Variable increment" section, the value is 10000.</p>	Value input for the pre-selected incremental feed

## Section 9

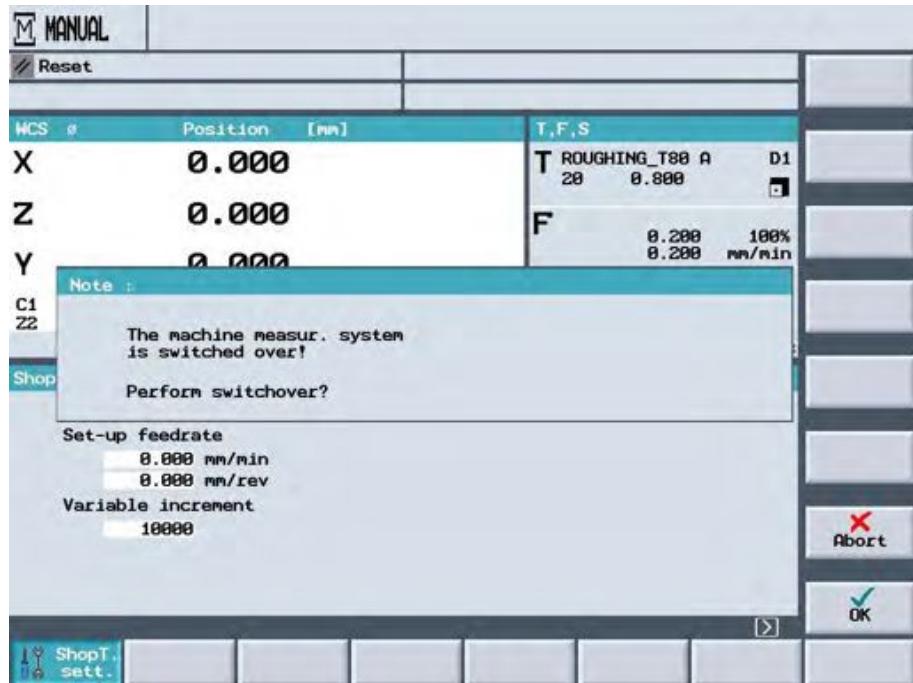
### ShopTurn settings

#### 9.3 Switching the measuring system

Inch

By pressing the VSK 5 "Inch" the following information mask will be displayed on the screen.

Notes



#### ShopTurn Settings

#### Vertical Softkey-strip (HSK)

Display area

Description



Pressing the VSK 7 "Abort" the pre-selection Measuring System Switching will be stopped. Return to the basic display ShopTurn settings.



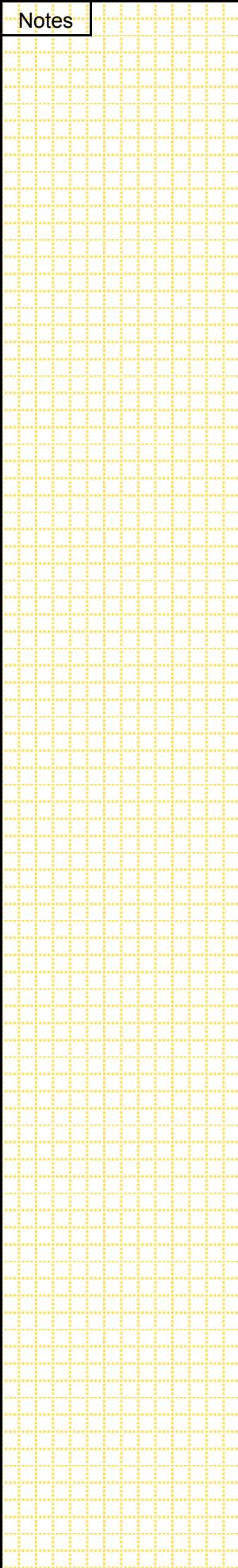
By pressing the VSK 8 "OK" the switching of the measuring system will be performed.

Hint:

Axis positions and path parameters can be shown in the manual mode either in "mm" or "Inch".

Tool corrections, Work offsets, axis positions and ShopTurn-settings will be converted to the respectively other measuring unit.

Notes



## 1 Brief description

**Objective of the module:**

This module explains the operation and programming of the manual machine under ShopTurn.

**Description of the module:**

The individual sections of the screen display will be described here

The Softkeys (setting-up and basic functions) available under ShopTurn for the manual machine will be described.

**Content of the module:**

Basic display ShopTurn

- Horizontal Softkey-strip
- Vertical Softkey-strip
- Parameter basic display Manual machine

Tool measurement

*See Module A130 - "Operating area Program - Various"*

Set WO

*See Module A103 - "Manual Operating area"*

Straight

*See Module A113 - "Operating area Program - Straight/Circle"*

Drilling

*See Module A115 - "Operating area Program - Drilling"*

Turning

*See Module A117 - "Operating area Program - Turning"*

Milling

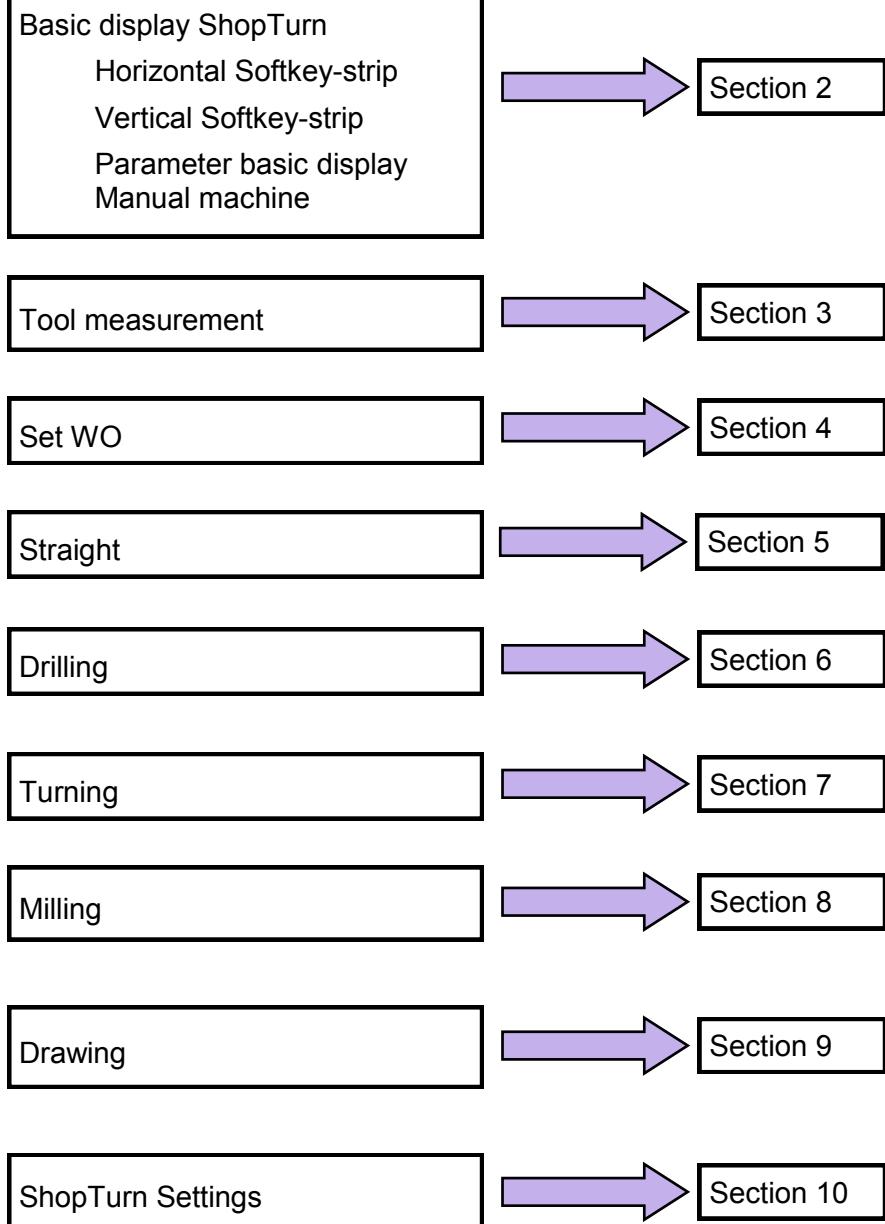
*See Module A121 - "Operating area Program - Milling"*

Drawing

ShopTurn Settings

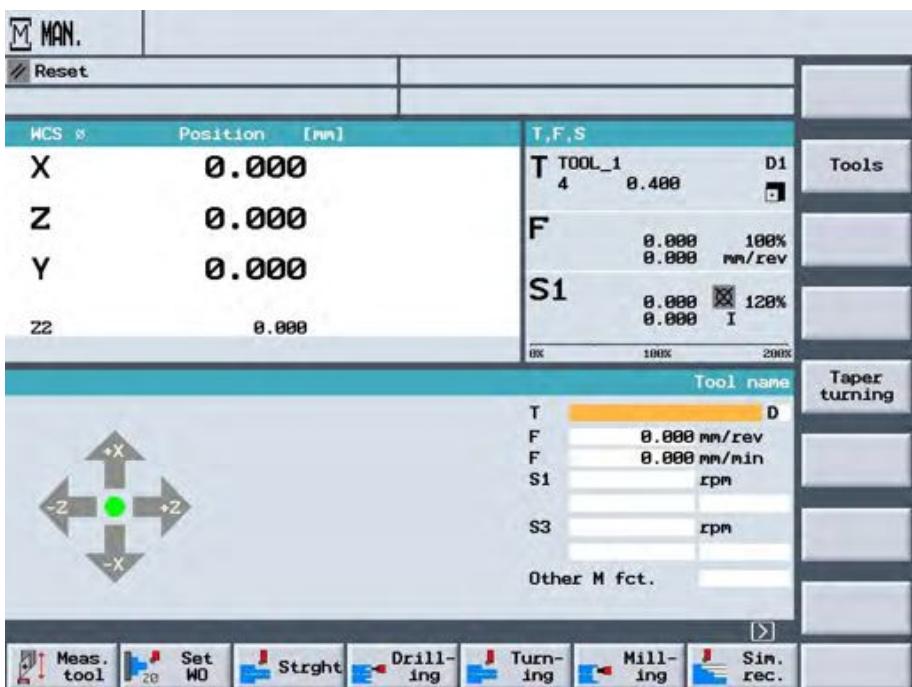
*See Module A103 - "Manual Operating area"*

## Operating Area Machine - Manual machine



## Section 2

### Basic display Manual machine



Notes

#### 2.1 Horizontal Softkey-strip



By pressing the HSK 1 "Meas. tool" the input mask for the section "Tool measurement" will be called up.

See Section 3 "Tool measurement".



By pressing the HSK 2 "Set WO" the input mask for the programming a Work offset will be called up.

See Section 4 "Set WO".



By pressing the HSK 3 "Straight" the input mask for the programming of a linear positioning will be called up.

See Section 5 "Straight".



By pressing the HSK 4 "Drilling" the input mask for the section "Drilling" will be called up.

See Section 6 "Drilling".



By pressing the HSK 5 "Turning" the input mask for the section "Turning" will be called up.

See Section 7 "Turning".

## Section 2

### Basic display Manual machine

Notes



By pressing the HSK 6 "Milling" the input mask for the section "Milling" will be called up.

*See Section 8 "Milling".*



By pressing the HSK 7 "Drawing" the simulation will be called up.

*See Section 9 "Drawing".*

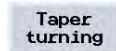


By pressing the HSK 1 "ShopTurn Settings" Settings can be entered in ShopTurn.

*See Section 10 "ShopTurn Settings".*

### 2.2 Vertical Softkey-strip

Tools



By pressing the VSK 2 "Tools" the Tool management will be called up.

Taper turning

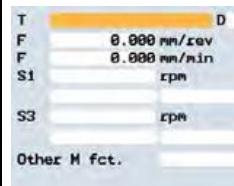
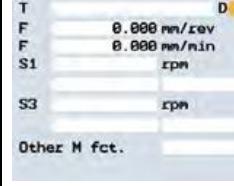
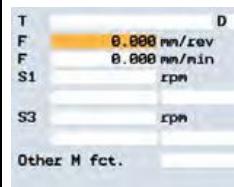
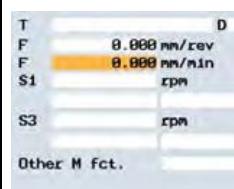
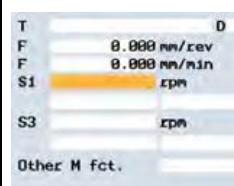
By pressing the VSK 5 "Taper turning" the e section "Taper turning" will be called up.

## Section 2

### Basic display Manual machine

#### 2.3 Parameter Basic display Manual machine

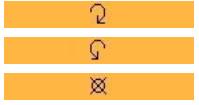
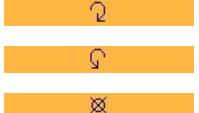
Notes

Parameter	Help picture	Description and Hints
T		Tool name
D		Cutting edge number
F		Feed with rotating spindle [mm/rev] Hint: Option [mm/rev, mm/min]
F		Feed with stationary spindle [mm/min] Hint: Option - if this input field is missing, it is possible to toggle between mm/rev and mm/min.
S1 / V1		Speed / Cutting speed for the main spindle [rev/min, m/min]

## Section 2

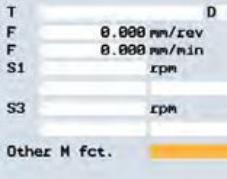
### Basic display Manual machine

Notes

Parameter	Help picture	Description and Hints
Direction of rotation		<p>Direction of rotation</p> <p>Alternatively:</p>  <p>Hint: Option - if this input field is missing, appropriate keys will be found on the operating panel. The documentation/hints by the machine manufacturer must be observed.</p>
Gear stage		<p>Gear stage</p> <p>Alternatively:</p>  <p>Hint: This field is shown only if gear stages are available.</p>
S3		<p>Tool spindle speed</p> <p>Hint: Option - The documentation/hints by the machine manufacturer must be observed.</p>
Direction of rotation		<p>Direction of rotation of the tool spindle</p>  <p>Hint: Option - The documentation/hints by the machine manufacturer must be observed</p>
Gear stage		<p>Gear stage for the tool spindle</p> <p>Alternatively:</p>  <p>Hint: Option - The documentation/hints by the machine manufacturer must be observed</p> <p>This field is shown only if gear stages are available.</p>

## Section 2

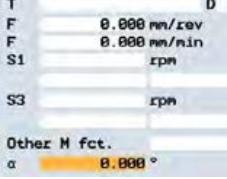
### Basic display Manual machine

Parameter	Help picture	Description and Hints
Other M-Fct.		Other M-Functions Example: M08 Coolant ON

Notes

Taper turning

By pressing the VSK 5 "Taper turning" the following parameter display appears.

Parameter	Help picture	Description and Hints
$\alpha$		Taper angle

## Section 3

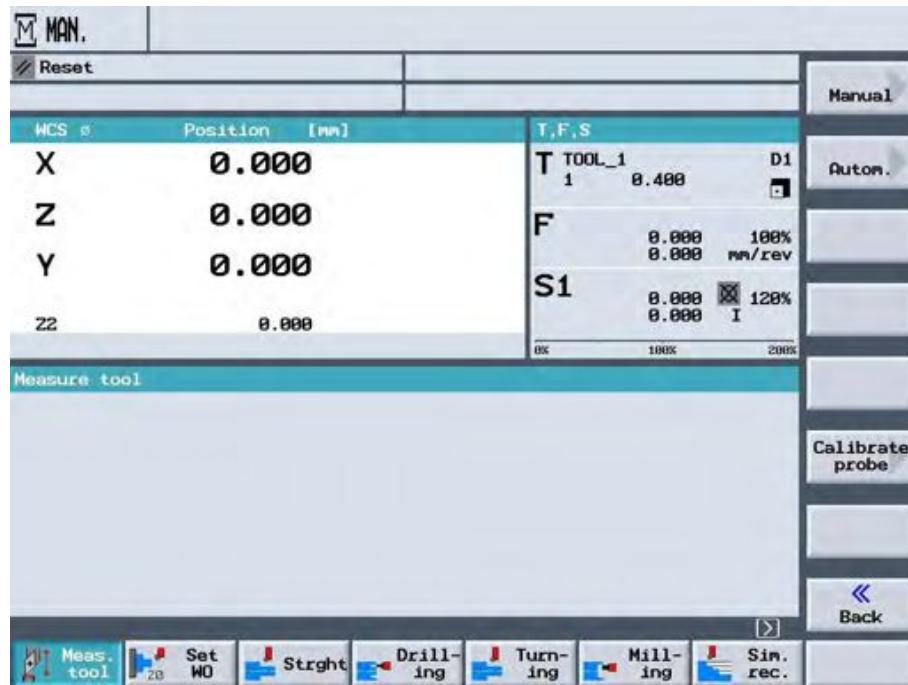
Notes

Tool measurement call-up



By pressing the VSK 1 "Measure tool" the input mask for the section "Tool measurement" will be called up.

See Module A130 - "Operating area Program - Various".



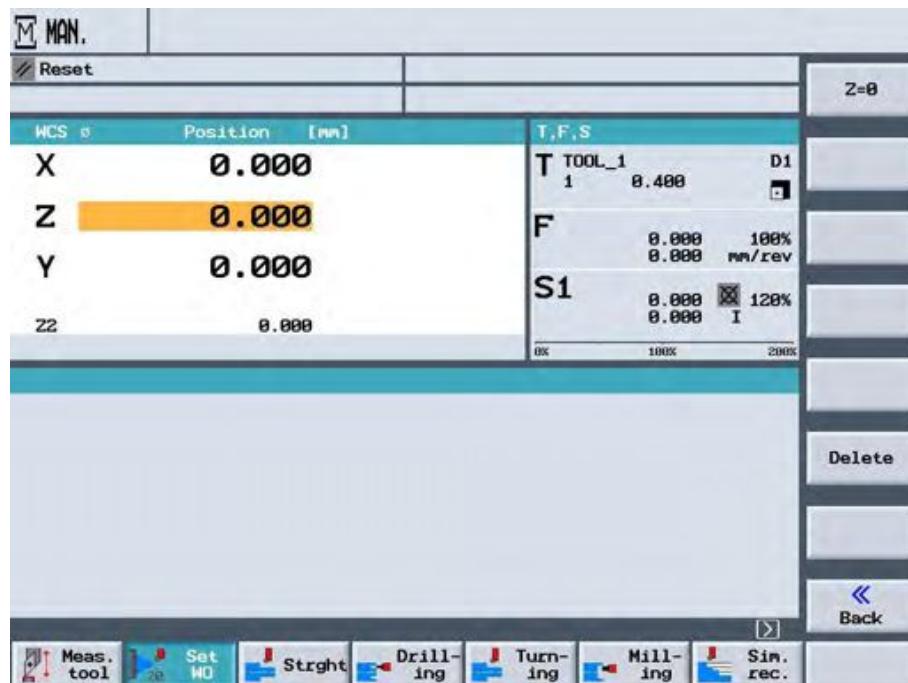
## Section 4

Set WO



By pressing the VSK 2 "Set WO" the input mask for the programming of a Work offset will be called up.

See Module A103 - "Manual operating area".



## Section 5

Straight



By pressing the VSK 3 "Straight" the input mask for the programming of a linear positioning will be called up.

See Module A113 - "Operating area Program - Straight/Circle".

Notes

The input mask for "Straight" programming shows the following parameters:

WCS	Position [mm]	T, F, S
X	0.000	T TOOL_1 1 0.400 D1
Z	0.000	F 0.000 100% 0.000 mm/min
Y	0.000	S1 0.000 120% 0.000 I
Z2	0.000	OK 100K 200K

Below the parameters, there is a diagram illustrating a linear movement from an absolute position ( $X_{abs}$ ,  $Z_{abs}$ ) to an incremental position ( $X_{inc}$ ,  $Z_{inc}$ ). The diagram shows a green vector representing the movement along the X-axis.

Tool tip: Feedrate F too small

Tool bar buttons: Meas. tool, Set WO, Straight, Drilling, Turning, Milling, Sim. rec.

## Section 6

Drilling



By pressing the VSK 4 "Drilling" the input mask for the section "Drilling" will be called up.

See Module A115 - "Operating area Program - Drilling".

The input mask for "Drilling" programming shows the following parameters:

WCS	Position [mm]	T, F, S
X	0.000	T TOOL_1 1 0.400 D1
Z	0.000	F 0.000 100% 0.000 mm/min
Y	0.000	S1 0.000 120% 0.000 I
Z2	0.000	OK 100K 200K

On the right side, a vertical stack of buttons lists drilling modes:

- Drilling centric
- Thread centric
- Drilling Reaming
- Deep hole drilling
- Tapping thread

Tool tip: Feedrate F too small

Tool bar buttons: Meas. tool, Set WO, Straight, Drilling, Turning, Milling, Sim. rec.

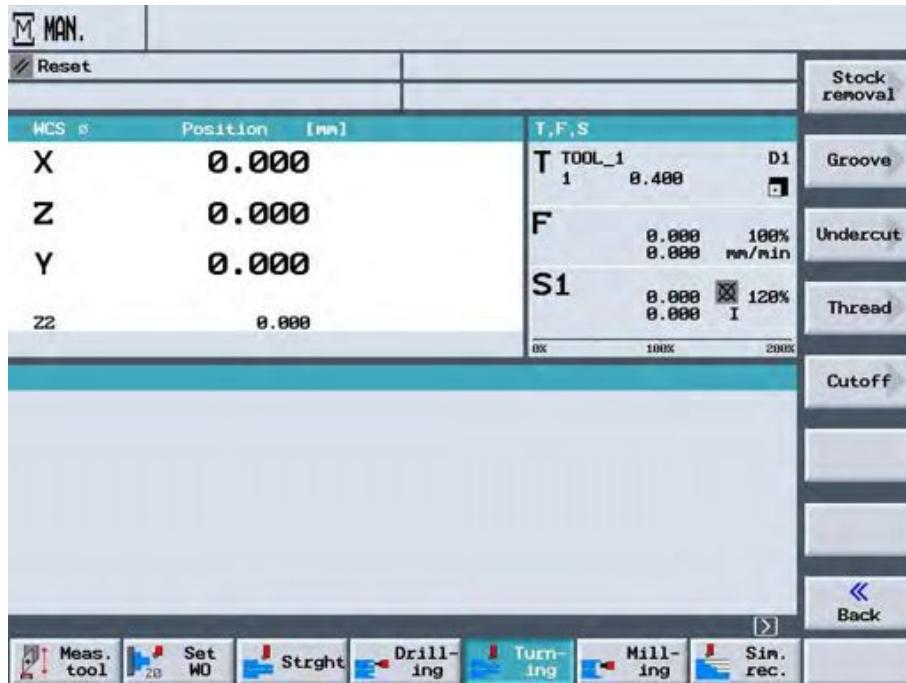
## Section 7

### Turning

Notes



By pressing the VSK 5 "Turning" the input mask for the section "Turning" will be called up.  
See Module A117 - "Operating area Program - Turning".



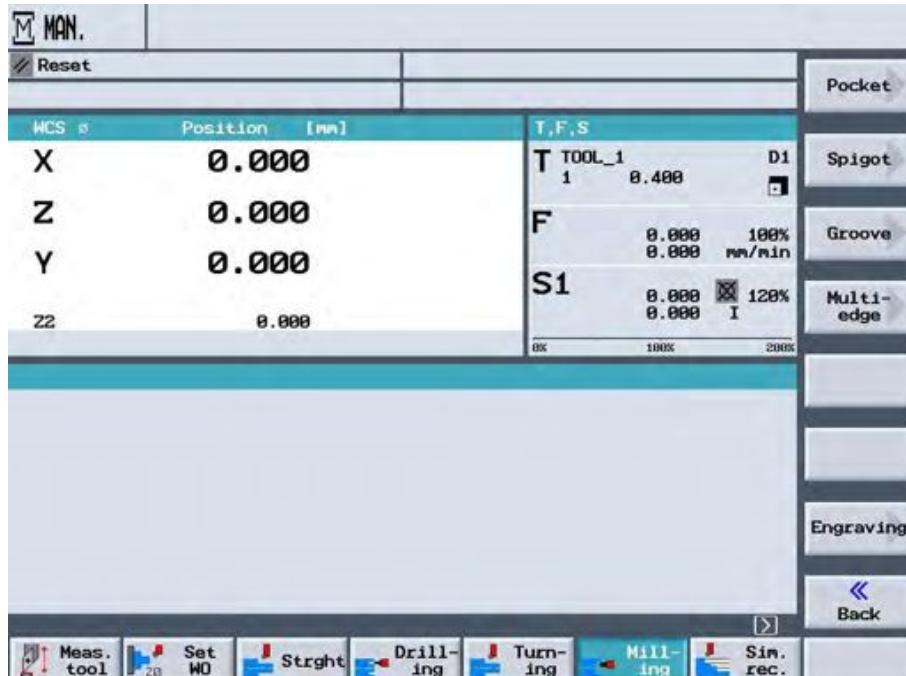
## Section 8

### Milling



By pressing the VSK 6 "Milling" the input mask for the section Milling will be called up.

See Module A121 - "Operating area Program - Milling".



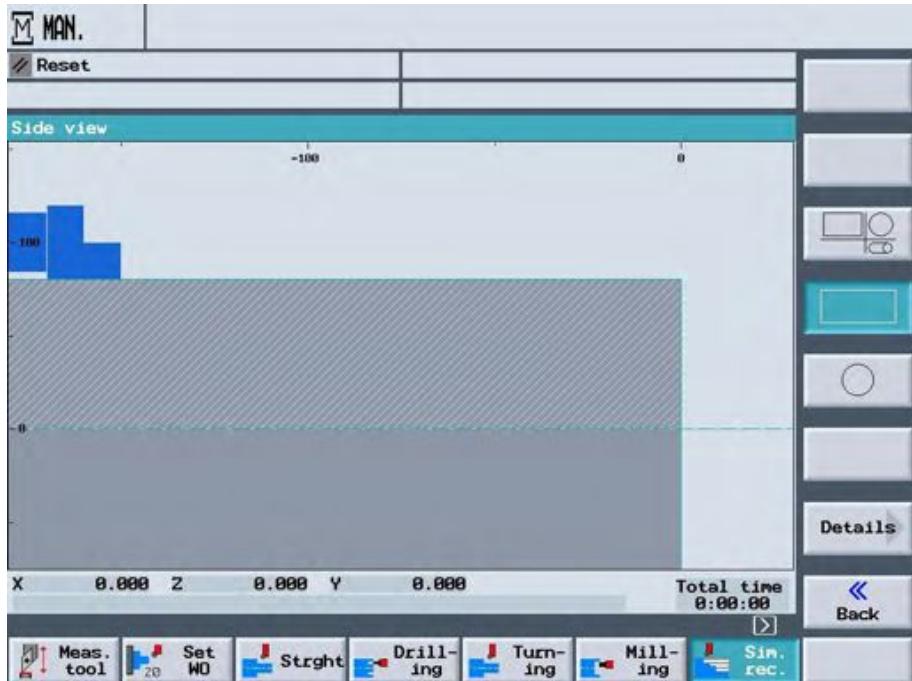
## Section 9

### Drawing



By pressing the HSK 7 "Sim. rec." the simulation will be activated.

Notes

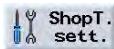


## Section 10

### ShopTurn Settings



By pressing the key "Extension" on the OP the selection of further Softkeys is possible.

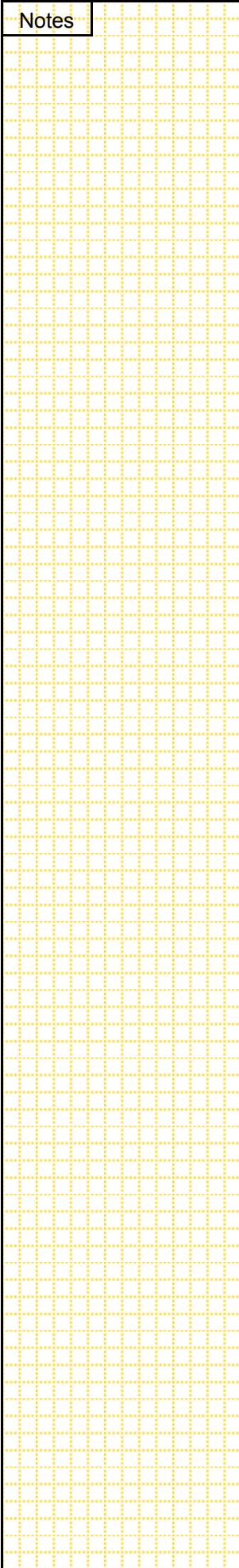


By pressing the HSK 1 "ShopT. Sett." it is possible to enter Settings in ShopTurn.

See Module A103 - "Manual operating area".



Notes



## 1 Brief description

**Objective of the module:**

This module enables you to handle programs and files under ShopTurn.

**Description of the module:**

All NC-Programs, which are created with the ShopTurn Editor, are stored in the NC-work memory. These programs can be accessed via the program manager for:

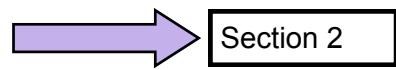
execution,  
alteration,  
copying,  
re-naming or  
deletion

**Content of the module:**

- Call-up and layout of the program manager
- Management of NC-programs
  - Opening a directory and an NC-program
  - Starting a new NC-program
  - Re-naming an NC-program
  - Marking an NC-program
  - Copy and insertion of an NC-program
  - Cutting / clearing an NC-program
  - Executing an NC-program
- Loading / unloading an NC-program
- Saving / re-loading of tooling data

## Operating Area - Program manager

Call-up and layout of the program manager

**Section 2**

Management of NC-programs  
Opening a directory and an NC-program  
Starting a new NC-program  
Re-naming an NC-program  
Marking an NC-program  
Copy and insertion of an NC-program  
Cutting / clearing an NC-program  
Executing an NC-program

**Section 3**

Loading / unloading an NC-program

**Section 4**

Saving / re-loading of tooling data

**Section 5**

## Section 2

### Call-up and layout of the program manager

The program manager can be called up as follows:



Press the Hotkey "Program Manager" on the CNC-keyboard or on the operator-panel (OP).

or



On the OP press the key "Menu Select".

Notes

The following ShopTurn Softkey strip appears.



Selection of the program manager.

The program manager with an overview of the directories opens:



- ① Header line of the program manager
- ② Working window „Directory and program overview“
- ③ Memory status display (the display of Hard disk: only for PCU 50)
- ④ Horizontal Softkey strip with indication of the available storage media, e.g. NC, USB-Stick, Network drive. The Softkeys "Part progs." and "Sub-progs." can be optionally enabled. The documentation by the machine manufacturer must be observed.
- ⑤ Vertical Softkey strip for management of the directories and NC-programs.

## Section 2

Notes

Call-up and layout of the program manager

- ① The top line of the program manager is subdivided as follows:

**Name** of the files or programs

The name may consist of up to 24 characters.

**Type** (Symbols)



Directory: WPD - work piece Directory



Program: MPF - Main program file  
SPF - Sub-program file



Zero offset/Tool data:INI - Initial file

Jump back to the main directory



Select the uppermost line of the program manager and return to the main directory by pressing the key "INPUT".

**Loaded (only available for PCU 50)**

(X) The program is loaded into the NC-memory

( ) The program is available on the hard disk or on an external drive.

See Section "NC-programs load / unload"

**Size**

"NCK-Dir." is shown in the directory overview. This refers to a directory.

The size of the NC-program is stated in the selected directory.

**Date/Time** (of first saving or last alteration)

- ② Working window „Directory and list of programs“

If there are more than one files / programs than can be shown on one page, the cursor keys can be used for paging or to scroll up and down.

- ③ Memory status display

The free space available on the storage medium or the NC-memory is shown. The display for the capacity on the hard disk is only available in case of a PCU 50. Further to this see the documentation by the machine manufacturer.

## Section 2

### Call-up and layout of the program manager

#### ④ Horizontal Softkey strip - HSK

All the available storage media are shown..

The documentation by the machine manufacturer must be observed.

##### Display



##### Description

By pressing the HSK 1 "NC" all files / NC-programs on the NC / hard disk are shown.



By pressing the HSK 2 "Part progs." the respective directory on the NC is called up.



By pressing the HSK 3 "Sub-progs." the respective directory on the NC is called up.



By pressing the HSK 4 "G:" the programs / directories on the USB-Stick are shown. The drive letter G: shown on the Softkey may also be replaced by a text, e.g. USB-Stick or PCMCIA Card (only in case of a PCU 20). Here the documentation by the machine manufacturer must be observed.



By pressing the HSK 5 "NET\_DRIVE" the programs / files on a released network drive will be shown.

#### Notes

##### Hint:

The display and layout of the horizontal Softkeys can vary. Please refer to the documentation by the machine manufacturer.

## Section 2

Notes

Call-up and layout of the program manager

Alternative operation using the Cursor keys:



Selection of a program in the program-  
Manager



Selection of a program in the program direc-  
tory

Selection of a block in the program



Open the selected program directory

Opens the selected program in the editor  
(The operation sector changes from Directory  
to Program)

Opens the selected block



Closes the selected directory

Closes the selected block

(Alterations in the program block are stored.  
If the alterations are not to be saved, the in-  
put mask must be closed by means of the  
VSK 7 "Abort".

Changing the mode-of-operation from Program editor to Directory by  
means of the Cursor key is not possible.

## Section 2

### Call-up and layout of the program manager

#### ⑤ Vertical Softkey strip - VSK

The following functions are available on the vertical Softkey strip for the management of the NC-programs. There may be further sub-functions depending on the function that has been selected. The sub-functions are explained in the description of the respective function.

Display

 Execute

Description

By pressing the VSK 1 "Execute" the selected NC-program can be called up for the machining. This Softkey is shown only if the cursor is located on an NC-program. (See Section 3.7)

 New

By pressing the VSK 2 "New" a new file can be added to the directory.

A new ShopTurn- or a G-Code-Program can be produced in the selected directory. (See Section 3.2)

 Rename

By pressing the VSK 3 "Rename" the selected program or directory can be renamed. (See Section 3.3)

 Mark

By pressing the VSK 4 "Mark" several programs or files can be marked, for instance for copying or cutting out. (See Section 3.4)

 Copy

By pressing the VSK 5 "Copy" one or more program(s) or file(s) can be copied. (See Section 3.5)

 Paste

By pressing the VSK 6 "Paste" one or more program(s) or file(s) can be inserted into another directory or the selected storage medium. (See Section 3.5)

 Cut

By pressing the VSK 7 "Cut" one or more program(s) or file(s) can be cut out or deleted . (See Section 3.6)

 Continue

By pressing the VSK 8 "Continue" functions for the data exchange between NC and hard disk as well as for special functions. (see next page)

Notes

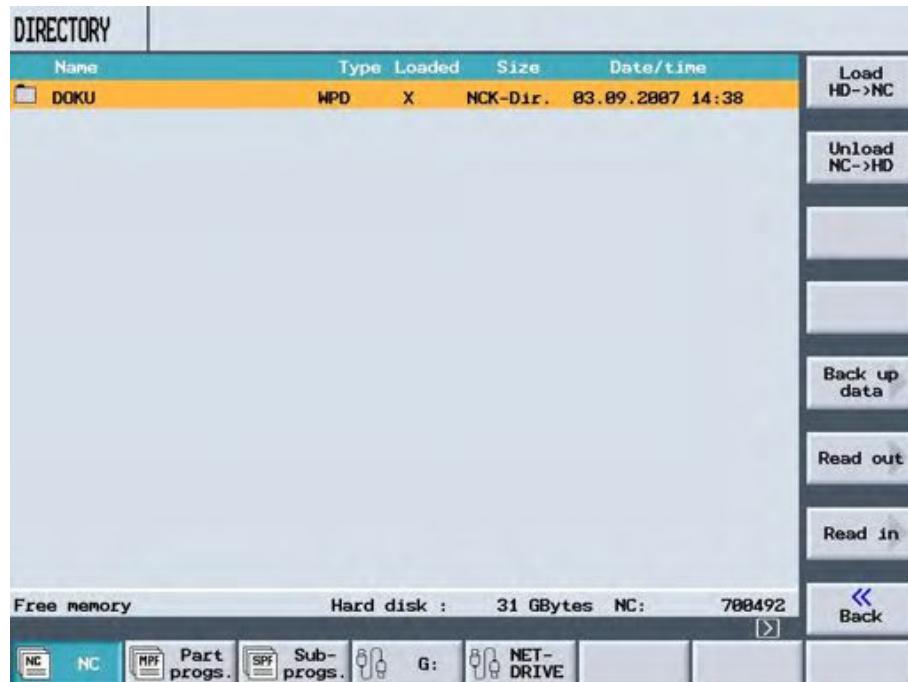
## Section 2

### Call-up and layout of the program manager

Notes

Continue

After the VSK 8 "Continue" has been actuated, the vertical Softkey strip changes and the following functions are now available:



Load  
HD->NC

By pressing the VSK 1 "Load HD→NC" the selected NC-program / directory is transferred from the hard disk into the memory of the NC. (See Section 4) (only for PCU 50; for the PCU 20 this possibility is available only in case of an external storage medium)

Unload  
NC->HD

By pressing the VSK 2 "Unload NC→HD" the selected NC-program / directory will be unloaded from the NC memory back to the hard disk. (See Section 4) (only for PCU 50; for the PCU 20 this possibility is available only in case of an external storage medium)

Execution  
hard disk

By pressing the VSK 3 "Execution hard disk" part of an NC-program will be loaded from the hard disk into the NC-memory and the display changes to the mode-of-operation "AUTO".

This Softkey is shown only if the selected program is not already loaded into the NC-memory.

By pressing the VSK 5 "Back up data" the tool data and zero-points in the directory will be saved. (See Section 5)

By pressing the VSK 6 "Read out" or VSK 7 "Read in" files and NC-programs can be read out 7 read-in via the V24-interface.

This function is not dealt with in this documentation since it is available only with a PCU 20 or PCU 50.2. In this case refer to the documentation by the machine manufacturer.

By pressing the VSK 8 "Back" the previous vertical Softkey strip will be shown.

<<  
Back

## Section 3

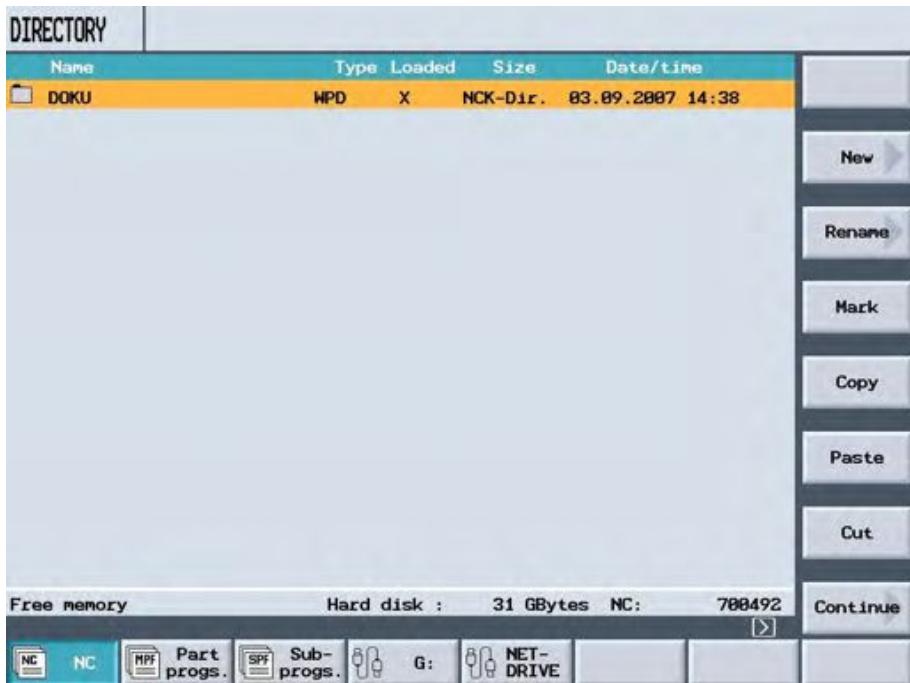
### Management of NC-programs

#### 3.1 Opening of directory and NC-program

##### 3.1.1 Opening a directory

Open the program manager as described in Section 2.

Notes



Select the storage medium to be opened on the horizontal Softkey strip, e.g. "NC".

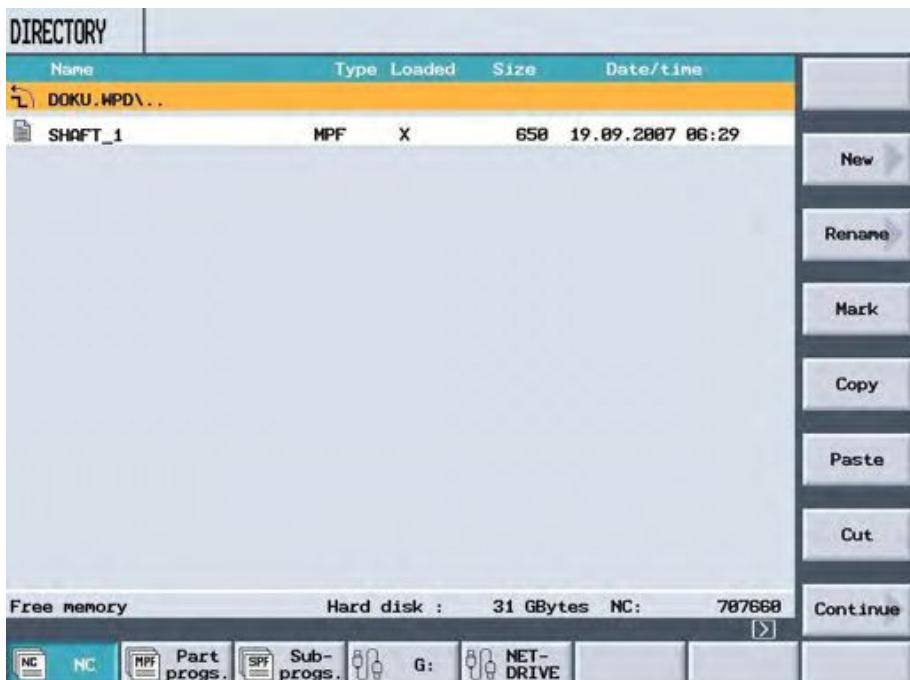


With the key "Cursor down" select the directory to be opened, e.g. DOKU.WPD.



or

With the key "INPUT" or "Cursor right" open the directory.



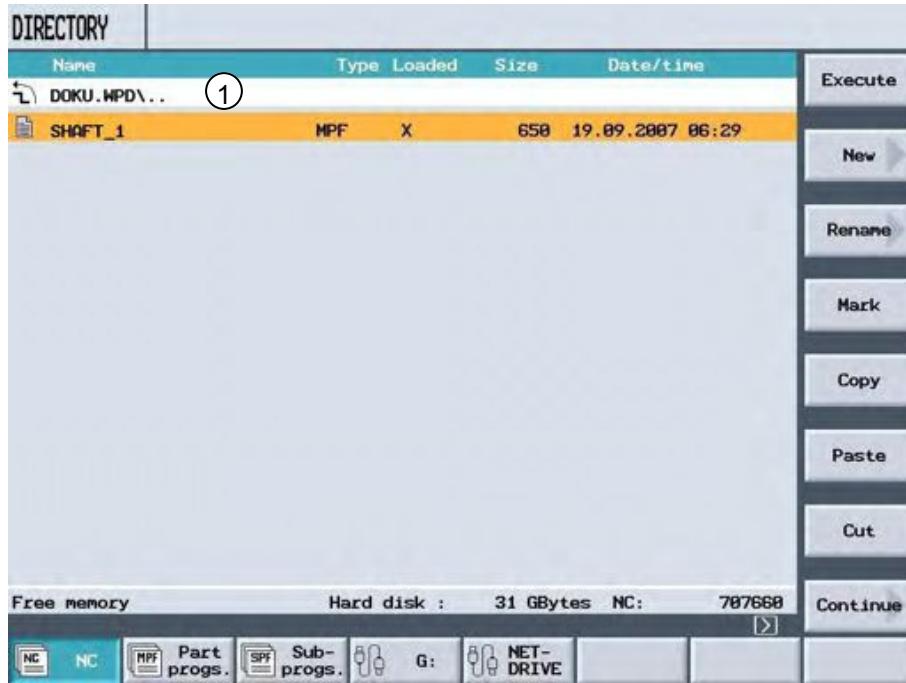
## Section 3

Notes

### Management of NC-programs

#### 3.1.2 Closing the directory

A respective directory was selected as described in Section 3.1.1.



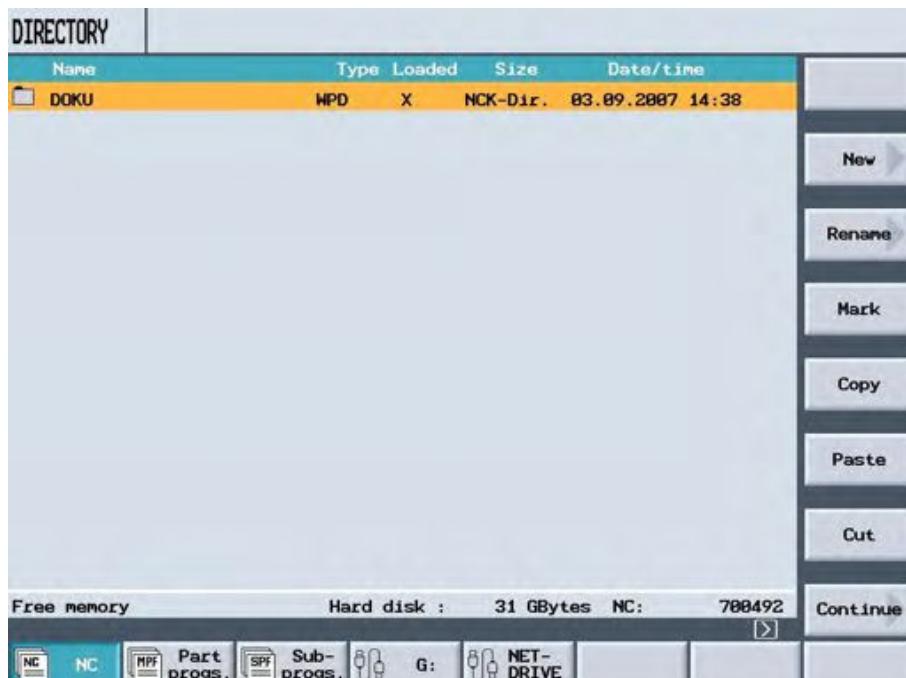
If the cursor is located on any one line, the key "Cursor left" can be pressed

or



select the topmost line with the cursor (return jump line ①) and press the key "INPUT" or "Cursor left".

The directory overview is shown once more.



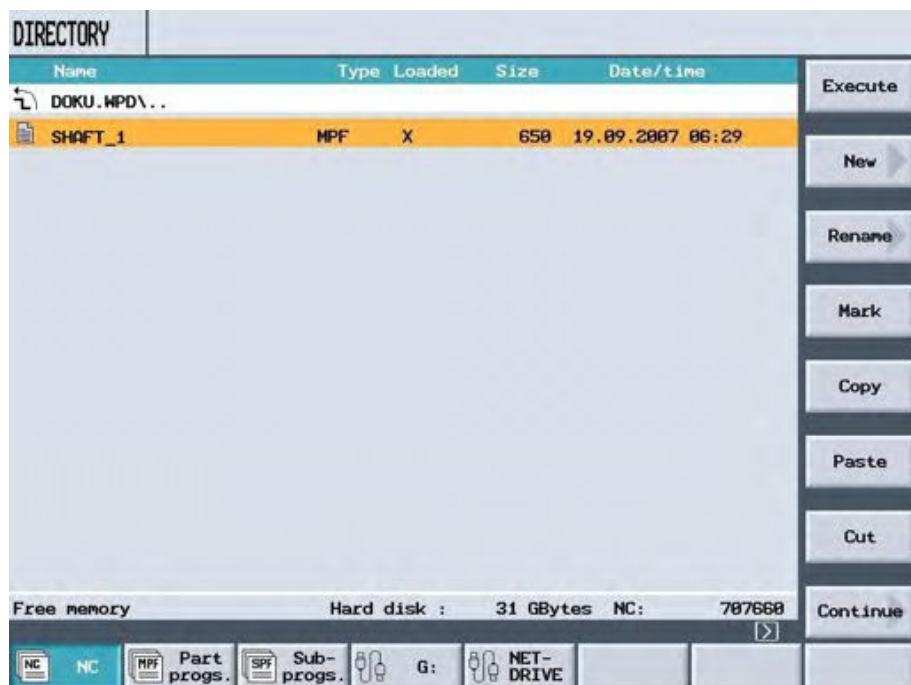
## Section 3

### Management of NC-programs

#### 3.1.3 Selecting and opening an NC-program

Open the respective directory as described in 3.1.1.

Notes



With the key "Cursor down" select the required NC-program.

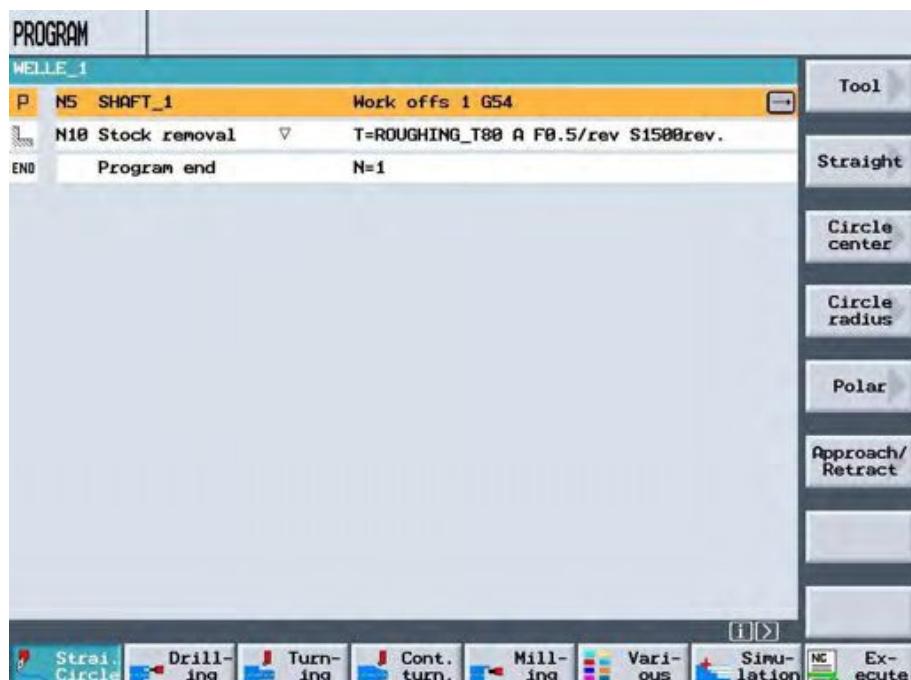


or



With the key "INPUT" or "Cursor right" open the program.

The selected program is shown in the operation sector "PROGRAM". The work plan of the program appears.



## Section 3

Notes

### Management of NC-programs

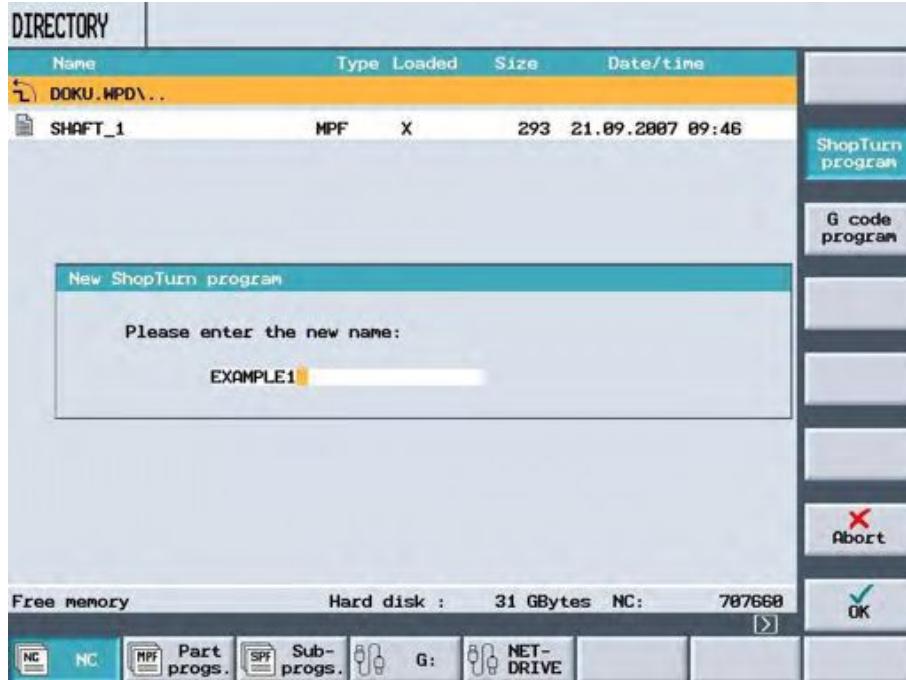
#### 3.2 Producing a new NC-program

Open the respective directory as described in Section 3.1.1.

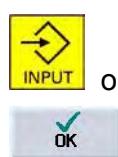


Press the VSK 2 "New" to create a new NC-part program.

The window "New ShopTurn Program" opens.



Press VSK 2 "ShopTurn Program".



Or



With the key "INPUT" or the VKS 8 "OK" accept the new program name.

The input mask for the new program for the input of the header line appears. See Section A111 - "Operating area Program - General Programming Structure".

**Hint:**

By pressing the VSK 3 "G-Code-Program" a G-Code-Program can now be written after input of the program name.

If the directory overview is already selected, a directory can be specified without any further pre-selection.

## Section 3

### Management of NC-programs

#### 3.3 Re-naming an NC-program

Open the respective directory as described in Section 3.1.1.



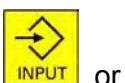
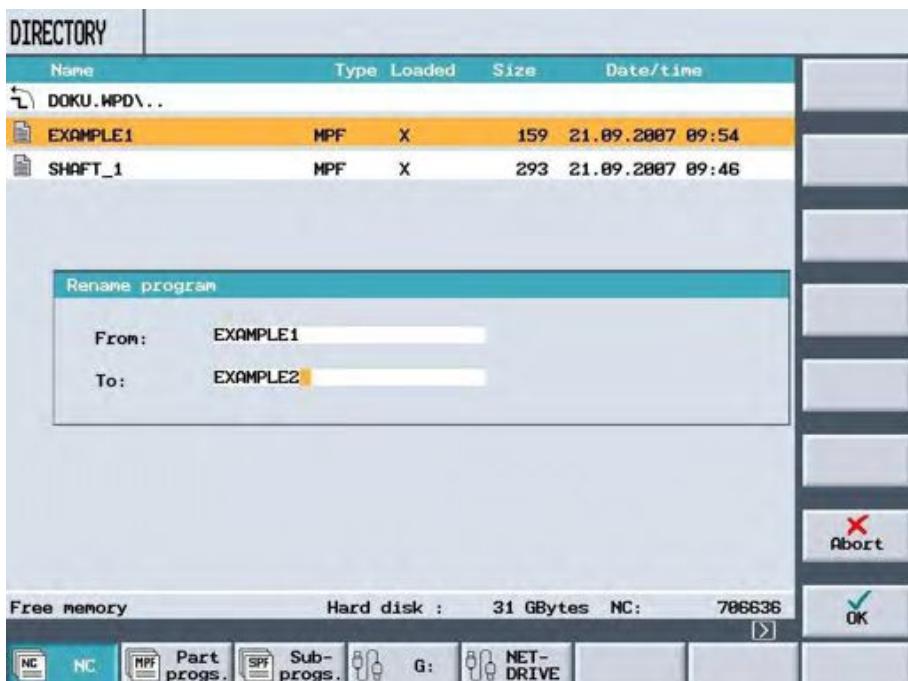
With the key "Cursor down" select the required NC-program.



Press VSK 3 „Rename“.

The window "Rename program" appears.

In the line "To:" enter the new program name, e.g. Example2.



With the key "INPUT" or the VKS 8 "OK" accept the new program name.



#### Hint:

If the respective directory overview is selected, the files can be renamed as required.

Notes

## Section 3

Notes

### Management of NC-programs

#### 3.4 Marking an NC-program

Open the respective directory as described in Section 3.1.1.



With the key "Cursor down" select the first NC-program to be marked.



Press the VSK 4 "Mark".



Select further NC-part programs with the key "Cursor down".



Subsequently the marked NC-programs can be transferred to the intermediate memory of the PCU 50 by pressing the key VSK 5 "Copy" or VSK 6 "Cut". (The Softkey "Cut" does not exist on PCU 20; instead there is the Softkey "Delete").

The programs in the intermediate memory can now be:

copied to another directory or to another storage medium

Completely deleted (cutting out without any further copying or insertion)

Deleted from this directory (cut out) and inserted into another directory or storage medium

**Hint:**

If the directory overview is selected, the respective files can be marked.

## Section 3

### Management of NC-programs

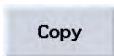
#### 3.5 Copying and insertion of an NC-program

Open the respective directory on the NC as described in Section 3.1.1.

Notes



With the key "Cursor down" select the required NC-program.



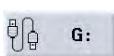
Press the VSK 5 "Copy".



If the program is to be copied to another directory on the NC, the file structure must be selected using the key "Cursor left" to



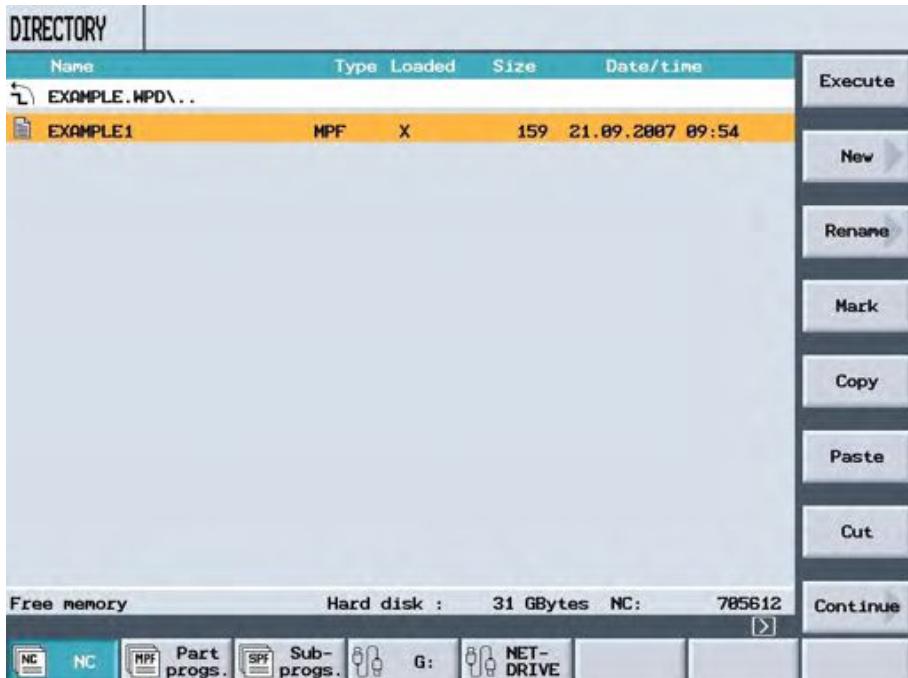
select the new directory. See also Section 3.1.1 - "Open directory".



Alternatively it is possible to select another storage medium on the horizontal Softkey strip, e.g. Drive G: (USB-Stick)



With the VSK 6 "Paste" the program can be written to the selected directory or the new storage medium.



**Hint:** If the directory overview is selected, the respective files can be copied and inserted.

If a program / directory already exists by the same name, a query is raised whether the new program / file should be renamed. See Section 3.3 „NC-part program rename“.

If a new name is not entered and the existing name is acknowledged by pressing the VSK 8 "OK", the old program / directory will be overwritten.

## Section 3

Notes

### Management of NC-programs

#### 3.6 Cutting / deleting an NC-program

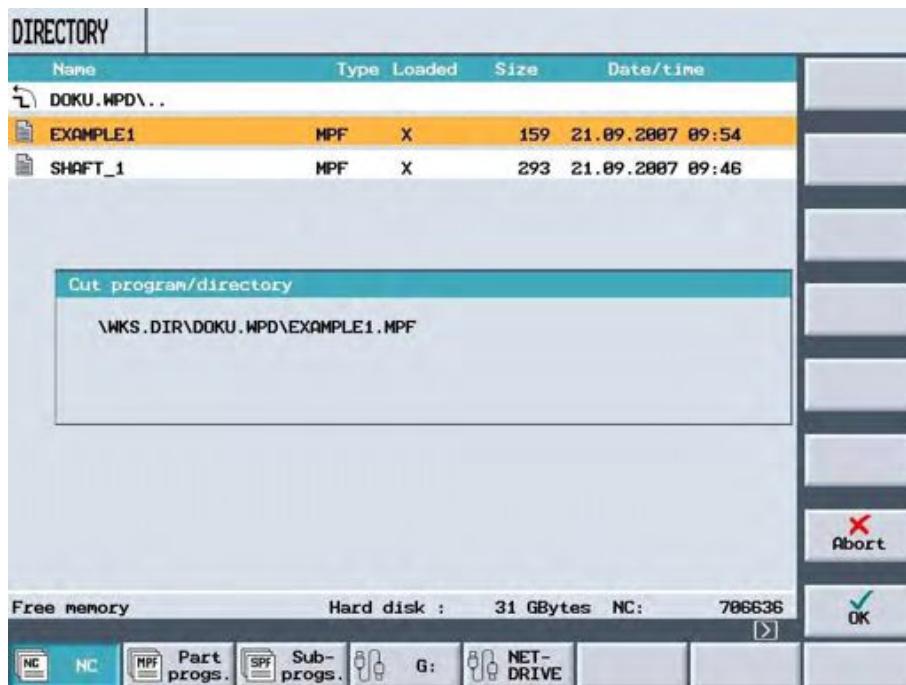
Open the respective directory as described in Section 3.1.1.



With the key "Cursor down" select the required NC-program.

Cut

Press the VSK 7 "Cut". (on PCU 20 "Delete")



Here follows a query whether to cut out the "Program / directory" (PCU 50) / Delete (PCU 20).



If the VSK 7 "Abort" is pressed, the cutting out will be terminated. The program/directory will not be deleted.



If the VSK 8 "OK" is pressed, the program will be cut out / deleted from the directory.

**Hint:**

If the directory overview is selected the respective files can be deleted.

After cutting, the program or directory can be inserted into another directory or another storage medium. See Section 3.5 - "NC-part program Copying and Inserting".

## Section 3

### Management of NC-programs

#### 3.7 Executing an NC-program

Open the respective directory as described in Section 3.1.1.

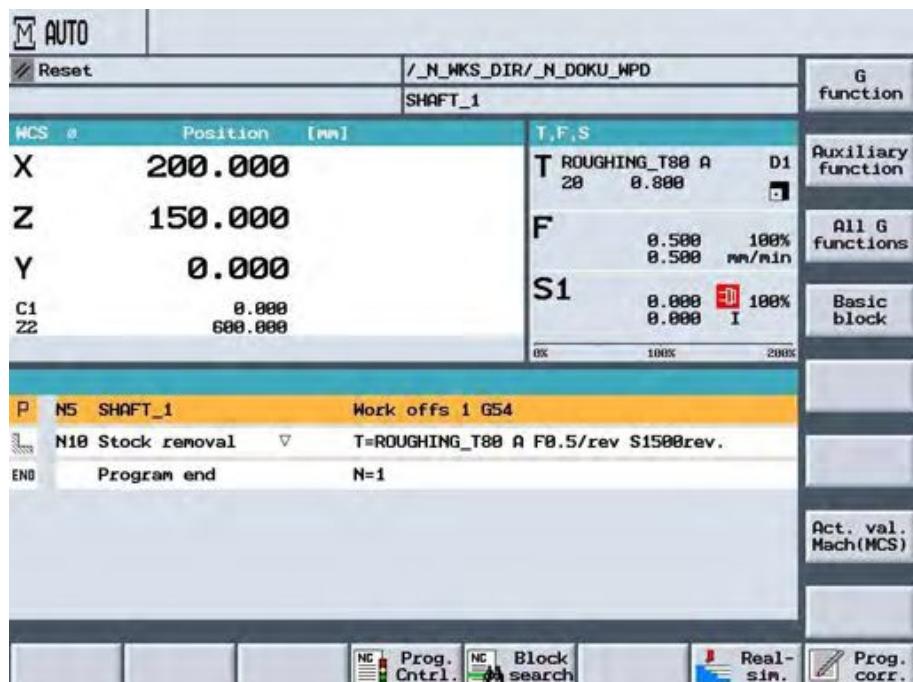


With the key „Cursor down“ select the required NC-program.

Execute

By pressing the VSK 1 "Execute" the selected program will be loaded and run in the Operation Sector "AUTO".

Notes



Switch ON the spindle drive.



Switch ON the feed drive.



Start the run of the program.

## Section 4

Notes

NC-programs load / unload

The following applies to the PCU 50:

Loaded: The NC-program is stored in the NC.

Not loaded: The NC-program is stored on the PCU 50. The function "Execution from the hard disk" is provided in this case.

Since the NC-memory provides only limited space, only the most important programs should be stored there.

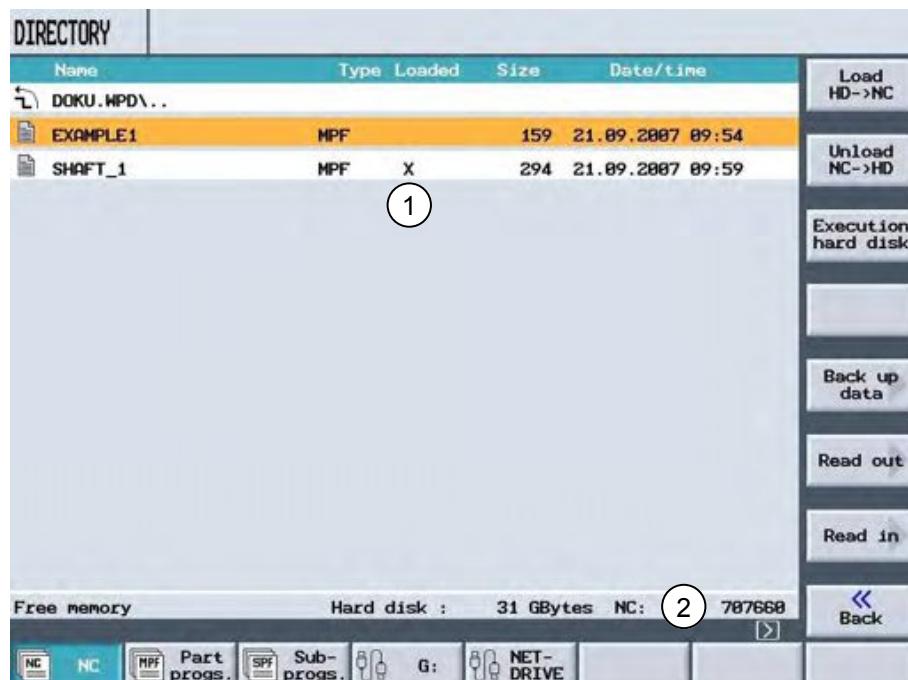
Open the respective directory as described in Section 3.1.1.



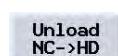
With the key "Cursor down" select the required NC-part program.

By pressing the VSK 8 "Continue" change to the next Softkey strip.

By pressing the VSK 1 "Manual load" load the NC-program into the NC-memory - only available for PCU 50.



- ① An "X" appears in the column "Loaded".
- ② The used NC-memory space increases, the space still available is reduced.



By pressing the VSK 2 "Unload NC→HD" the selected NC-program will be unloaded from the NC-memory and stored on the hard disk.

The "X" in the column „Loaded“ is deleted.

The NC-memory is once more available.

**Hint:**

NC-programs, which are activated and being executed, cannot be unloaded to the hard disk.

#### 5.1 Saving the tooling data

Apart from the NC-programs it is also possible to save the tool data and zero-point offsets.

This possibility can only be used to store the required tool and zero-point data for a particular program. If this program has to be re-run at a later stage it will be simple to revert to this tooling.

Open the respective directory as described in Section 3.1.1.



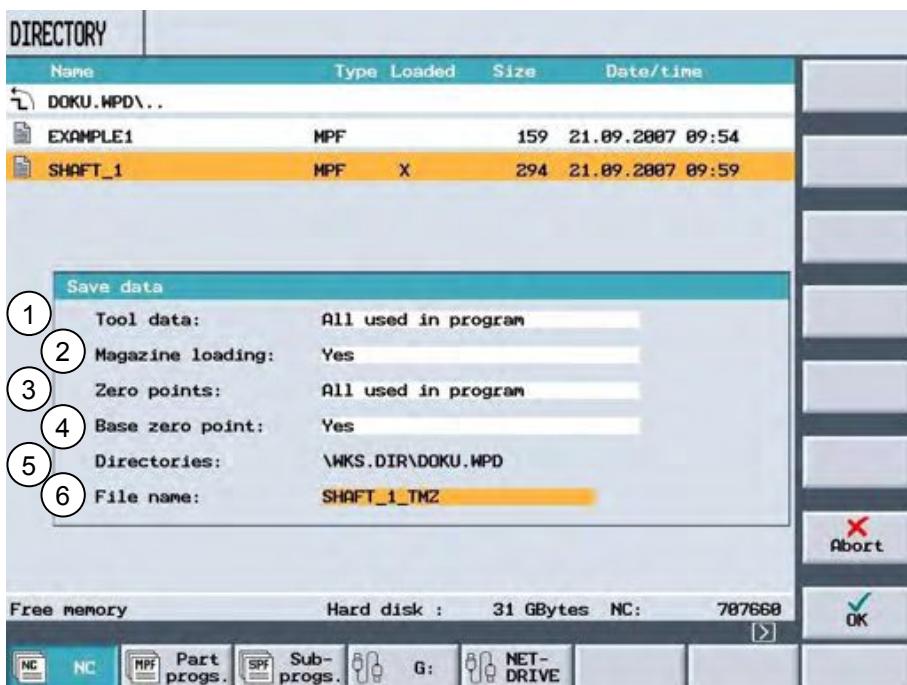
With the key "Cursor down" select the program of which the tool data and zero-points are to be stored.

Continue

By pressing the VSK 8 "Continue" change to the next Softkey strip.

Back up data

By pressing the VSK 5 "Back up data" call-up this function.



The following data can be saved:

- ① Tool data:
  - 1.1 None
  - 1.2 Complete tool list
  - 1.3 All those used in the program
- ② Magazine utilization (only in case of 1.2 and 1.3):  
Yes  
No

Notes

## Section 5

### Saving / re-loading of tooling data

Notes

- ③ Zero-points:
- 3.1 None
  - 3.2 Complete tool list
  - 3.3 All those used in the program
- ④ Basic zero-point (only in case of 3.2 and 3.3):
- Yes
  - No
- ⑤ Directory:
- The current directory path is shown.
- ⑥ File name:
- Here a file name for the saved data can be entered. As a default the name of the NC-part program + TMT is prompted



By pressing the VSK 8 "OK" the file WELLE\_1\_TMZ.INI will be created.

DIRECTORY					
Name	Type	Loaded	Size	Date/time	
DOKU.WPD\..					Load HD->NC
SHAFT_1_TMZ	INI	1	1319	21.09.2007 18:12	Unload NC->HD
EXAMPLE1	MPF	X	159	21.09.2007 09:54	Execution hard disk
SHAFT_1	MPF	X	294	21.09.2007 09:59	Back up data
					Read out
					Read in
					Back

- ① INI-files will not be loaded (only for PCU 50).

## Section 5

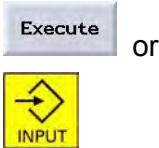
### Saving / re-loading of tooling data

#### 5.2 Loading the tooling data

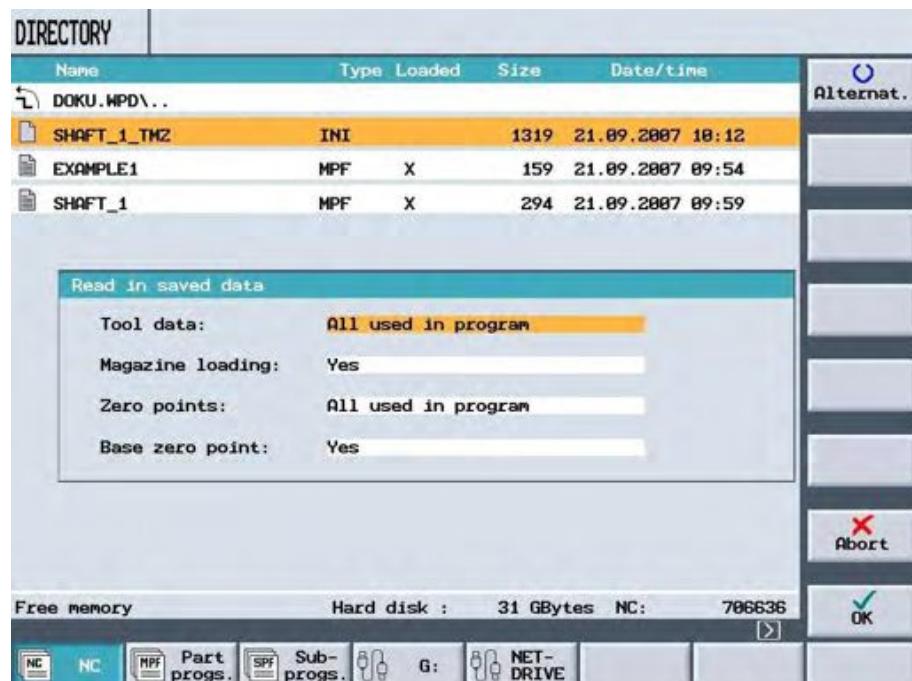
Open the respective directory as described in Section 3.1.1.



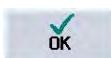
With the key "Cursor down" select the stored tool-/zero-point data (e.g. WELLE\_1\_TMZ.INI) that are to be loaded.



By pressing the VSK 1 "Execute" or the key INPUT the window "Saved data load" appears.



With the VSK 1 "Alternat." select the required tool- and zero-point data that are to be loaded. See also Section 5.1.



By pressing the VSK 8 "OK" the data will be loaded.

Depending on which data have been selected ShopTurn reacts as follows:

##### All tool correction data

Firstly all existing data will be cleared and then the saved data will be read in.

##### All tool offset data used in the program

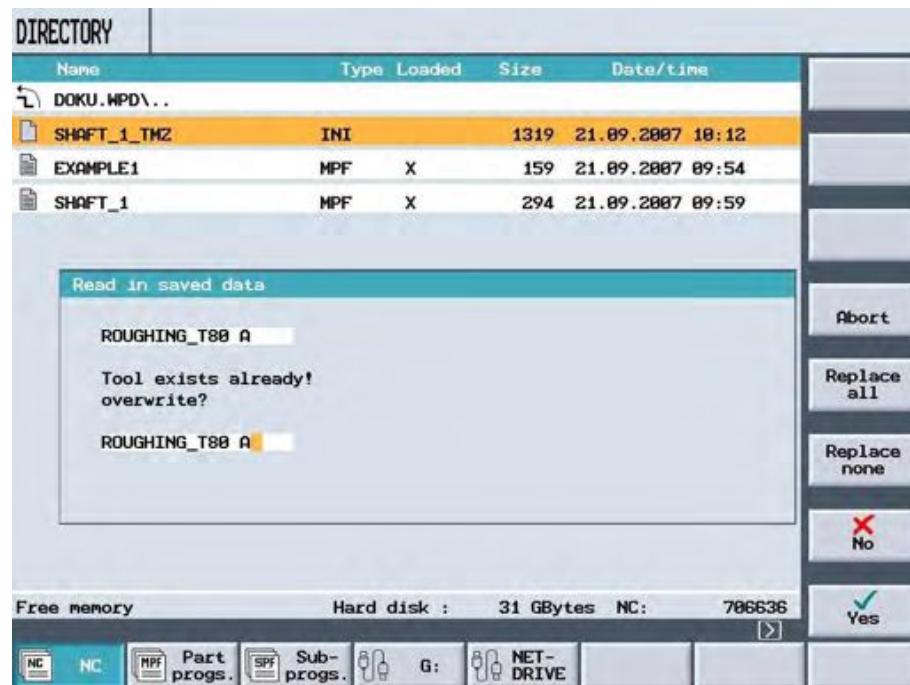
If at least one of the tools to be read in already exists in the tool management, one of the two following possibilities may be used.

Notes

## Section 5

### Saving / re-loading of tooling data

Notes



By pressing the VSK 5 "Replace all", all tool data will be loaded. Any further tools that may already exist will be overwritten without a prior query.

By pressing the VSK 6 "Replace none", the loading will be stopped.

By pressing the VSK 7 "No", the old tool remains. If the old tool is not in the saved magazine location, it will be re-sited.

By pressing the VSK 8 "Yes", the old tool will be overwritten.

#### Hint:

In case of tool management without loading/unloading the old tool will not be cleared; in case of the variant with loading/unloading, the old tool will previously be unloaded.

If before the transfer the tool name is changed with "Yes", the tool will be added to the tool list.

#### Zero-point offsets

Any existing zero-point offsets will always be overwritten during loading.

#### Magazine utilization

If the magazine utilization is not read in as well, the tools will be entered into the tool list without a location number.

## 1 Brief description

**Objective of the module:**

This module explains the usage, operation and handling of the tool management under ShopTurn.

**Description of the module:**

All tool data relevant for the machining, (e.g. tool length and radius corrections, wear data and turret location) are entered in the tool management and handled from there.

The tool management contains the following sub-menus:

- the list of tools
- the tool wear
- the turret assignment

An additional list for any specific machine can be configured by the machine manufacturer. The documentation by the machine manufacturer must be observed.

**Content of the module:**

Pre-selection and functions of the tool management

- Pre-selection of the tool management
- Functions of the tool management

List of tools

- Overview of the list of tools
- Vertical Softkey strip

Tool wear

- Pre-selection tool wear
- Overview tool wear
- Vertical Softkey strip

Magazine management

- Pre-selection and overview of the magazine management
- Vertical Softkey strip

Example: Defining a new tool

Example: Loading the magazine

## Operating Area - Tool management

Pre-selection and functions of the tool management

Pre-selection of the tool management

Functions of the tool management

Section 2

List of tools

Overview list of tools

Vertical Softkey strip

Section 3

Tool wear

Pre-selection tool wear

Overview tool wear

Vertical Softkey strip

Section 4

Magazine management

Pre-selection and overview magazine management

Vertical Softkey strip

Section 5

Example: Defining a new tool

Section 6

Example: Loading the magazine

Section 7

## Section 2

### Pre-selection and functions of the tool management

#### 2.1 Pre-selection of the tool management

The tool management can be pre-selected as follows:

On the full CNC-keyboard or the OP press the Hotkey "Offset" to open the tool management immediately.

or



Press the key "Menu Select" on the OP.

The following horizontal ShopTurn Softkey strip of appears.



By pressing the HSK 5 "Tool zero point" select the tool management / work offset range .

The tool management (list of tools) opens:

OFFSET									
Loc	Typ	Tool name	DP 1st cutting edge				Inslt length	12	Alternat.
			Lngth	X	Lngth	Z	Radius		
1	■	ROUGHING_T80 A	1	55.840	39.124	8.800	95.0 80	12.0	
2	□	DRILL_32	1	0.000	185.124	32.000	180.0		
3	□	FINISHING_T35 A	1	125.726	57.370	8.400	93.0 35	12.0	
4	□	ROUGHING_T80 I	1	-8.950	122.457	8.800	95.0 80	10.0	
5	□	PLUNGE_CUTTER_3 A	1	85.124	44.124	8.200	3.000	8.0	
6	□	FINISHING_T35 I	1	-12.658	121.807	8.400	95.0 35	8.0	
7	□	THREADING_T1.5	1	66.326	33.333	0.050			
8	□	CUTTER_8	1	87.833	74.621	8.800		3	
9	□	PLUNGE_CUTTER_3 I	1	-11.736	135.124	8.100	3.000	4.0	
10	□	DRILL_3	1	0.000	185.124	3.000	118.0		
11	□	BUTTON_TOOL_8	1	88.112	38.123	2.800			
12	□	THREADCUTTER_M6	1	0.000	145.132	3.000	1.000		
13									
14									

The list of tools can, depending on the requirements, consist of the following:

- at least one turret
- and tools that are not assigned to a turret location.

The configuration shown here as an example can differ from the configuration of the actual machine .

Hint:

The documentation by the machine manufacturer must be observed.

Notes

## Section 2

Notes

Pre-selection and functions of the tool management

### 2.2 Functions of the tool management

#### Display



#### Description

By pressing the HSK 1 the list of tools is called up.

*See Section 3 "List of tools".*



By pressing the HSK 2 the tool wear list is called up.

*See Section 4 "Tool wear".*



By pressing the HSK 4 the magazine management is called up.

*See Section 5 "Magazine".*



By pressing the HSK 5 the work offset is called up.

*See Module A107 - Operating area - Work offset".*



By pressing the HSK 6 the R-Parameter is called up.

*See Module A110 - Operating area - R-Parameter / Spindles".*



By pressing the key "Softkey-strip extension" and the HSK 2 the settings for the spindles will be called up.

*See Module A110 - Operating area - Spindles".*



By pressing the Taste "Softkey strip extension" and the HSK 8 the User data is called up.

## Section 3

### List of tools

#### 3.1 Overview of the list of tools

All tools and their correction data, which are stored in the NC, are shown in the tool list, no matter whether they are assigned to a turret location or not. All the usual milling, turning and boring tools as well as measuring probes are provided by default. The necessary geometrical and technological tool data can be assigned to these tool types.

Different kinds of tool correction data are required depending on the tool type.

Overview of the list of tools:

Loc	Typ	Tool name	DP 1st cutting edge				Insrt	12
			Lngh	X	Lngh	Z	Radius	
1	■	ROUGHING_T80 A	1	55.840	39.124	0.800	95.0 80	12.0
2	□	DRILL_32	1	0.000	185.124	32.000	188.0	
3	□	FINISHING_T35 A	1	125.726	57.378	0.400	93.0 35	12.0
4	■	ROUGHING_T80 I	1	-8.950	122.457	0.800	95.0 80	10.0
5	□	PLUNGE_CUTTER_3 A	1	85.124	44.124	0.200	3.000	8.0
6	□	FINISHING_T35 I	1	-12.658	121.887	0.400	95.0 35	8.0
7	□	THREADING_T1.5	1	66.326	33.333	0.050		
8	□	CUTTER_8	1	87.833	74.621	0.800		3
9	□	PLUNGE_CUTTER_3 I	1	-11.736	135.124	0.100	3.000	4.0
10	□	DRILL_3	1	0.000	185.124	3.000	118.0	
11	□	BUTTON_TOOL_8	1	88.112	38.123	2.000		
12	□	THREADCUTTER_M6	1	0.000	145.132	3.000	1.000	
13								
14								

Notes

- ① Number of the tool location (Turret location))
- ② Type of tool
- ③ Denomination of the tool
- ④ Duplo-number of the tool
- ⑤ Tool corrections (tool length, tool radius, tool diameter)
- ⑥ Reference direction for the tool carrier angle
- ⑦ Carrier angle of a turning tool , angle of the tool point on a drill, tip width of a grooving tool, thread pitch of a tap
- ... ⑧ Tip angle of a turning tool, number of teeth on a milling cutter
- ... ⑨ Tip length of a turning tool or grooving tool
- ⑩ Tool-specific functions such as spindle rotation, coolant 1 and 2

## Section 3

Notes

List of tools

①

### PL - Location number

The location number refers to the turret location.

The symbols have the following meaning:

1

Turret location number

If there is more than one turret on the machine, the location number is entered first followed by the turret number.

1/1

e.g. Location number 1 in turret 1

1/2

Location number 1 in turret 2

Tools without an assignment to a tool location are listed below the turret without a location number. If any of these tools are required for a particular program, they must be assigned to a turret location before starting the program execution.

②

### Tool Types

The following tool types are available for selection.

Roughing tool

Finishing tool

Parting tool

Thread cutting tool

Milling cutter

Drill

Button tool

Stopper

Tapr

Boring tool

3D-key



By pressing the VSK 1 "Alternat." select the required cutting edge position.

Depending on the type of tool only some of the tool correction data are released.

Depending on the type of tool up to 9 different cutting edge positions are available. For clarification these possibilities are shown below for a roughing tool:

#### ③ Tool name

A name in clear text or a T-number can be entered for identification of the tools. Standard names are provided by default when new tools are defined.

#### ④ Duplo-number

A "1" is entered here by default. If a new tool is defined, which bears the same name as one that is already in existence, this number will be increased to the next digit, i.e. "2". Thus it is possible to define tools of similar type.

#### ⑤ Tool corrections

The tool corrections available for selection depends on the chosen tool type.

The following tool corrections can be entered.

Length X

Tool length correction in X-direction

This value can be determined with the function "Measure tool". See Section 6 "Measure tool". If the tool is measured up externally, the measurement can be entered here.

Length Z

Tool length correction in X-direction

This value can be determined with the function "Measure tool". See Section 6 "Measure tool". If the tool is measured up externally, the measurement can be entered here.

Radius resp. Ø

Radius resp. diameter of the tool

For milling and drilling tools it is also possible to state the diameter; in case of turning tools only the radius is entered. The switching of radius or diameter input is specified by a machine datum.

#### ⑥ Reference direction for the carrier angle

## Section 3

Notes

List of tools

(7)

### Additional tool data

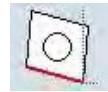
Depending on the selected tool type the following tool data will be shown here.

Carrier angle

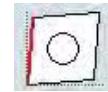
Carrier angle of a cutting tool (roughing tool or finishing tool)

The carrier angle is taken into account when machining undercuts. The size of the angle is shown by means of a graphic (e.g. tip angle constant 80°).

15°



95°



Angle

Tool tip angle for a drill

If when drilling the infeed is to be as far as the shank and not just to the tool point, the control takes the tool angle into account.

Tip width

Tip width of a grooving tool

The tip width is required by ShopTurn for the evaluation of the grooving cycles.

Pitch

Input of the thread pitch of a tap in mm/rev.

If the machine is set to the Inch-system, the pitch must be entered in threads/ ".

(8)

### Additional tool data

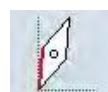
Depending on the selected tool type the following tool data will be shown here.

Tip angle

Tip angle of a cutting tool

The tip angle is taken into account for undercuts. The angle value is shown by means of a graphic depiction, (e.g. carrier angle constant 35°)

35°



80°



N

Number of teeth on a milling cutter

The control determines from this the rotary feed rate if in the program the feed per tooth is selected.

(9)

### Tip length of a cutting tool or grooving tool

The tip length is required by ShopTurn for the depiction of tools during simulation of part programs.

## Section 3

### List of tools

#### ⑩ Tool specific functions



Direction of spindle rotation

⊗ Spindle OFF

↖ Spindle rotates CCW

↗ Spindle rotates CW



Coolant application 1 and 2



or



Coolant ON-OFF

Notes

## Section 3

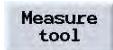
Notes

### List of tools

#### 3.2 Vertical Softkey strip



If there are any alternative input possibilities for a selected parameter, they can be chosen using the VSK 1 "Alternative".



By pressing the VSK 2 "Tool meas." this function can be selected.

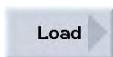
See Module A103 - "Manual Operating area ShopMill, Section 6 "Tool measuring".



By pressing the VSK 3 "Tool delete" the selected tool is removed from the tool management.



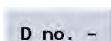
By pressing the VSK 4 "Unload" the tool in the turret is removed from the turret. The tool is shown in the list below the turret. See also Section 6 "Magazine unloading and loading".



By pressing the VSK 4 "Load" the tool can be loaded from the tool list to a turret location. The next free turret location is offered. See also Section 7 "Magazine unloading and loading".



By pressing the VSK 6 "Cutting edges" the respective menu with further Softkeys is called up.



With the Softkeys "D-no +" resp. "D-no -" the correction data of the cutting edge with the next higher or lower cutting edge number can be shown.



With the Softkey "New tool edge" it is possible to define a new tool edge. Instead of the input fields for the 1st tool edge the input fields for the correction data for the 2nd tool edge is shown in the tool list.



The Softkey "Delete cutting edge" permits the deletion of correction data for a tool edge. It is only possible to delete the data of the cutting edge with the highest number.

Hint:

In case of tools with several cutting edges, each edge will be given its own set of correction data. For each tool up to 9 cutting edges can be defined. Gaps in the numbering are not permissible, i.e., if 3 edges are required for a tool, these must be numbered 1 to 3.

## Section 3

### List of tools

Sort

By pressing the VSK 7 "Sort" the tools can be sorted according to the following criteria:

- Magazine
- Name
- Type
- T-number

The respective Softkeys are provided on the vertical Softkey strip.

New tool

By pressing the VSK 8 "New tool", a new tool can be defined. See *Section 6 "Defining a new tool"*.

Notes

## Section 4

Notes

### Tool wear

Tools that are being used for some time can become worn. This wear can be measured and entered into the tool wear list. ShopTurn will then take these data into account for the evaluation of the tool length or radius correction. Additive correction data for tools can be entered into the tool wear list.

#### 4.1 Pre-selection tool wear



By pressing the HSK 2, the function "Tool wear" in the tool management can be selected.

Loc	Typ	Tool name	DP 1st cutting edge						Cutting edges	Sort
			ΔLgth X	ΔLgth Z	ΔRadius	T	Prewarn	Tool Climit		
1	■	ROUGHING_T88 A	1	0.000	0.000	0.000	T	0.0	0.0	
2	□	DRILL_32	1	0.000	0.000	0.000				
3	■	FINISHING_T35 A	1	0.000	0.000	0.000	C	0	0	
4	■	ROUGHING_T88 I	1	0.000	0.000	0.000				
5	□	PLUNGE_CUTTER_3 A	1	0.000	0.000	0.000				
6	■	FINISHING_T35 I	1	0.000	0.000	0.000				
7	■	THREADING_T1.5	1	0.000	0.000	0.000				
8	■	CUTTER_8	1	0.000	0.000	0.000				
9	■	PLUNGE_CUTTER_3 I	1	0.000	0.000	0.000				
10	□	DRILL_3	1	0.000	0.000	0.000				
11	□	BUTTON_TOOL_8	1	0.000	0.000	0.000				
12	□	THREADCUTTER_M6	1	0.000	0.000	0.000				
13										
14										

- ① Number of tool location (display only, see *Section 3 "List of tools"*)
- ② Tool type (display only, see *Section 3 "List of tools"*)
- ③ Tool name (display only, see *Section 3 "List of tools"*)
- ④ Duplo-number of the corresponding tool (replacement tool) (display only, see *Section 3 „List of tools“*)
- ⑤ Wear corrections (Length, Radius, diameter)
- ⑥ Anwahl Werkzeugüberwachung (Zeit / Stückzahl)
- ⑦ Pre-warning limiting value
- ⑧ Tool life / count number (number of loadings)
- ⑨ Tool condition

## Section 4

### Tool wear

#### 4.2 Overview tool wear

The columns 1 - 4 are display-only fields. The input is made in the list of tools, see *Section 3 „List of tools“*.

Notes

##### ⑤ Wear corrections

$\Delta$  Length X,  
 $\Delta$  Length Z      Change in length  
Any changes in the tool length are entered in these fields.

$\Delta$  Radius,  
 $\Delta \emptyset$       Change in radius/diameter  
Any changes in the tool radius/diameter are entered in this field.

ShopTurn checks whether the entered values do not exceed the incremental or absolute upper limits.  
The incremental upper limit is the maximum difference between the present and the new wear value.  
The absolute upper limit states the maximum total value that can be entered.  
Please observe the hints in the documentation by the machine manufacturer.

##### ⑥ Tool monitoring

The tool monitoring can be activated by means of a machine datum. Please observe the documentation by the machine manufacturer.

T (Tool life)      With tool life T (Time) the time of actual usage of the tool with machining feed rate in minutes is being monitored.

C (Counts)      The count C counts the number of times the tool has been loaded into the spindle.

W (Wear)      With wear W (Wear) the maximum value of the wear parameter  $\Delta$ length X,  $\Delta$ length Z or  $\Delta$ radius or  $\Delta \emptyset$  in the wear list is being monitored.

Once the specified remaining tool life, count or wear value is attained, the tool will be blocked. The tool will no longer be used for the next tool change. If available a corresponding tool (replacement tool) will be loaded instead. The monitoring refers to the selected cutting edge.

##### ⑦ Pre-warning limit

The pre-warning limit states a tool life, count number or wear value when a first warning is issued.

## Section 4

Notes

### Tool wear

#### ⑧ Tool life / count number

Input field for the monitoring limits for the tool life, the count number or the wear.

#### ⑨ Tool condition

**B**

This defines a blocked tool. This will be automatically set when the tool life, the number-of-machined or the wear limit has been reached. Tools can also be blocked manually if they are not to be used any more or the tool life has expired.

**O**

This signifies oversized tools for which for example the dimensions are such that they reach into the neighbouring tool space in the turret. Once loaded, the neighbouring turret locations cannot be used any more.

Oversized tools should be loaded in a fixed turret location. This, however, must be defined by the manufacturer.

## Section 5

### Magazine management

#### 5.1 Pre-selection and overview of the Magazine management



By pressing the HSK 4 of the tool management the function "Magazine" can be selected.

Notes

Loc	Typ	Tool name	DP Loc.	Tool disabl	State	Block magazine loc.
1		ROUGHING_T80 A	1			
2		DRILL_32	1			
3		FINISHING_T35 A	1			
4		ROUGHING_T80 I	1			
5		PLUNGE_CUTTER_3 A	1			
6		FINISHING_T35 I	1			
7		THREADING_T1.5	1			
8		CUTTER_8	1			
9		PLUNGE_CUTTER_3 I	1			
10		DRILL_3	1			
11		BUTTON_TOOL_8	1			
12		THREADCUTTER_M6	1			
13						
14						

- ① Number of tool locations (display only, see *Section 3 "List of tools"*)
- ② Tool type (display only, see *Section 3 „List of tools“*)
- ③ Tool name (display only, see *Section 3 „List of tools“*)
- ④ Duplo-number (display only, see *Section 3 „List of tools“*)
- ⑤ Blocked location
- ⑥ Tool condition (display only, see *Section 4 „Tool wear“*)

The columns Type and Tool name

- these indicate whether a turret space is free or occupied.

Column Location disabled

- shows whether a location is blocked. The blockage can be set manually in order to signify a damaged location. The tool in a blocked location can no longer be selected.

Column Tool state

- shows whether the tool is

B: blocked (tool life, number-off or wear limit is reached)

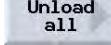
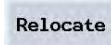
O: is oversized.

## Section 5

Notes

### Magazine management

#### 5.2 Vertical Softkey strip

Display area	Description
 <b>Unload all</b>	By pressing the VSK 4 "Unload all" all the tools in the turret will be unloaded and then shifted in the tool list behind the turret.
 <b>Relocate</b>	By pressing the VSK 5 "Relocate" tools can be re-located to another unused turret location or loaded into the spindle.

## Section 6

### Example: Defining a new tool



Press the key "Menu Select" for instance on the OP. See also Section 2 "Pre-selection and functions of the tool management".

By pressing the HSK 5 "ToolsWOS." select the tool management / work offset.

Notes

OFFSET										
Tool list										
Loc	Typ	Tool name	DP 1st cutting edge					Inslt lngth	12	
			Lngth	X	Lngth	Z	Radius			
1	■	ROUGHING_T80_A	1	55.848	39.124	8.800	←	95.0 80	12.0	
2	□	DRILL_32	1	0.000	185.124	32.000		180.0	2	
3	■	FINISHING_T35_A	1	125.726	57.370	0.400	←	93.0 35	12.0	
4	■	ROUGHING_T80_I	1	-8.950	122.457	8.800	←	95.0 80	10.0	
5	□	PLUNGE_CUTTER_3_A	1	85.124	44.124	0.200		3.000	8.0	
6	□	FINISHING_T35_I	1	-12.658	121.807	0.400	←	95.0 35	8.0	
7	■	THREADING_T1.5	1	66.326	33.333	0.050		2	2	
8	■	CUTTER_8	1	87.833	74.621	8.800		3	2	
9	□	PLUNGE_CUTTER_3_I	1	-11.736	135.124	0.100		3.000	4.0	
10	□	DRILL_3	1	0.000	185.124	3.000		118.0	2	
11	□	BUTTON_TOOL_8	1	88.112	38.123	2.000			2	
12	□	THREADCUTTER_M6	1	0.000	145.132	3.000		1.000	2	
13										
14										



Call up the selections for new tools.  
The vertical Softkey strip tenders a tool pre-selection.

OFFSET										
Tool list										
Loc	Typ	Tool name	DP 1st cutting edge					Inslt lngth	12	
			Lngth	X	Lngth	Z	Radius			
1	■	ROUGHING_T80_A	1	55.848	39.124	8.800	←	95.0 80	12.0	
2	□	DRILL_32	1	0.000	185.124	32.000		180.0	2	
3	■	FINISHING_T35_A	1	125.726	57.370	0.400	←	93.0 35	12.0	
4	■	ROUGHING_T80_I	1	-8.950	122.457	8.800	←	95.0 80	10.0	
5	□	PLUNGE_CUTTER_3_A	1	85.124	44.124	0.200		3.000	8.0	
6	□	FINISHING_T35_I	1	-12.658	121.807	0.400	←	95.0 35	8.0	
7	■	THREADING_T1.5	1	66.326	33.333	0.050		2	2	
8	■	CUTTER_8	1	87.833	74.621	8.800		3	2	
9	□	PLUNGE_CUTTER_3_I	1	-11.736	135.124	0.100		3.000	4.0	
10	□	DRILL_3	1	0.000	185.124	3.000		118.0	2	
11	□	BUTTON_TOOL_8	1	88.112	38.123	2.000			2	
12	□	THREADCUTTER_M6	1	0.000	145.132	3.000		1.000	2	
13										
14										

## Section 6

Notes

Example: Defining a new tool

Roughing tool

Select the required tool with the required cutting edge position, e.g. a roughing tool.  
The tool will be assigned to the selected location.

OFFSET		Tool list								
Loc	Typ	Tool name	DP 1st cutting edge				Insert angle	Insert length		
			Length X	Length Z	Radius	Ref. direction				
1	■	ROUGHING_T88 A	1 55.840	39.124	8.800 ←	95.888	12.0			
2	□	DRILL_32	1 0.000	185.124	32.000	180.0				
3	■	FINISHING_T35 A	1 125.726	57.370	0.400 ←	93.035	12.0			
4	■	ROUGHING_T88 I	1 -8.950	122.457	8.800 ←	95.888	18.0			
5	□	PLUNGE_CUTTER_3 A	1 85.124	44.124	8.200	3.000		8.0		
6	■	FINISHING_T35 I	1 -12.658	121.807	0.400 ←	95.035	8.0			
7	■	THREADING_T1.5	1 66.326	33.333	0.850					
8	■	CUTTER_8	1 87.833	74.621	8.800		3			
9	■	PLUNGE_CUTTER_3 I	1 -11.736	135.124	0.100	3.000		4.0		
10	□	DRILL_3	1 0.000	185.124	3.000	118.0				
11	□	BUTTON_TOOL_8	1 88.112	38.123	2.000					
12	□	THREADCUTTER_M6	1 0.000	145.132	3.000	1.000				
13	■	ROUGHING_TOOL	8 0.000	0.000	8.800 ←	93.055	11.0			
14										

Cutting edges  
Sort  
New tool

Following this the tool name can be altered, e.g. roughing tool\_85.

If necessary, enter further tool data such as length, radius, reference direction etc..

OFFSET		Tool list								
Loc	Typ	Tool name	DP 1st cutting edge				Insert angle	Insert length		
			Length X	Length Z	Radius	Ref. direction				
1	■	ROUGHING_T88 A	1 55.840	39.124	8.800 ←	95.888	12.0			
2	□	DRILL_32	1 0.000	185.124	32.000	180.0				
3	■	FINISHING_T35 A	1 125.726	57.370	0.400 ←	93.035	12.0			
4	■	ROUGHING_T88 I	1 -8.950	122.457	8.800 ←	95.888	18.0			
5	□	PLUNGE_CUTTER_3 A	1 85.124	44.124	8.200	3.000		8.0		
6	■	FINISHING_T35 I	1 -12.658	121.807	0.400 ←	95.035	8.0			
7	■	THREADING_T1.5	1 66.326	33.333	0.850					
8	■	CUTTER_8	1 87.833	74.621	8.800		3			
9	■	PLUNGE_CUTTER_3 I	1 -11.736	135.124	0.100	3.000		4.0		
10	□	DRILL_3	1 0.000	185.124	3.000	118.0				
11	□	BUTTON_TOOL_8	1 88.112	38.123	2.000					
12	□	THREADCUTTER_M6	1 0.000	145.132	3.000	1.000				
13	■	ROUGHING_TOOL	8 0.000	0.000	8.800 ←	93.085	11.0	X		
14										

Measure tool  
Delete tool  
Unload  
Cutting edges  
Sort

## Section 7

### Example: Loading the magazine



Press the key "Menu Select" for instance on the OP. See also Section 2 "Pre-selection and functions of the tool management".



By pressing the HSK 5 "Tools WO" select the tool management / Work offset.



Select the required tool from the list of tools.

The screenshot shows the "OFFSET" window with the "Tool list" tab selected. The table displays tool information with columns: Loc, Typ, Tool name, DP 1st cutting edge, Lngth X, Lngth Z, Radius, Inslt, and Inslt length. The tool "ROUGHING\_T80 I" is highlighted. A context menu is open on the right side of the table, listing options: Alternat., Measure tool, Delete tool, Unload, Cutting edges, and Sort. At the bottom of the window, there are buttons for Tool list, Tool wear, OEM Werkz. daten, Magazine, Work offset, R vari., and R vari..

Load

The window "Empty location" is called up.  
Depending of the set search strategy the next free location will be shown.

The screenshot shows the "OFFSET" window with the "Tool list" tab selected. The table displays tool information. An empty row at location 24 is highlighted with the label "Empty location". The "Loc." column shows the value "1". A context menu is open on the right side of the table, listing options: Alternat., Measure tool, Delete tool, Unload, Cutting edges, and Sort. At the bottom of the window, there are buttons for Tool list, Tool wear, OEM Werkz. daten, Magazine, Work offset, R vari., and R vari.. On the far right, there are two buttons: "Abort" with a red X and "OK" with a green checkmark.

Notes

## Section 7

### Example: Loading the magazine

Notes



By pressing the VSK 8 "OK" the tool will be assigned to the selected turret space.

**Hint:**

If in the field "Magazine" or "Location" a "0" is entered, it is possible to search in all available turrets for the next free location.

OFFSET									
Loc	Typ	Tool name	DP 1st cutting edge				Insr t length	12	Alternat.
			Length X	Length Z	Radius				
1	□	ROUGHING_T80 I	1 -8.950	122.457	0.800	←	95.0 80	18.0	
2									
3									
4									
5	□	PLUNGE_CUTTER_3 A	1 85.124	44.124	0.200	3.000	8.0	2	
6	◀	FINISHING_T35 I	1 -12.658	121.807	0.400	←	95.0 35	8.0	2
7	□	THREADING_T1.5	1 66.326	33.333	0.050			2	
8	□	CUTTER_8	1 87.833	74.621	0.800		3	2	
9	□	PLUNGE_CUTTER_3 I	1 -11.736	135.124	0.100	3.000	4.0	2	
10	□	DRILL_3	1 0.000	185.124	3.000	118.0		2	
11	□	BUTTON_TOOL_8	1 88.112	38.123	2.000			2	
12	□	THREADCUTTER_M6	1 0.000	145.132	3.000	1.000		2	
13	□	ROUGHING_TOOL	1 58.600	40.500	0.800	←	93.0 85	11.0	X
14									

## 1 Brief description

**Objective of the module:**

This module enables you to understand and use the operating and programming philosophy of the selectable and programmable zero point offsets under ShopTurn.

**Description of the module:**

The menu “Work offset” permits the input of the offset and rotation values of the selectable zero point offsets (ZPO) as well as the display of the programmed zero point offsets.

**Content of the module:**

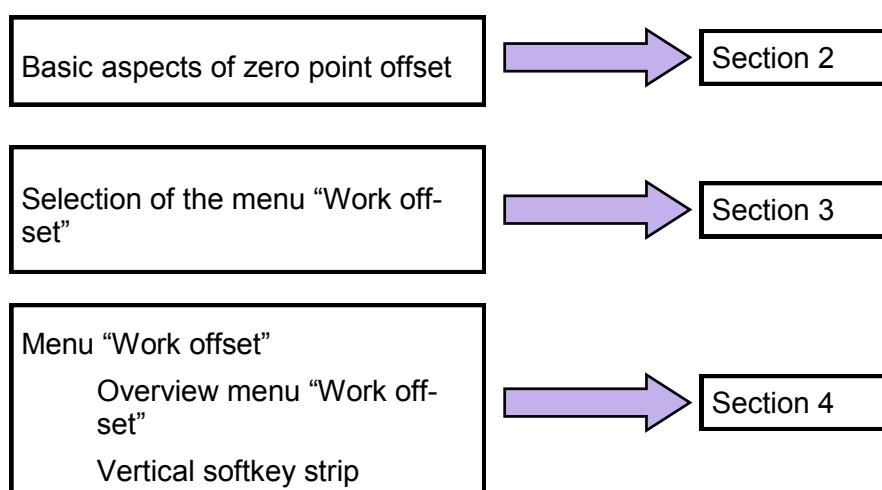
Basic aspects of zero point offset

Selection of the menu “Work offset”

Menu “Work offset”

- Overview menu “Work offset”

- Vertical softkey strip



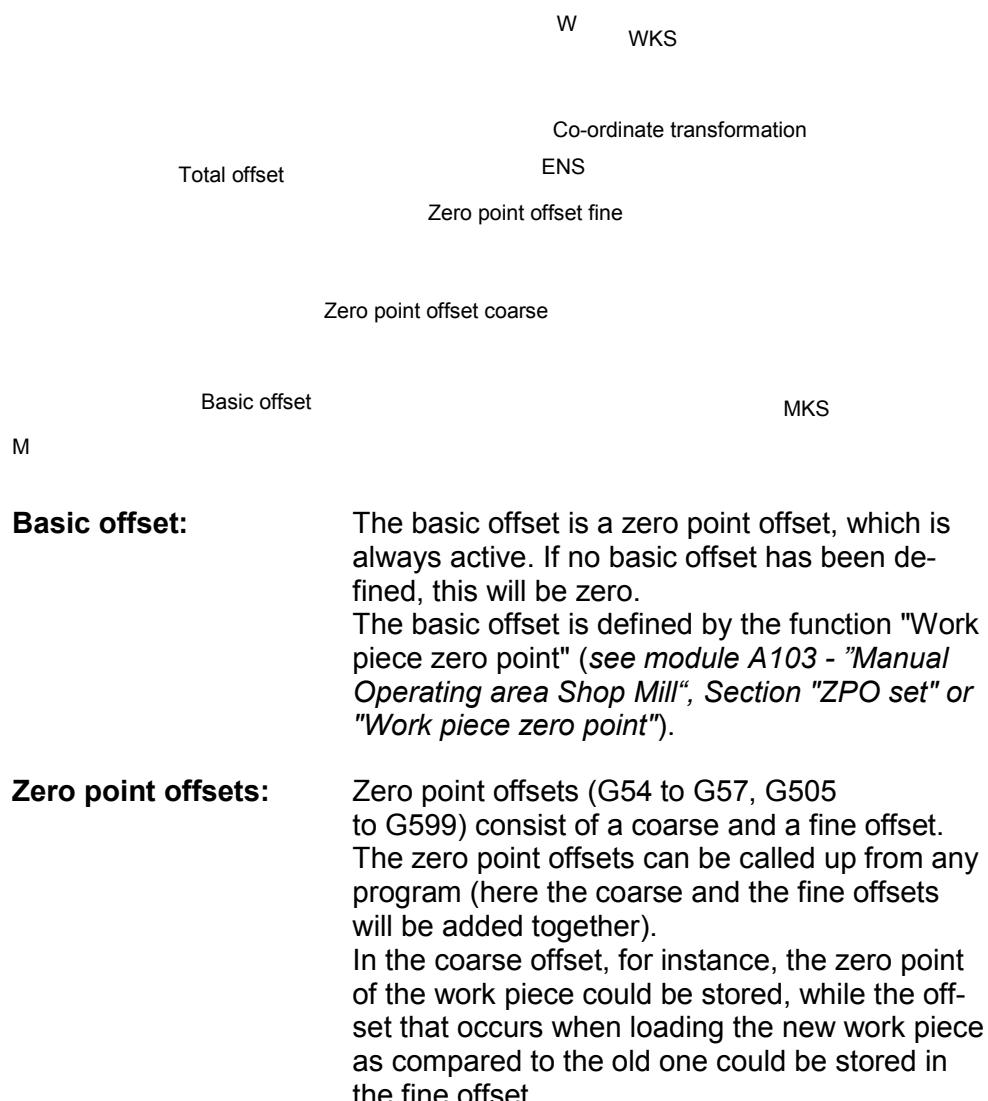
## Section 2

Notes

### Basic aspects of zero point offset

The actual value display of the axis co-ordinates after referencing refers to the machine zero point (M) of the machine co-ordinate system (MCS). However, the program for machining the work piece refers to the work piece zero point (W) of the work piece co-ordinate system (WCS). The distance between the machine zero point and the work piece zero point can vary depending on how the work piece has been loaded. This zero point offset is taken into account during machining from the program and can consist of a number of offsets. In case of ShopTurn the actual value display of the positions refers to the Szs co-ordinate system (selectable zero point system). The position of the active tool relative to the work piece zero point will be displayed.

The offsets cumulate as follows:



## Section 2

### Basic aspects of zero point offset

Notes

The fine offsets must be determined by the machine manufacturer.  
The documentation by the machine manufacturer must be regarded.

#### Co-ordinate transformation

Co-ordinate transformations are always programmed for a particular program only. They are defined by:

- Offset
- Rotation
- Scaling
- Mirroring

These transformations can be new or can be cumulative to the active ZPO.

*These programmable ZPO are described in module A134 - "Operating area Program - Edit Programming Various: Transformations".*

#### Total offset:

The total offset is the sum of all individual offsets and co-ordinate transformations.

## Section 3

Notes

Selection of the menu “Work offset”

The menu “Work offset” can be selected as follows:

By pressing the hotkey “Offset“ either on the CNC keyboard or the OP the Tool Management / Zero Point Offset will open immediately.

or



Press the key “Menu Select“ on the OP.

The following horizontal Softkey-strip of ShopTurn is displayed.



Pre-select the range Tool management / Zero Point Offset by pressing the HSK 5 “Tools WOs“.



Open the menu “Work offset” by pressing the HSK 5 “Work offset“.

The menu „Work offset“ is opened:

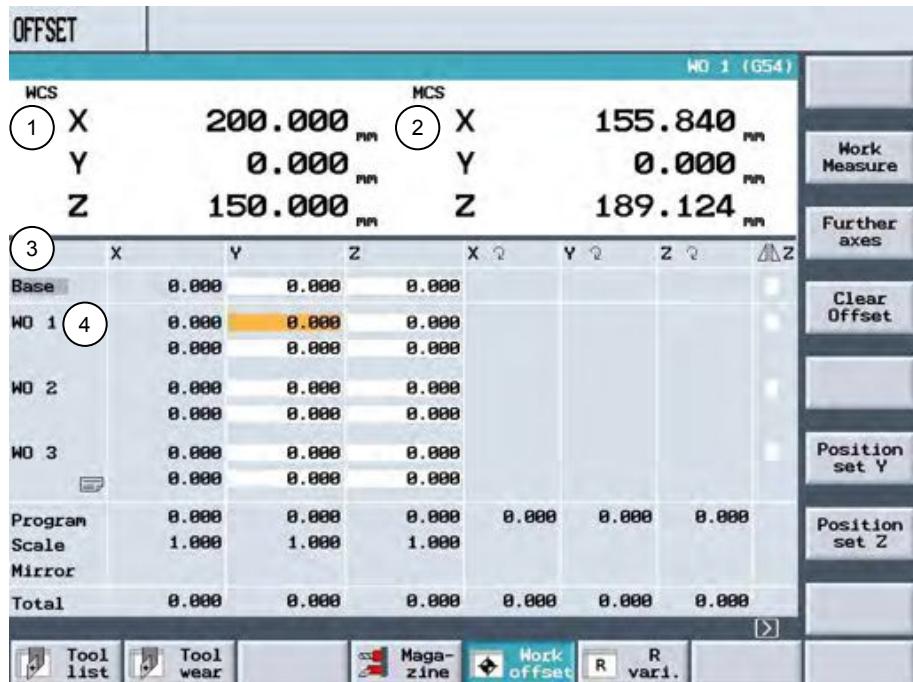
OFFSET				WO 1 (G54)				Work Measure Further axes Clear Offset Position set Y Position set Z	
WCS		MCS		X		Y			
X	200.000	mm	X	155.840	mm				
Y	0.000	mm	Y	0.000	mm				
Z	150.000	mm	Z	189.124	mm				
X	Y	Z		X	Y	Z	AZ		
Base	0.000	0.000	0.000						
WO 1	0.000	0.000	0.000						
	0.000	0.000	0.000						
WO 2	0.000	0.000	0.000						
	0.000	0.000	0.000						
WO 3	0.000	0.000	0.000						
	0.000	0.000	0.000						
Program	0.000	0.000	0.000	0.000	0.000	0.000			
Scale	1.000	1.000	1.000						
Mirror									
Total	0.000	0.000	0.000	0.000	0.000	0.000			

## Section 4

### Menu “Work offset”

#### 4.1 Overview menu “Work offset”

Notes



- ① Position display in the work piece co-ordinate system (WCS)
- ② Position display in the machine co-ordinate system (MCS)
- ③ List of zero point offsets for the X, Y and Z-axis  
The following selectable ZPO can be entered:  
Basic offset (G500)  
ZPO 1 - 4 (G54 - G57) with the coarse and fine offsets and possibly further ZPO (from G 505)  
Rotations

The programmable Frames offsets will be displayed:

Program / Trans, Rot  
Scale  
Mirror

and the resulting total offset.

Assignment of the columns:

- |                                   |                      |
|-----------------------------------|----------------------|
| <b>X</b>                          | Display of offsets   |
| <b>X <math>\varnothing</math></b> | Display of rotations |

- ④ The active ZPO is shown on a dark grey background, e.g. ZPO 1.

## Section 4

Notes

Menu "Work offset"

### Selectable zero point offsets:

#### Basic offset

Here the co-ordinates of the basic offset can be entered.

#### Basic reference (G500)

#### Zero point offsets

#### ZPO1 - ZPO4

The co-ordinates of the individual zero point offsets (1st line coarse offset, 2nd line fine offset) and the angle, by which the co-ordinate system may be rotated about an axis, can be entered here.

The fine offsets must have been entered by the machine manufacturer.



If further ZPO are available, they can be displayed using the keys "Page down" and „Page up“.

### Programmable zero point offsets (co-ordinate transformations)

#### Program

The programmable frames, such as Trans and Rot are shown here.

#### Scale

The programmable frame Scale is displayed here.

#### Mirror

The programmable frame Mirror is displayed here.

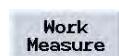
The co-ordinate transformations can be programmed either as new or cumulative. This determines the effect on the active zero point offset.

#### Total offset

#### Total

The total offset resulting from the basic offset and all active zero point offsets and co-ordinate transformations is displayed here.

## 4.2 Vertical softkey strip



By pressing the VSK 2 "Work measure" the display changes to the manual operating sector of Shop-Turn. The function "Zero point work piece set" is called up. See module 103 - "Manual operating area ShopTurn", Section 5 "Zero point work piece set".



By pressing the VSK 3 "Further axes" the ZPO for the rotational axes can be displayed and the associated offsets can be entered.

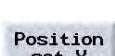
The documentation by the machine manufacturer must be heeded.



By pressing the VSK 4 "ZPO clear" the values for the coarse and fine offsets can be cleared.



By pressing the VSK 6 - 7 "Set Y" or „Set Z“ the present position value of the axis will be transferred from the position readout to the respective coarse offset.



## 1 Brief description

**Objective of the module:**

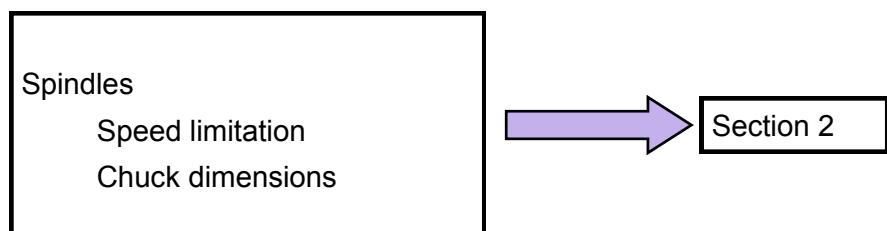
This module explains the function of the Spindles.

**Description of the module:**

The page “Spindles” allows the user to set data for the spindle (including the sub spindle if present).

**Content of the module:**

Spindles



## Section 3

Notes

### Spindles

#### 4.1 Selection menu "Spindles"

The menu "Spindles" can be selected as follows:

Press the Hotkey "Offset" on the CNC-full keyboard or on the OP (operating panel). This opens the tool management / Zero point offset display directly.

oder



On the OP press the key "Menu Select".

The following horizontal Softkey-strip of ShopTurn is displayed.



By pressing the HSK 5 "Tools WOs" select the range Tool management / Zero point offsets.

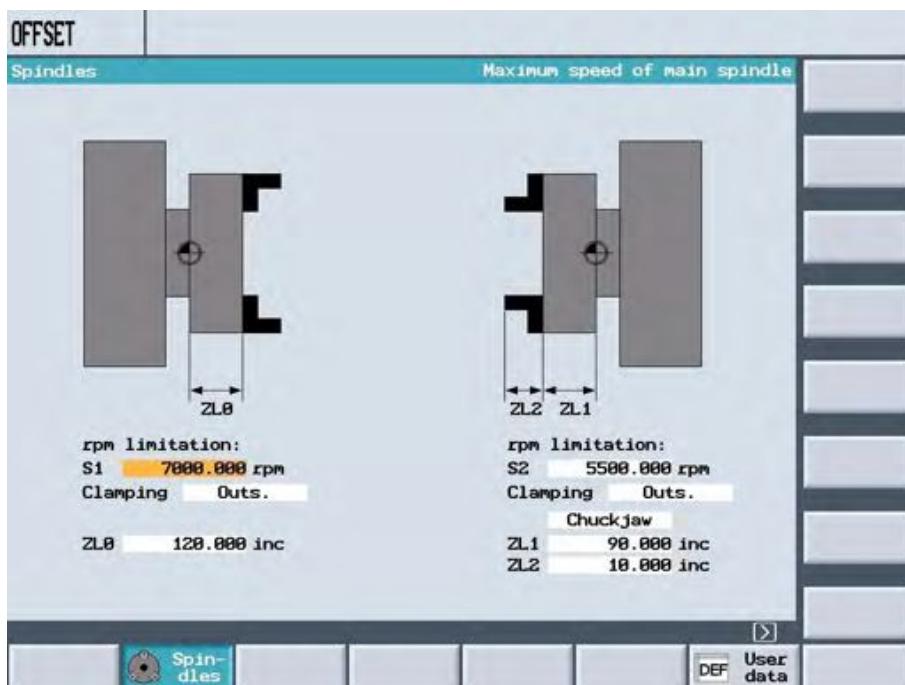


Expand the horizontal Softkey-strip.



Pressing the HSK 2 "Spindles" opens the menu.

The menu „Spindles“ is opened:

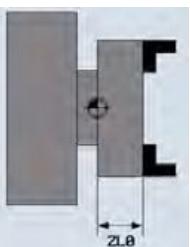
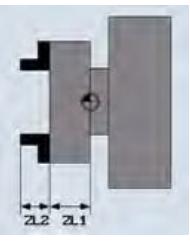


**Hint:**

The parameter "Clamping" is optional. The documentation by the machine manufacturer must be observed.

## Section 3

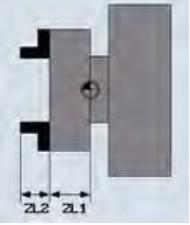
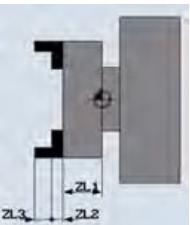
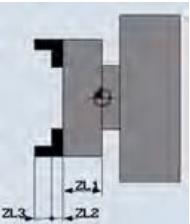
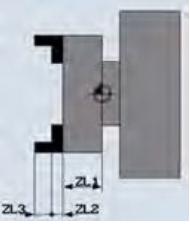
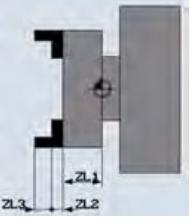
### Spindles

Parameter	Help picture	Description and hints	Notes
S1  rpm limitation: S1 0.000 rpm Clamping Outs.  ZL0 0.000 inc		Speed limitation for the main spindle [rev/min]	
Clamping  rpm limitation: S1 0.000 rpm Clamping Outs.  ZL0 0.000 inc		Main spindle: Work piece clamping internal or external	
ZL0  rpm limitation: S1 0.000 rpm Clamping Outs.  ZL0 0.000 inc		Main spindle chuck dimensions [inc]	
S2  rpm limitation: S2 0.000 rpm Clamping Outs. Chuckjaw ZL1 0.000 inc ZL2 0.000 inc		Speed limitation for the counter spindle [rev/min]  Hint: The index depends on the machine datum. The documentation by the machine manufacturer must be observed.	
Clamping  rpm limitation: S2 0.000 rpm Clamping Outs. Chuckjaw ZL1 0.000 inc ZL2 0.000 inc		Counter spindle:  Work piece clamping external or internal	

## Section 3

Notes

### Spindles

Parameter	Help picture	Description and hints
Jaw type	<p>rpm limitation: S2 0.000 rpm Clamping Outs. <b>Chuckjaw</b> ZL1 0.000 inc ZL2 0.000 inc</p>  	Dimensions of the front edge
ZL1	<p>rpm limitation: S2 0.000 rpm Clamping Outs. <b>Chuckjaw</b> <b>ZL1 0.000 inc</b> ZL2 0.000 inc ZL3 0.000 inc</p> 	Chuck dimensions for counter spindle [inc]
ZL2	<p>rpm limitation: S2 0.000 rpm Clamping Outs. <b>Chuckjaw</b> ZL1 0.000 inc <b>ZL2 0.000 inc</b> ZL3 0.000 inc</p> 	Setting dimensions of counter spindle [inc]
ZL3	<p>rpm limitation: S2 0.000 rpm Clamping Outs. <b>Chuckjaw</b> ZL1 0.000 inc ZL2 0.000 inc <b>ZL3 0.000 inc</b></p> 	Jaw dimensions of counter spindle - (only for dimensioning of the setting edge) [inc]

## 1 Brief description

**Objective of the module:**

This module will help to understand the structure of the program editor and the work plan in the operating area "Program".

**Description of the module:**

Apart from the manual operation area, ShopTurn provides the means of creating NC-programs on the control unit directly. These programs can be created as linked chain programs, G-Code programs or as a mix of the two.

This module shows the general layout of linked chain programs:

- the program editor (Depiction of work plan)
- the program editor (Depiction of program step editor) and
- the insertion of G-Code blocks in a work plan

The creation of G-Code-programs is described in the module *A138 "Operating area Program - Programming G-Code editor"*.

**Content of the module:**

- Structure of the program editors  
(depiction of work plan)
  - Pre-requisites for opening the program editors
  - Layout of the work plan in the program editor
  - Functions of the horizontal Softkey strip in the program editor
  - General functions for editing programs
- Structure of the program editors (depiction of program step editor)
- Insertion of G-Code blocks in step chain programs

## Operating Area - General Program structure

Structure of the program editors  
(depiction of work plan)

Pre-requisites for opening  
the program editors

Layout of the work plan in  
the program editor

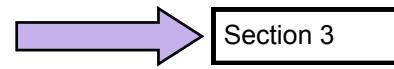
Functions of the horizontal  
Softkey strip in the program  
editor

General functions for editing  
programs



Section 2

Structure of the program editors  
(depiction of program step editor)



Section 3

Insertion of G-Code blocks in step  
chain programs



Section 4

## Section 2

### Structure of the program editors (depiction of work plan)

#### 2.1 Pre-requisites for opening the program editors

A new linked step program has been started, see module *A107 "Operating area Program Manager"*, Section 3 "NC-part program management"

An already existing linked step program is opened for editing, see module *A107 "Operating area Program Manager"*, Section 3 "NC-part program management"

##### Hint:

If the program editor is called up without these pre-requisites, there will be the message "Action not possible since no program is selected".

Notes

#### 2.2 Layout of the work plan in the program editor

After a new linked step program including the program head has been started or an existing linked step program has been opened, the work plan is shown as follows.



Example: Linked step program with stock removal and drilling cycle

- ① Program name
  - ② Work plan
- ① The program name is shown in the top line of the work plan window.

**HOLE\_PATTERN**

## Section 2

Notes

### Structure of the program editors (depiction of work plan)

(2)

#### Work plan

All program steps are listed in the work plan. It is subdivided into three sections:



#### Program head:

The program head contains the raw material dimensions for the simulation and parameters that are applicable to the whole of the program; e.g.

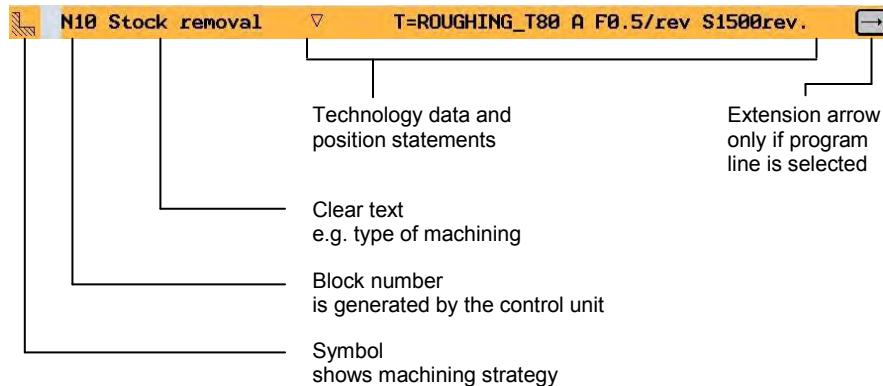
Zero point offset

Dimension units in mm or inch

Retraction plane, Tool changing point, Safety distance and Speed limitations

See module A112 - "Operating area Program - Program header"

#### Program block:



Extension arrow: With the cursor-key "Cursor to right" the program editor for the selected step (block) can be opened.

#### Interlinked machining:



The control links the technology blocks and position blocks or geometry blocks respectively automatically. These blocks are highlighted by a square bracket immediately next to the machining symbol from the start to the end of the linkage.

Technology blocks are blocks that describe the machining such as for instance Centring and Drilling.

Position blocks or geometry blocks respectively describe the positions where machining takes place, e.g. holes or geometries such as internal or external contours.

## Section 2

### Structure of the program editors (depiction of work plan)

#### 2.3 Functions of the horizontal Softkey strip in the program editor

This section lists the general functions available for programming under ShopTurn. The functions will then be described in the ensuing modules and explained by means of suitable examples.

Horizontal Softkey strip:



By pressing the HSK 1 "Straig. Circle" the functions for the programming of straights and circles will be called up.

*See Module A113 - "Operating area Program - Programming Straight/circle".*



By pressing the HSK 2 „Drilling“ the following drilling cycles will be called up:

- Concentric drilling
- Concentric tapping
- Drilling and reaming
- Deep hole drilling
- Thread cutting and milling
- Position patterns

*See Module A115 - "Operating area Program - Programming centring, drilling, Tapping and positions".*



By pressing the HSK 3 "Turning" the following cycles for turning are called up.

- Stock removal
- Turning of grooves
- Turning of relief cuts
- Cutting of threads
- Parting

*See Module A117 - "Operating area Program - Programming Turning Standard cycles".*



By pressing HSK 4 "Cont. turn." the cycles for the turning of complex contours are called up.

*See Module A119- "Operating area Program - Programming Contour turning".*



By pressing the HSK 3 "Milling" the following milling cycles are called up.

- Milling of pockets
- Milling of spigots
- Milling of slots
- Milling of multiple edges
- Milling of engravings
- Milling of contours

*See Module A121 - "Operating area Program - Programming Milling Standard cycles" and A123 - "Operating area Program - Programming Contour milling".*

Notes

## Section 2

### Structure of the program editors (depiction of work plan)

Notes



By pressing the HSK 5 “Various“ the following functions will be called up:

- Set marker
- Repeats
- Sub-program
- Counter spindle
- Settings
- Transformations (offsets, rotation, scaling, mirroring and rotation C-axis)

See Module A127 / A131 / A133 / A134



By pressing the HSK 7 “Simulation“ the programmed linked step can be simulated. The following perspectives are then available:

- 3-windows view
- Side elevation
- End elevation
- 3 dimensional view
- Details



By pressing the HSK 8 “Execute“ the programmed linked step can be selected and executed on the machine.

#### 2.4 General functions for editing programs



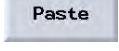
By pressing the key “Extension“ the following functions can be selected for editing the work planes:



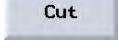
By pressing the VSK 1 “Mark“ one or more program steps can be marked, for instance for copying or cutting (deleting).



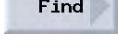
By pressing the VSK 2 “Copy“ one or more program steps can be copied.



By pressing the VSK 3 “Paste“ one or more marked program steps can be inserted behind the selected program step (current cursor position). The pasting is possible both in the original program as also in another linked step program.



By pressing the VSK 4 “Cut“ one / several program steps can be cut out or deleted. Cut program sections will be held in the intermediate memory and can be re-inserted with Softkey “Paste“ (see “Paste“).



By pressing the VSK 5 “Find“ it is possible to search for the contents of individual program steps.

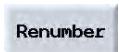
A window, in which the search statement can be entered, will be opened.

The search can be continued thereafter.

By pressing the VSK 7 “Renumber“ the program steps will be re-numbered.

The first block number (N5) and the spacing of the block numbers (in five's) are fixed for linked step programs.

The vertical Softkey strip for editing the program steps will be un-selected.



## Section 3

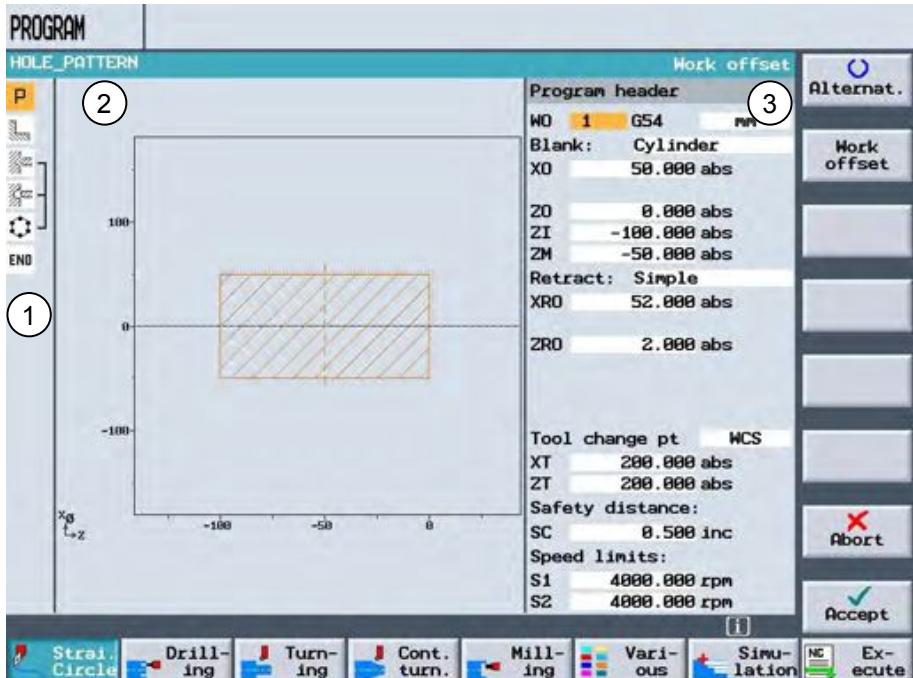
### Structure of the program editors (depiction of program step editor)

After selection of a program with the program manager, the individual program steps, for instance the program head, can be opened.



With the cursor-key "Cursor right" the program editor for the selected step can be opened, for instance for the program head.

Notes



Example: Program editor for the program head

- ① Symbolic depiction of the interlinked program steps
- ② Graphic range for the scaled depiction of the working step or help pictures  
In case of the scaled depiction the respective working plane will be shown including the scale.



Using the key "Help" it is possible to toggle between the scaled depiction of the working step and the display for static help pictures.

- ③ Input range for the functions / the cycle

## Section 4

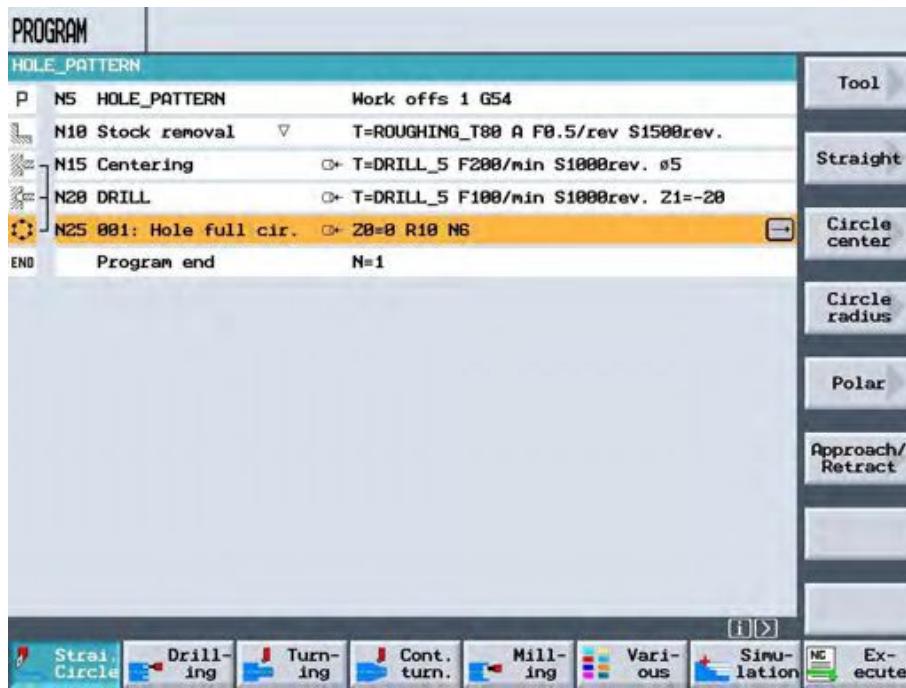
Notes

### Insertion of G-Code blocks in step chain programs

In the work plan G-Code-program steps can also be inserted alongside the ShopMill program.



Select the program step, after which the G-Code -program step is to be inserted, e.g. N25.



Insertion of an empty line. The number for the next block will automatically be added, e.g. N30.

Enter G-Code, e.g. G0 Z10 X100



## 1 Brief description

**Objective of the module:**

This module explains the layout and the parameters of the program header.

**Description of the module:**

A Program header and an End-of-program is automatically defined when a new program has been created. The parameters specified in the program header are valid for the entire program.

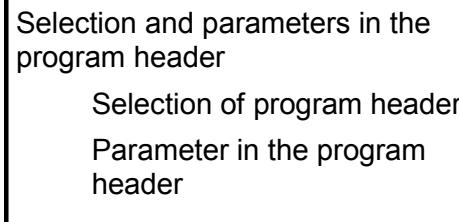
**Content of the module:**

Selection and parameters in the program header

- Selection of the program header
- Parameters in the program header

Vertical Softkey-strip

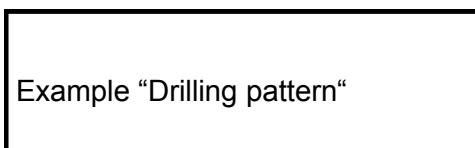
Example „Drilling pattern“



Section 2



Section 3



Section 4

## Section 2

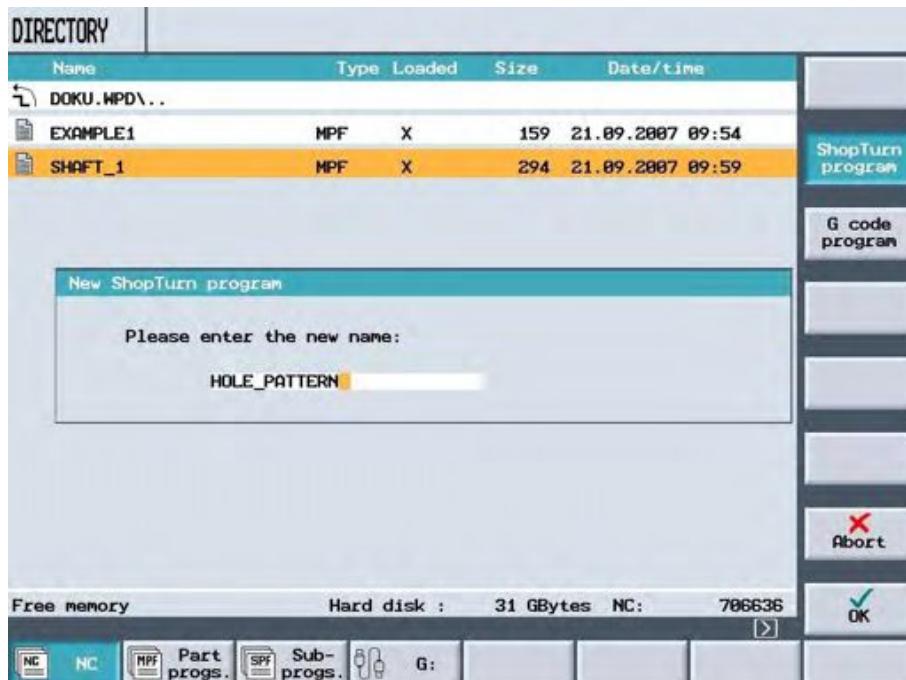
Notes

Selection and parameters in the program header

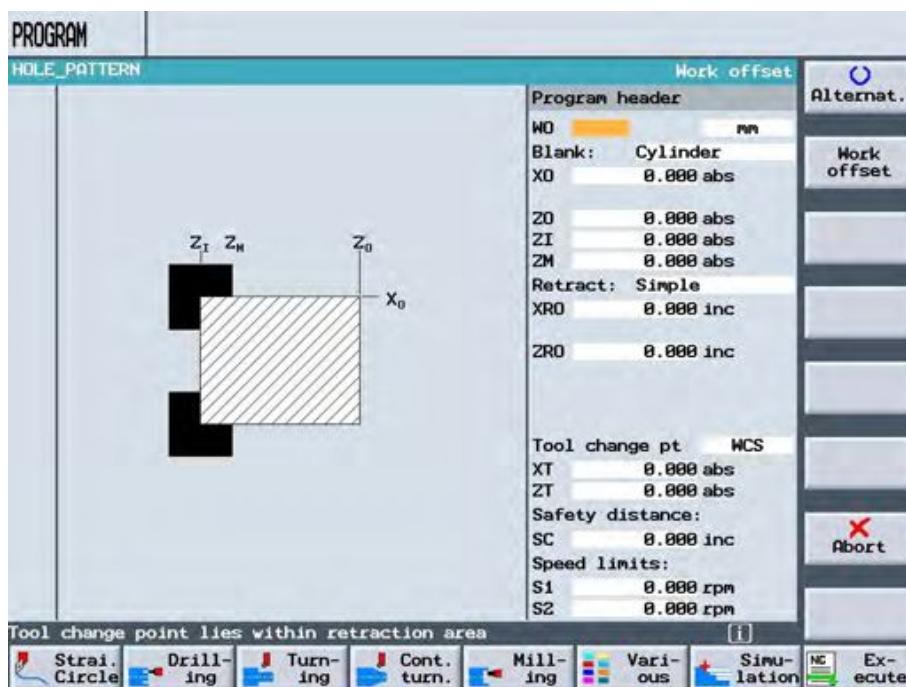
### 2.1 Selection Program header

Start a new linked chain program (ShopTurn-Program) in the program manager for instance with the name "Hole pattern".

See Module A107 - "Operating area Program manager", Section 3 "NC-part program management", Sub-section 3.2 "Making up a new NC-Part program".



Accept existing program name. The program editor (Image: Editor Program step) appears.



## Section 2

### Selection and parameters in the program header

#### 2.2 Parameter in the program header

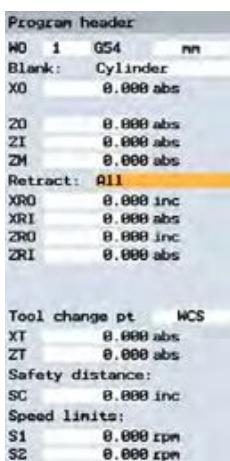
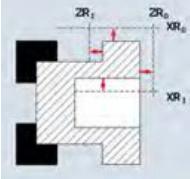
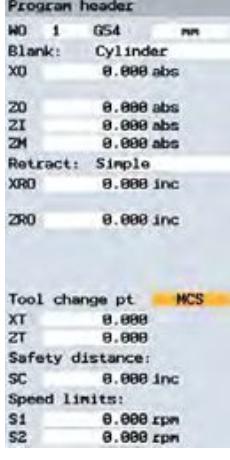
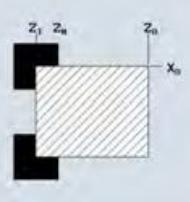
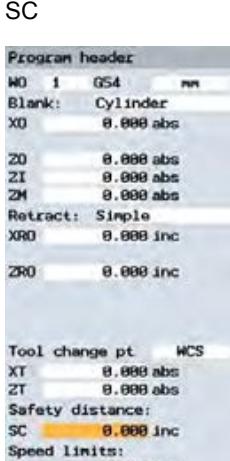
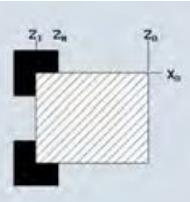
Notes

Parameter	Help picture	Description and hints
WO  Program header: W0 1 G54 mm Blank: Cylinder X0 0.000 abs  Z0 0.000 abs Z1 0.000 abs ZM 0.000 abs Retract: Simple XRO 0.000 inc  ZRO 0.000 inc  Tool change pt WCS XT 0.000 abs ZT 0.000 abs Safety distance: SC 0.000 inc Speed limits: S1 0.000 rpm S2 0.000 rpm		Zero point offset  Either the basic ZPO or the ZPOs 1 - 4 can be selected.  The input of further ZPOs depends on the machine configuration. The documentation by the machine manufacturer must be observed.
Dimension unit  Program header: W0 1 G54 mm Blank: Cylinder X0 0.000 abs  Z0 0.000 abs Z1 0.000 abs ZM 0.000 abs Retract: Simple XRO 0.000 inc  ZRO 0.000 inc  Tool change pt WCS XT 0.000 abs ZT 0.000 abs Safety distance: SC 0.000 inc Speed limits: S1 0.000 rpm S2 0.000 rpm		mm Millimetre in INCH  The chosen dimension unit is valid for the entire program.
Raw material  Program header: W0 1 G54 mm Blank: Cylinder X0 0.000 abs  Z0 0.000 abs Z1 0.000 abs ZM 0.000 abs Retract: Simple XRO 0.000 inc  ZRO 0.000 inc  Tool change pt WCS XT 0.000 abs ZT 0.000 abs Safety distance: SC 0.000 inc Speed limits: S1 0.000 rpm S2 0.000 rpm	  	The raw material is defined by way of shape (cylinder, tube, rectangle, N-cornered) and the dimensions [abs].  W: Raw material width - only rectangles L: Raw material length - only rectangles N: NR of corners - only N-cornered L: Edge length - only N-cornered SW: A/C flats - only N-cornered  XI: Internal diameter (abs or inc) - only for tube  XA: External diameter (abs) - only for cylinder and tube ZA: Original dimension (abs) ZI: End dimension (abs or inc) ZB: Machining dimension (abs or inc)

## Section 2

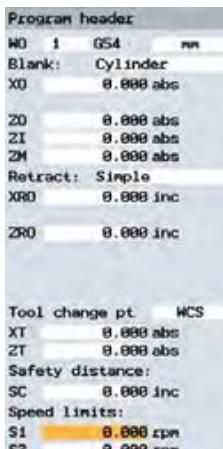
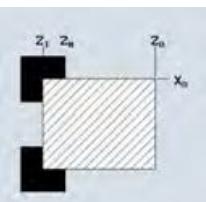
### Selection and parameters in the program header

Notes

Parameter	Help picture	Description and hints
<b>Retraction</b> 		Type of retractions: simple, extended, all The retraction range marks the range outside of which unlimited traversing of the axes is possible without danger of collision. XRA: outer retraction level in X-direction (abs) XRI: inner retraction level in X-direction [abs, inc] ZRA: outer retraction level in Z-direction [abs] ZRI: inner retraction level in Z-direction [abs, inc]
<b>Tool changing point</b> 		Tool changing point The turret traverses such that the reference point is at the tool changing point and indexes the required tool to the machining position. The changing point must be far enough back from the retraction plane such that no tool reaches into the retraction range during indexing. It is possible to specify the current tool position as the tool changing position (teaching tool changing point) or to enter the coordinates of the tool changing point XT and ZT directly into the parameter mask. The teaching of the tool changing point is possible only if the machine co-ordinate system (MCS) is selected.
<b>SC</b> 		Safety distance The safety distance defines as to how closely the tool is permitted to approach the work piece. It must lie within the retraction plane. [inc]

## Section 2

### Selection and parameters in the program header

Parameter	Help picture	Description and hints	Notes
S1, S2    Tool change pt    WCS XT    0.000 abs ZT    0.000 abs Safety distance: SC    0.000 inc Speed limits: S1    0.000 rpm S2    0.000 rpm		To ensure that when machining with constant cutting speed (the spindle speed depends on the work piece diameter) the spindle speed does not exceed permissible values, it is possible to limit the spindle speed by means of parameter S1 for the main spindle and S2 for the Counter spindle. The spindle speed limit applies also for absolute programmed spindle speeds.	

All parameters specified in the program header, with the exception of the raw material shape and the dimensional units, can be altered anywhere in the program. Additionally there is the possibility to change the basic settings for the sense of rotation for the milling. The way to change these settings is described in the *Module A133 - “Operating area Program editor - Programming Various Settings”*.

## Section 3

### Vertical Softkey-strip



If a parameter has been selected, for which other alternative inputs are possible, these can be called up by pressing the VSK 1 “Alternat.”.



By pressing the VSK 2 “Work offset“ the menu “Zero point offsets“ will be called up.



By pressing the VSK 7 “Abort“, the input for the program header will be terminated.



By pressing the VSK 8 “Accept.“, the program header will be accepted for the new program.

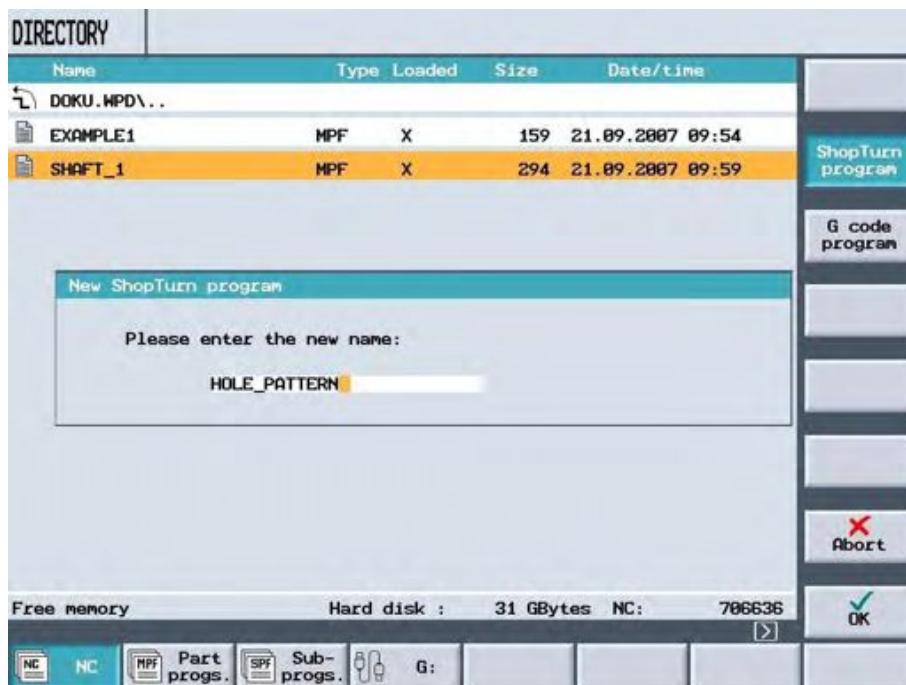
## Section 4

Notes

Example “Drilling pattern“

Start a new linked chain program (ShopTurn-program) in the program manager for instance with the name “Hole pattern“.

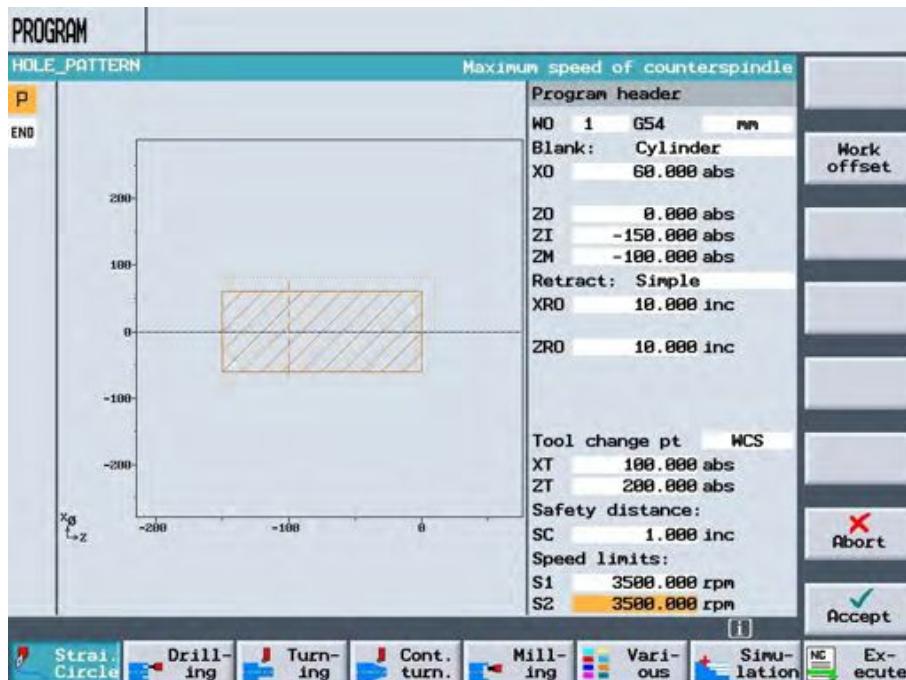
See Module A107 - “Operating area Program manager“, Section 3 “NC-part program management“, Sub-section 3.2 “Making up a new NC-Part program“.



OK

Press to accept the program name. The program editor (View: Editor program step) will be opened.

Enter the parameter.



Accept

Completion of the input for the program header. The program header will be transferred to the chain-link program and the display changes to the work plan.

## 1 Brief description

**Objective of the module:**

This module shows how to program straight and circular path motions.

**Description of the module::**

This functions is intended for the straight forward machining of straight and circular paths. Complex types of machining, such as contours and tangential transitions, should be machined by use of the functions "Contour turning" and "Stock removal" - see *module A117 "Operating area Program - Programming of turning standard cycles"* and *A1219 "Operating area Program - Programming of contour turning"*.

**Inhalt des Moduls:**

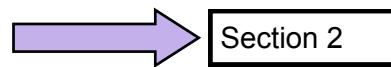
- Pre-selection and general functions under "Straight/Circle"
  - Pre-selection of the function "Straights/Circles"
  - Vertical Softkey strip
- Tool selection under Straights/Circles
- Programming of a straight
- Programming of a circle by use of the centre
- Programming of a circle by use of the radius
- Programming by use of polar co-ordinates
  - Vertical Softkey strip
  - Pole
  - Straight polar
  - Circular polar
- Programming of an approach/retraction cycle

# Operating Area Program Editor- Programming Straights/Circles

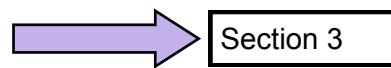
Pre-selection and general  
functions under "Straight/Circle"

Pre-selection of the function  
"Straight/Circle"

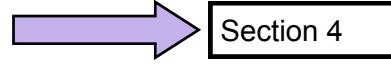
Vertical Softkey strip

**Section 2**

Tool selection under Straights/  
Circles

**Section 3**

Programming of a straight

**Section 4**

Programming of a circle by use of  
the centre

**Section 5**

Programming of a circle by use of  
the radius

**Section 6**

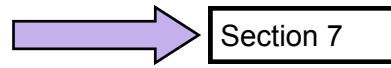
Programming by use of polar co-  
ordinates

Vertical Softkey strip

Pole

Straight polar

Circular polar

**Section 7**

Programming of an approach/  
retraction cycl

**Section 8**

## Section 2

### Pre-selection and general functions under „Straight/Circle“

#### 2.1 Pre-selection of the function “Straight/Circle”

After establishing the program head, the program blocks for the machining can be programmed, for instance, for straight and circular motions.



Pressing the HSK 1 "Straight/Circle" selects the range for the programming of straights and circular arcs.



Notes

## Section 2

Notes

Pre-selection and general functions under „Straight/Circle“

### 2.2 Vertical Softkey strip

Tool

Pressing the VSK 1 "Tool" selects the function tool selection.

See *Section 3 "Tool selection under Straights/Circles"*.

Straight

Pressing the VSK 2 "Straight" selects the function "Straight with destination point".

See *Section 4 "Programming of a straight"*.

Circle center

Pressing the VSK 3 "Circle centre" selects the function „Circle with destination and circle centre point and co-ordinates of circle centre“.

See *Section 5 "Programming of a circle by use of the centre"*.

Circle radius

Pressing the VSK 4 "Circle radius" selects the function "Circle with destination point co-ordinates and circle radius".

See *Section 6 "Programming of a circle by use of the radius"*.

Polar

Pressing the VSK 5 "Polar" selects the function „Straight/Circle with destination point co-ordinates and angle specification“.

The following functions are available:

- Pole specification
- Straight polar
- Circular polar

See *Section 7 "Programming by use of polar co-ordinates"*.

Approach/  
Retract

By pressing the VSK 6 "Approach/retract" the window for the programming of an Approach/retraction cycle will be called up.

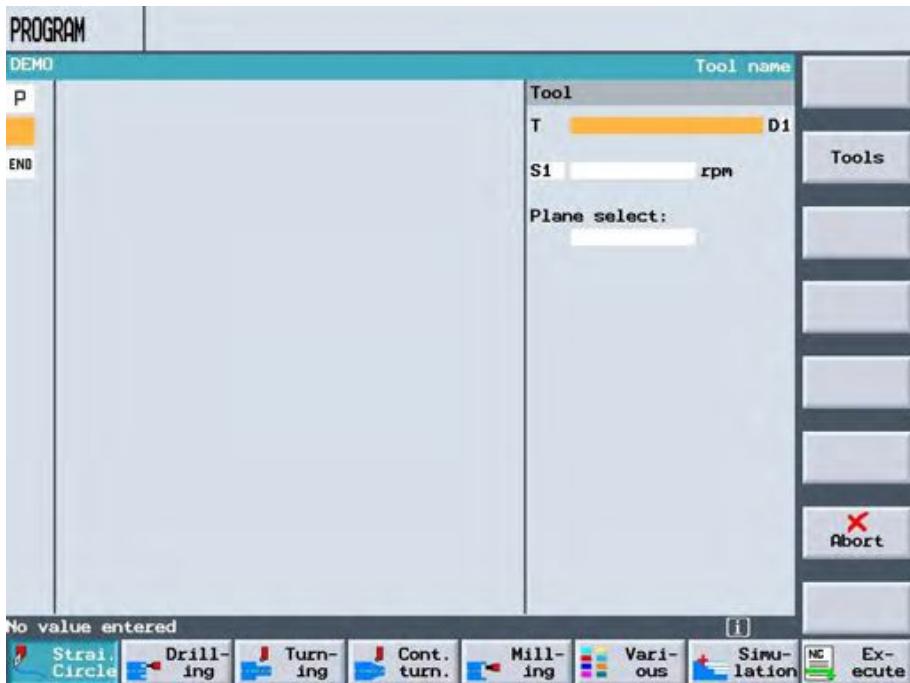
## Section 3

### Tool selection under Straights/Circles

Tool

Pressing the VSK 1 "Tools" opens the window for the programming of tools will be displayed.

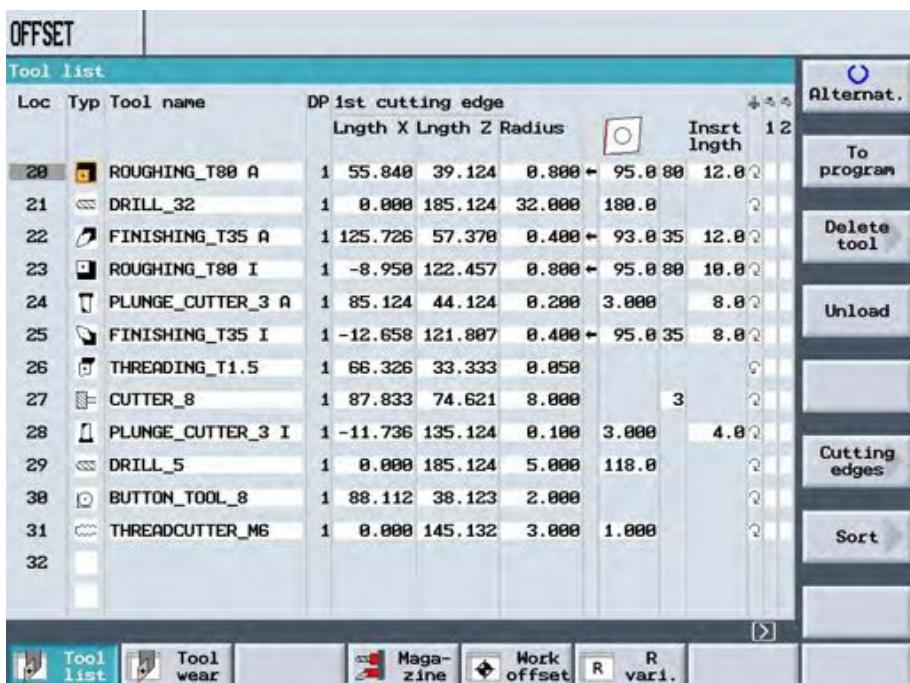
Notes



Vertical Softkey strip:

Tools

By pressing the VSK 2 "Tools" the tool management will be opened. A tool can be transferred to the program.



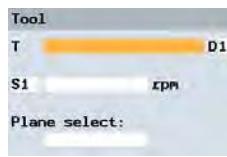
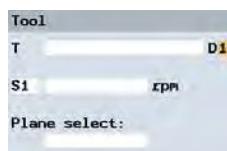
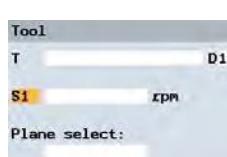
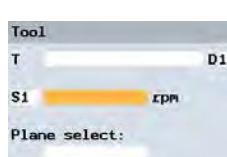
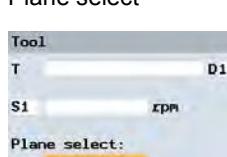
To program

By pressing the VSK 2 "To program" the chosen tool (current cursor position) is transferred to the program.

## Section 3

Notes

### Tool selection under Straights/Circles

Parameter	Help picture	Description and hints
T		Tool name
D		Cutting edge number
Spindle		Selection between main spindle, tool spindle or Counter spindle
Spindle speed		Speed [rpm] alternative Cutting speed [mm/min]
Plane select		Selection between the machining planes Turning, Face/face C, Perimeter/Perimeter C, Face Y and Perimeter Y  Additionally: for Perimeter/Perimeter C: Cylinder diameter, for Face Y: Positioning angle for the machining range CP, for Perimeter Y: Reference point C0

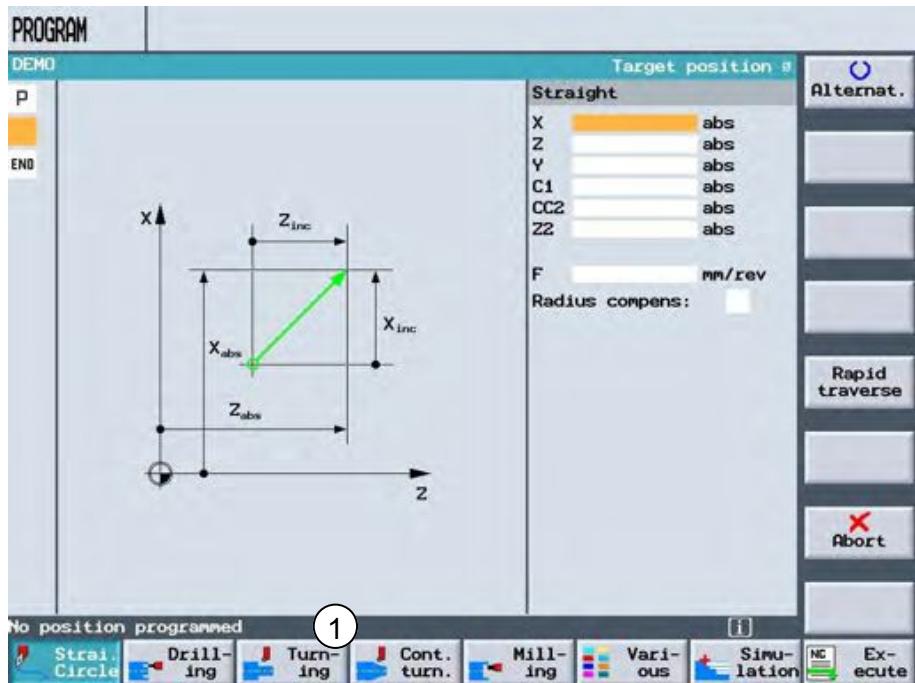
## Section 4

### Programming of a straight

Straight

Pressing the VSK 2 "Straight" opens the window for programming straights.

Notes



- ① This message will disappear after a destination point has been entered.

#### Hint:

The tool traverses at the programmed feed rate or with rapid traverse from the present position to the programmed destination point.

## Section 4

### Programming of a straight

Notes

Parameter	Help picture	Description and hints
X		Destination point X-axis [abs, inc]
Z		Destination point Y-axis [abs, inc]
Further axes		Depending on the machine configuration further axes can be shown. Please refer to the documentation by the machine manufacturer.
F		Feed rate [mm/tooth, mm/min, mm/rev]  By pressing the VSK 5 „Rapid traverse“ the rapid traverse rate will be transferred to the parameter field „F“.
Radius compens		Statement as to which side of the contour in the direction of traverse the milling cutter passes. Alternatively:   Radius correction to left of contour  Radius correction to right of contour  Radius correction OFF  The selected radius correction remains activated as specified beforehand.

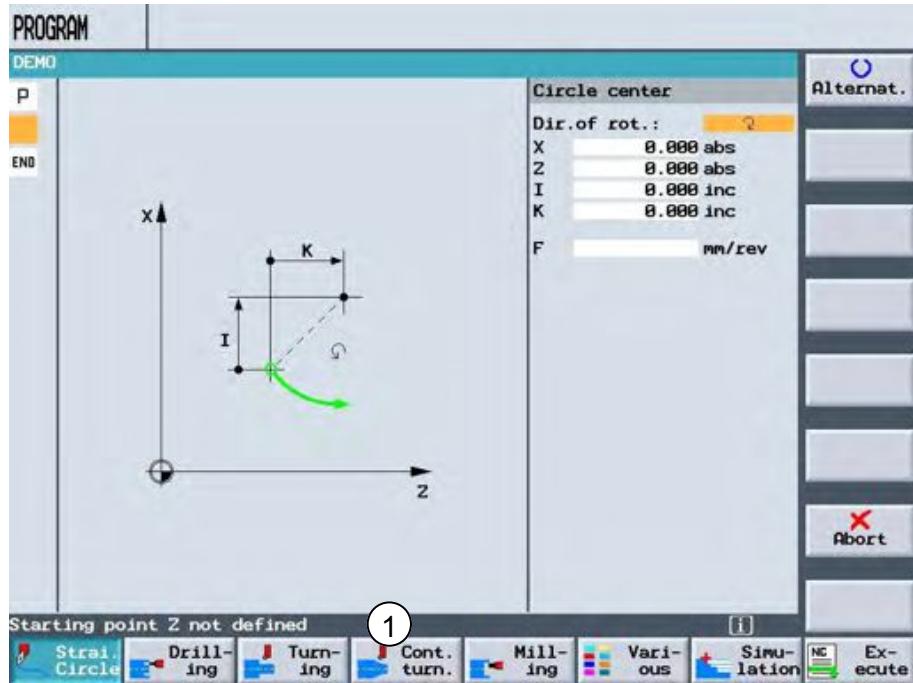
## Section 5

### Programming of a circle by use of the centre

Circle center

Pressing the VSK 3 "Circle centre" opens the window for the programming of circles by statement of the centre point co-ordinates and the circle end point/circle destination point.

Notes



- ① Before the entry of any circle parameters, a starting point must be specified, for instance by the input of a straight.

#### Hint:

The tool moves along a circular path from the current position to the programmed circle destination point.

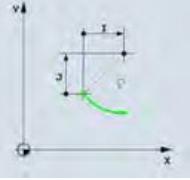
The position of the centre point must be known. Only the machining feed rate can be used. A tool must be programmed before any circular motion ensues.

## Section 5

### Programming of a circle by use of the centre

Notes

Parameter	Help picture	Description and hints
Direction of rotation		The motion takes place in the programmed direction from the circle starting point to the destination point. This direction can be programmed either clockwise or anti-clockwise.
F		Feed rate [mm/tooth, mm/min, mm/rev]

Face/Face C / Face Y:		see plane selection in Section 3)
X		Destination point X-axis [abs, inc]
Y		Destination point Y-axis [abs, inc]
I		Distance of the circle starting point to the centre point in the X direction [inc]
J		Distance of the circle starting point to the centre point in the Y direction [inc]

## Section 5

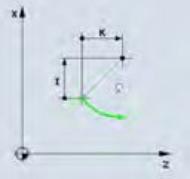
### Programming of a circle by use of the centre

Parameter	Help picture	Description and hints	Notes
Perimeter/Perimeter C / Perimeter Y:		(see plane selection in Section 3)	
Y		Destination point Y-axis [abs, inc]	
Z		Destination point Z-axis [abs, inc]	
J		Distance of the circle starting point to the centre point in the Y direction [inc]	
K		Distance of the circle starting point to the centre point in Z direction [inc]	

## Section 5

### Programming of a circle by use of the centre

Notes

Parameter	Help picture	Description and hints
Turning:		(see plane selection in Section 3)
X	 <b>Circle center</b> Dir.of rot.: <input checked="" type="radio"/> X 0.000 abs Z 0.000 abs I 0.000 inc K 0.000 inc F mm/min	Destination point X-axis [abs, inc]
Z		Destination point Z-axis [abs, inc]
I		Distance of the circle starting point to the centre point in the X direction [inc]
K		Distance of the circle starting point to the centre point in the Z direction [inc]

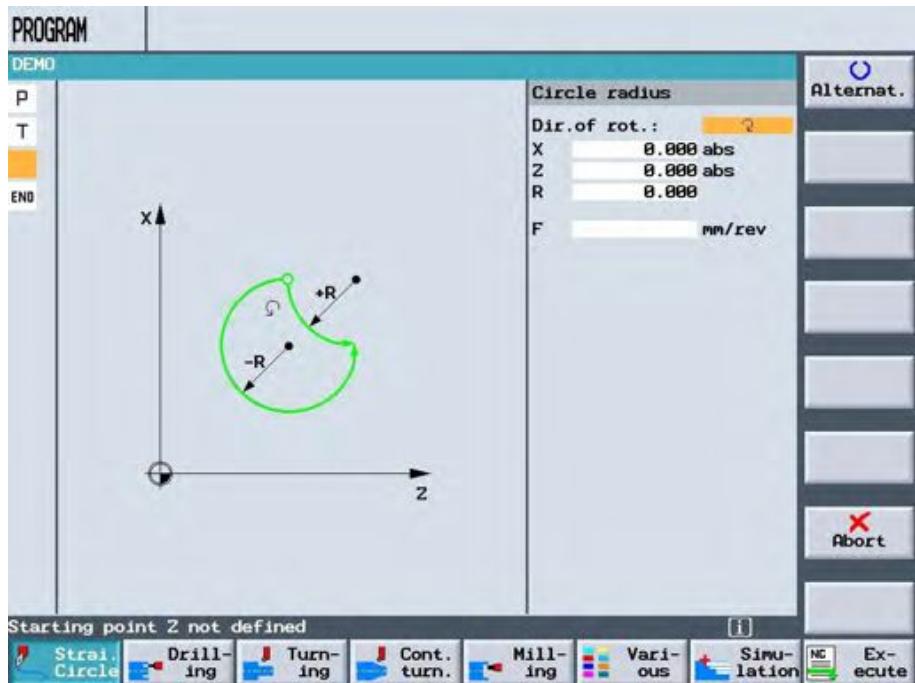
## Section 6

### Programming of a circle by use of the radius

Circle  
radius

Pressing the VSK 4 "Circle radius" opens the window for the programming of circles by statement of the radius and the circle end point/circle destination point.

Notes



#### Hint:

The tool moves along a circular path with the programmed radius from the current position to the programmed circle destination point. The position of the centre point must be known. Only the machining feed rate can be used. A tool must be programmed before any circular motion ensues.

## Section 6

### Programming of a circle by use of the radius

Notes

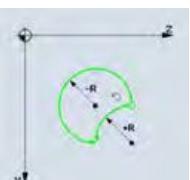
Parameter	Help picture	Description and hints
Direction of rotation		The motion takes place in the programmed direction from the circle starting point to the destination point. This direction can be programmed either clockwise or anti-clockwise.  In Uhrzeigersinn  Gegen Uhrzeigersinn
F		Feed rate [mm/tooth, mm/min, mm/rev]

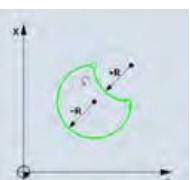
Face/Face C / Face Y:	(see plane selection in Section 3)	
X		Destination point X-axis [abs, inc]
Y		Destination point Y-axis [abs, inc]
R		Radius of the circular arc The selection of the required arc is achieved by the input of a positive or negative sign.

## Section 6

### Programming of a circle by use of the radius

Notes

Parameter	Help picture	Description and hints
Perimeter/Perimeter C / Perimeter Y:		(see plane selection in Section 3)
Y	 <b>Circle radius</b> Dir.of rot.: Q Y 0.000 abs Z 0.000 abs R 0.000 F mm/rev	Destination point Y-axis [abs, inc]
Z		Destination point Z-axis [abs, inc]
R		Radius of the circular arc  The selection of the required arc is achieved by the input of a positive or negative sign.

Turning:		(see plane selection in Section 3)
X	 <b>Circle radius</b> Dir.of rot.: Q X 0.000 abs Z 0.000 abs R 0.000 F mm/rev	Destination point X-axis [abs, inc]
Z		Destination point Z-axis [abs, inc]
R		Radius of the circular arc  The selection of the required arc is achieved by the input of a positive or negative sign.

## Section 7

### Programming by use of polar co-ordinates

Notes

Polar

Pressing the VSK 5 "Polar" opens the window for programming the polar co-ordinate.

If the drawing is dimensioned from a central point (pole) with radius and angles, they can be programmed as polar co-ordinate.

Straights and circles can be programmed by use of polar co-ordinates.



#### 7.1 Vertical Softkey strip

Pole

Pressing the VSK 1 "Pole" opens the window for programming the reference point.

See Section 7.1 "Programming Pole".

Straight polar

Pressing the VSK 2 "Straight polar" opens the window for programming a straight by stating the length and the angle.

See Section 7.2 "Straight polar".

Circle polar

Pressing the VSK 3 "Circular polar" opens the window for the programming of a circle by stating the direction of rotation and an angle.

See Section 7.3 "Circular polar".

## Section 7

### Programming by use of polar co-ordinates

#### 7.2 Pole

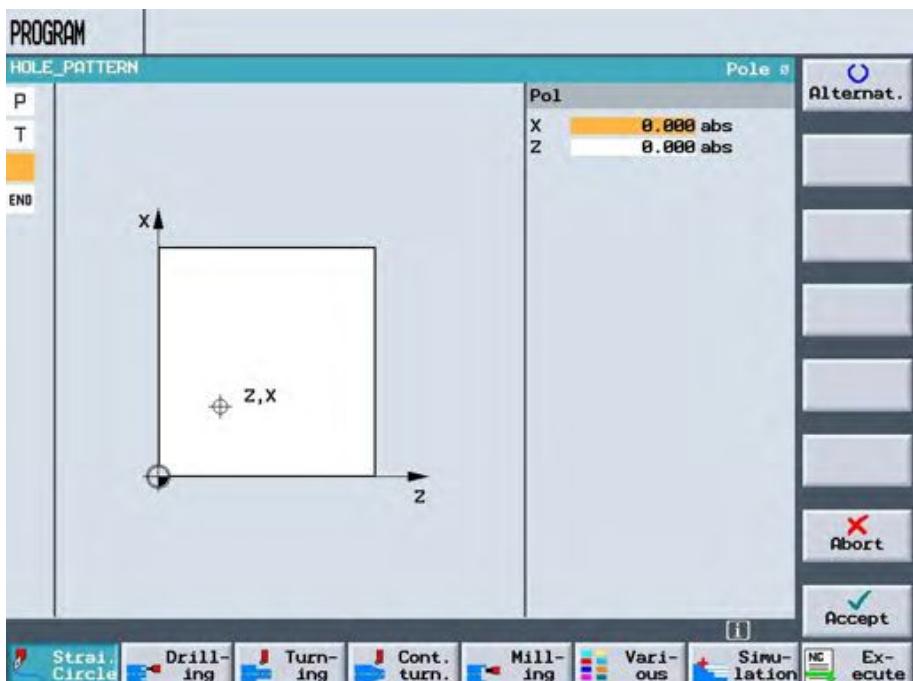
Before programming a straight or a circle in polar co-ordinates the pole must be defined. This pole is the reference point of the polar co-ordinate system.

Following this the angle for the first straight or the first circle must be programmed in absolute co-ordinates.

The angles of any further straights or circular arcs may then be programmed either absolute or incremental.

Pole

Pressing the VSK 1 "Pole" opens the window for programming the pole (reference point).



Parameter	Help picture	Description and hints
Face/Face C / Face Y:	(see plane selection in Section 3)	
X  Pol X 0.000 abs Y 0.000 abs		Position of Pole (reference point) X-axis [abs, inc]  Hint: The pre-selection of "inc" is possible only if previously a starting point has been specified as absolute.
Y  Pol X 0.000 abs Y 0.000 abs		Position Pole (reference point) Y-axis [abs, inc]  Hint: The pre-selection of "inc" is possible only if previously a starting point has been specified as absolute.

Notes

## Section 7

### Programming of a circle by use of the radius

Notes

Parameter	Help picture	Description and hints
Perimeter/Perimeter C / Perimeter Y:		(see plane selection in Section 3)
Y  Pol Y <b>0.000 abs</b> Z <b>0.000 abs</b>		Destination point Y-axis [abs, inc]
Z  Pol Y <b>0.000 abs</b> Z <b>0.000 abs</b>		Destination point Z-axis [abs, inc]

Turning:		(see plane selection in Section 3)
X  Pol X <b>0.000 abs</b> Z <b>0.000 abs</b>		Destination point X-axis [abs, inc]
Z  Pol X <b>0.000 abs</b> Z <b>0.000 abs</b>		Destination point Z-axis [abs, inc]

## Section 7

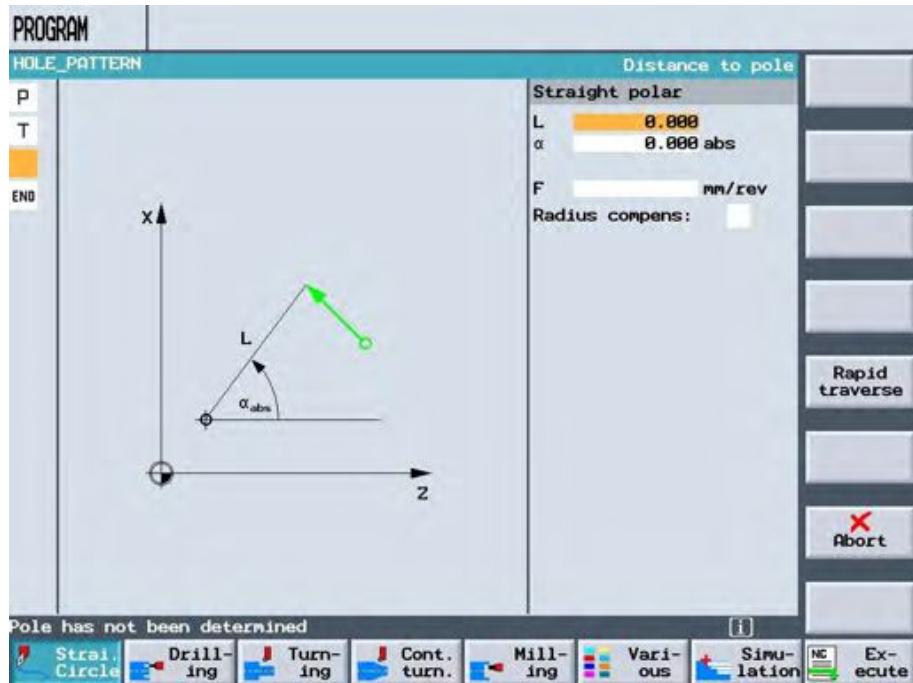
### Programming by use of polar co-ordinates

#### 7.2 Straight polar

A straight in the polar co-ordinate system is defined by a distance to the reference point (L) and an angle ( $\alpha$ ).

Straight polar

Pressing the VSK 2 "Straight polar" opens the window for programming of a straight by stating the length and an angle.



#### Hint:

The tool moves from the present position along a straight line to the programmed destination point either at feed rate or with rapid traverse.

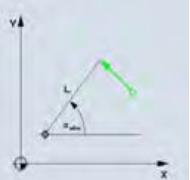
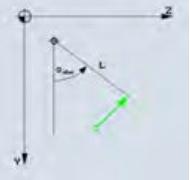
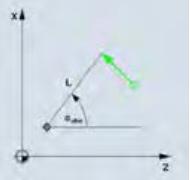
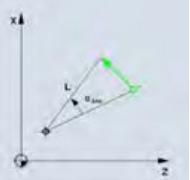
The first straight in polar co-ordinates after the pole has been defined must be programmed with an absolute angle. All further straights or arcs can also be programmed incrementally.

Notes

## Section 7

### Programming by use of polar co-ordinates

Notes

Parameter	Help picture	Description and hints
L		Distance from the pole to the destination point of the straight
$\alpha$		Polar angle First straight [abs] All further elements [abs, inc, positive, negative]
F		Feed rate [mm/tooth, mm/min, mm/rev] By pressing the VSK 5 "Rapid traverse" transfers the rapid traverse rate to the parameter field "F".
Radius compens		Statement as to which side of the contour in direction of motion the milling cutter is to be positioned. Alternatively:   Radius correction to the left of contour  Radius correction to the right of contour  Radius correction OFF  Selected radius correction remains as previously selected.

## Section 7

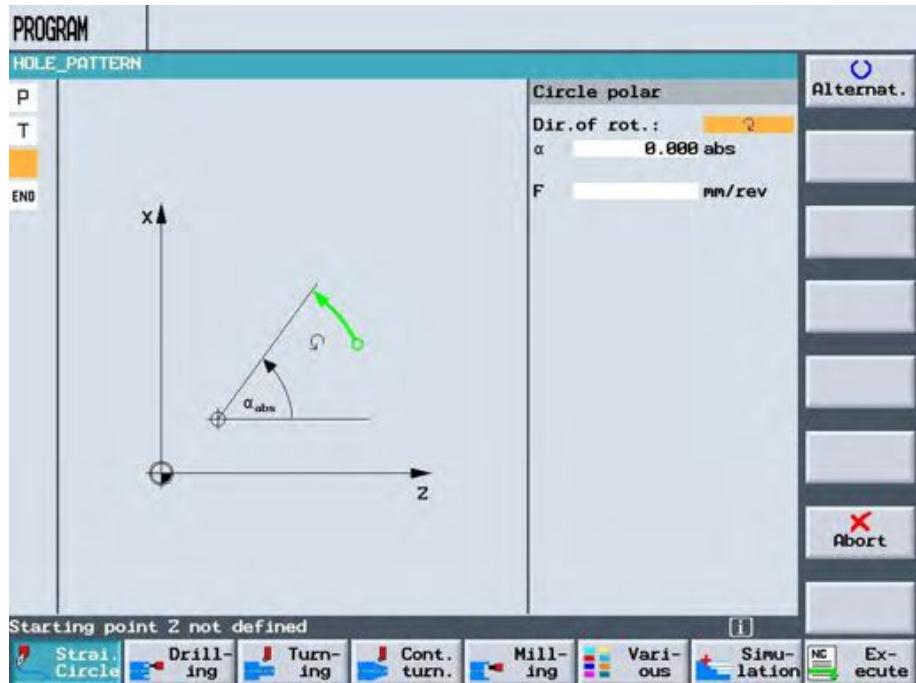
### Programming by use of polar co-ordinates

#### 7.3 Circular polar

A circular arc in the polar co-ordinate system is defined by an angle ( $\alpha$ ).

Circle  
polar

Pressing the VSK 3 "Circle polar" opens the window for programming of a circular arc by stating the direction of rotation and the polar angle.



#### Hint:

The tool moves from the present position along a circular arc to the programmed destination point (angle) at feed rate. The radius is given by the distance between the defined pole and the present position, i.e. arc starting point and arc destination point have the same distance from the pole.

The first arc can only commence at the end of a previously programmed straight.

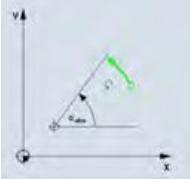
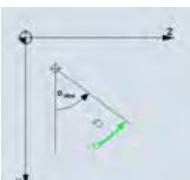
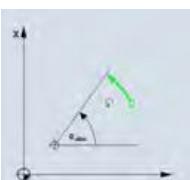
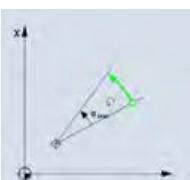
The first polar arc in polar co-ordinate after defining the pole must be programmed with an absolute angle. All further straight or arcs can also be programmed incrementally.

Notes

## Section 7

### Programming by use of polar co-ordinates

Notes

Parameter	Help picture	Description and hints
Direction of rotation	 	<p>The motion takes place in the programmed direction from the circle starting point to the destination point. This direction can be programmed either clockwise or anti-clockwise.</p> <p> Clockwise</p> <p> Anti-clockwise</p>
Polar angle		<p>First arc [abs]</p> <p>All further elements [abs, inc, positive, negative]</p>
F	 	<p>Feed rate</p> <p>[mm/tooth, mm/min, mm/rev]</p>

## Section 8

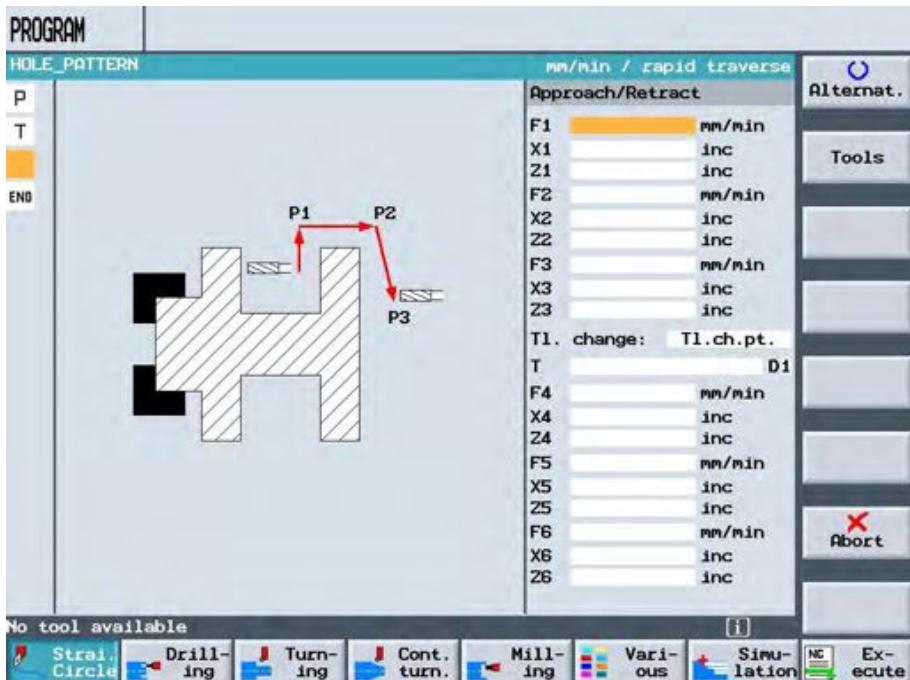
### Programming of an approach/retraction cycle

Approach/  
Retract

By pressing the VSK 6 "Approach/retract" the window for programming an approach/retraction cycle will be called up.

If the approach or retraction to or from a machining cycle is to be abbreviated or if a difficult geometrical situation should require a solution for the approach or the retraction, a special cycle can be used. ShopTurn in that case disregards the normal approach/retraction strategy (see Section "Approach/retraction to and from a machining cycle").

Notes



The approach/retraction cycle can be inserted between any two ShopTurn program blocks, however, not between linked program blocks. The origin for the approach/retraction cycle is always the safety distance that was operative for the preceding machining operation. If a tool change is to be carried out the approach of the tool changer can take place via a max. of three positions (P1 to P3) and the return to the next starting point via a max. of another 3 positions (P4 to P6). If there is no tool change a max. of 6 positions are available for the approach of the next starting point. If these 3, respectively 6 positions are not enough, the cycle can be called up repeatedly one after the other in order to programme further positions.

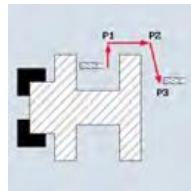
#### Caution

The tool moves directly from the last programmed position in the approach/retraction cycle to the start point for the next machining step.

## Section 8

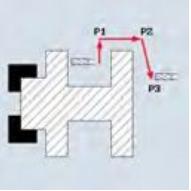
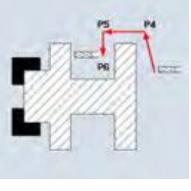
### Programming of an approach/retraction cycle

Notes

Parameter	Help picture	Description and hints
F1, F2, F3  	<p>Feed rate for the approach of the first position Alternatively rapid traverse,</p> <p>Feed rate for the approach of the second position Alternatively rapid traverse,</p> <p>Feed rate for the approach the third position Alternatively rapid traverse,</p>	
X1, X2, X3 Z1, Z2, Z3  	<p>X-value of 1st, 2nd or. 3rd position, Z- value of the 1st, 2nd, or 3rd position</p>	
Tl. change  	<p>Tl.ch.pt.: Approach the tool changing point from the last programmed position and carry out tool change.</p> <p>Directly: Carry out tool change at the last programmed position and not at the tool changing point,</p> <p>No: No tool change</p>	

## Section 8

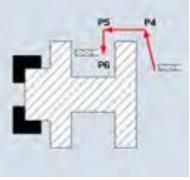
### Programming of an approach/retraction cycle

Parameter	Help picture	Description and hints	Notes
T		Tool name - not in case of "tool change" "NO"	
D		Cutting edge number - not in case of „tool change“ "NO"	
F4, F5, F6		Feed rate for the approach the fourth position Alternatively Rapid traverse, Feed rate for the approach of the fifth position Alternatively Rapid traverse, Feed rate for the approach of the sixth position Alternatively Rapid traverse,	

## Section 8

### Programming of an approach/retraction cycle

Notes

Parameter	Help picture	Description and hints
X4, X5, X6 Z4, Z5, Z6		X-value of the 4th, 5th and 6th position Z-value of the 4th, 5th and 6th position [abs, inc]

## 1 Brief description

**Objective of the module:**

This module explains the functions and programming of drilling and tapping, as a single function or patterns.

**Description of the module:**

This module describes the functions of centring, drilling, deep hole drilling, boring, tapping as well as the associated programming as single operations or patterns.

When programming holes and tapped threads the technology blocks must be entered first of all as they occur in the machining sequence, e.g.:

- centring
- deep hole drilling
- tapping

These are followed by the position details. Several position patterns are available for this.

The sequence, firstly technology block followed by position block, must be adhered to under all circumstances in case of drilling cycles.

**Content of the module:**

Selection and general functions for "Drilling"

- Selection of the function "Drilling"
- Vertical Softkey strip

Drilling centric

Thread centric

Drilling / Reaming

- Centring
- Drilling
- Reaming

Deep hole drilling

Thread cutting

- Vertical Softkey strip
- Tapping
- Thread milling

# Operating Area Program editor - Programming centring, drilling, thread cutting, positions

Position patterns

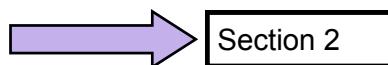
- Vertical Softkey strip
- Rectangular / polar positions
- Line, grid
- Full and partial circles

Repeat positions

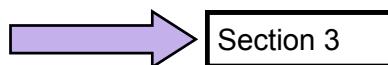
Selection and general functions  
under "Drilling"

Selection of function  
"Drilling"

Vertical Softkey strip



Drilling centric



Thread centric



Drilling / Reaming

Centring

Drilling

Reaming



Deep hole drilling



Thread cutting

Vertical Softkey strip

Tapping

Thread milling



Position patterns

Vertical Softkey strip

Rectangular / polar positions

Line, grid

Full and partial circle



Repeat positions



## Section 2

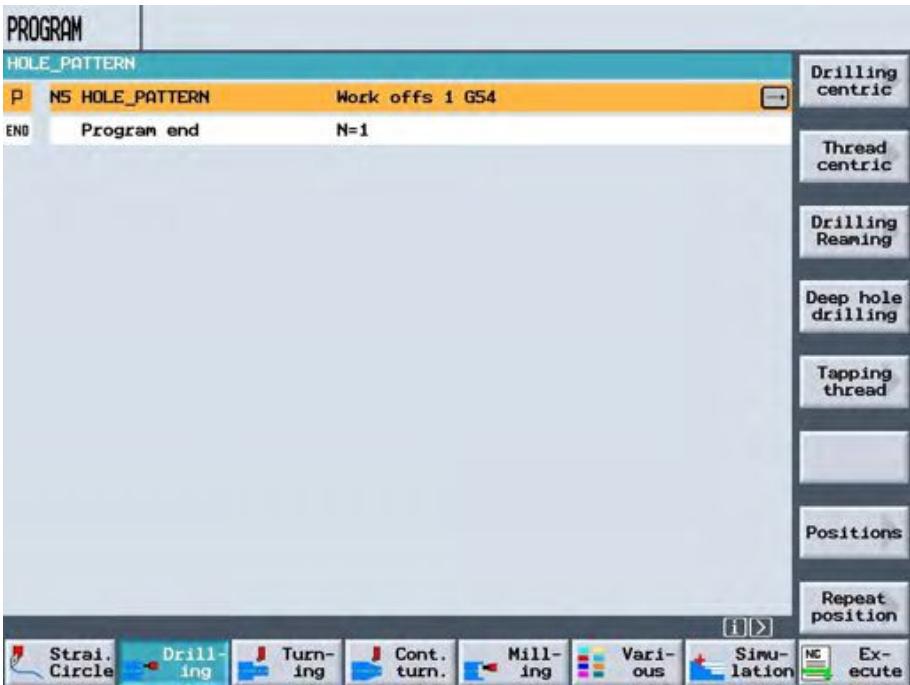
### Selection and general functions under “Drilling“

#### 2.1 Selection of function “Drilling“

After the program head has been written, the program blocks for machining, e.g. for holes and tapped holes, can be programmed.



Pressing the HSK 2 “Drilling“ selects the range for the programming of holes, tapped holes and position patterns.



Notes

## Section 2

Notes

Selection and general functions under "Drilling"

### 2.2 Vertical Softkey strip

Drilling  
centric

Pressing the VSK 1 "Drilling centric" calls up the function "Drilling centric".

See *Section 3 "Drilling centric"*.

Thread  
centric

Pressing the VSK 2 "Thread centric" calls up the function "Thread centric".

See *Section 4 "Thread centric"*.

Drilling  
Reaming

Pressing the VSK 3 "Drilling Reaming", the following functions are called up:

Centring

Drilling

Reaming

See *Section 5 "Drilling / Reaming"*.

Deep hole  
drilling

Pressing the VSK 4 "Deep hole drilling" calls up the function "Deep hole drilling".

See *Section 6 "Deep hole drilling"*.

Tapping  
thread

Pressing the VSK 5 "Tapping thread", the following functions are called up:

Tapping

Thread milling

See *Section 7 "Thread cutting"*.

Positions

Pressing the VSK 7 "Positions", the following functions are called up:

Rectangular / polar positions

Patterns (line, grid)

Full and partial circle

See *Section 8 "Position patterns"*.

Repeat  
position

Pressing the VSK 8 "Repeat position" calls up the function "Repeat position". An already programmed position pattern can be repeated.

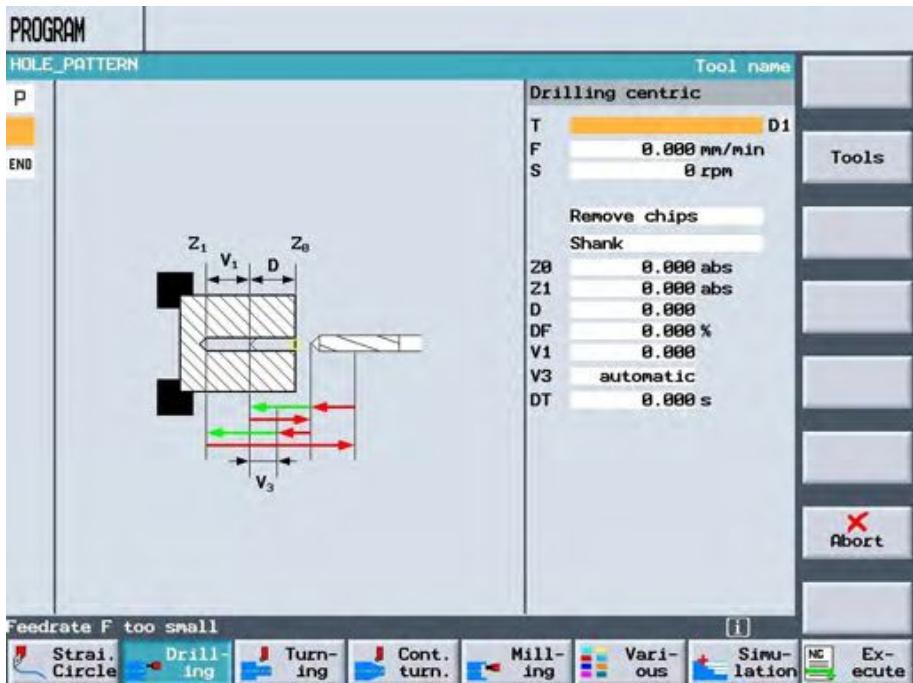
## Section 3

### Drilling centric

Drilling centric

By pressing the VSK 1 "Drilling centric" the window for the programming of holes on the axial centre of the work piece.

Notes



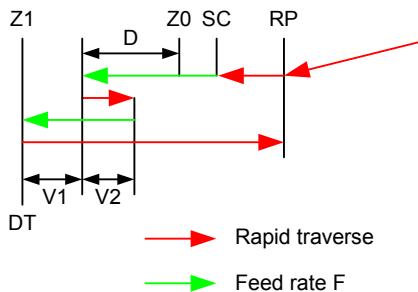
#### Approach / retraction to and from the work piece

After Start the tool traverses at rapid traverse rate to the programmed position taking into account the retraction plane and the safety distance.

#### Chip breaking:

The tool drills at the programmed feed rate (F) to the 1<sup>st</sup> depth of infeed. Once this has been attained, the tool is retracted by the amount (V2) for chip breaking before drilling proceeds to the next infeed depth, which can be reduced by the factor DF. This is repeated until the final depth Z1 is reached and the dwell DT has been completed. Following this the tool will be retracted with rapid traverse to the safety level.

#### Sequence diagram:



## Section 3

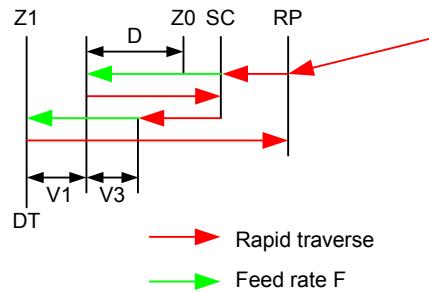
Notes

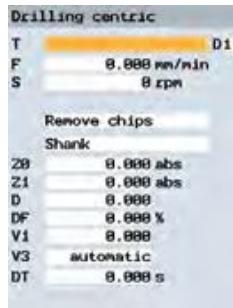
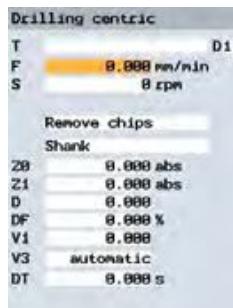
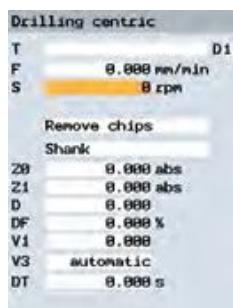
Drilling centric

### Chip clearance:

The tool drills at the programmed feed rate (F) to the 1<sup>st</sup> depth of infeed. Once this is reached, the tool is retracted with rapid traverse for chip clearance and then returns to the 1<sup>st</sup> infeed depth minus a small measure (V3) (pre-stop). Drilling then continues to the next infeed depth followed by a retraction for chip clearance. This is repeated until the final programmed depth (Z1) is attained. When the dwell has elapsed, the tool is retracted with rapid traverse to the safety level.

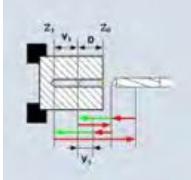
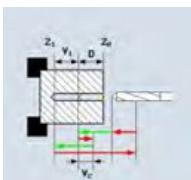
### Sequence diagram:



Parameter	Help picture	Description and hints
T + D		Name of tool and cutting edge number
F		Feed rate [mm/min, mm/rev]
S or V		Spindle speed or cutting speed [Rev/min, m/min]

## Section 3

### Drilling centric

Parameter	Help picture	Description and hints	Notes																																										
Chip clearance, Chip breaking	<p><b>Drilling centric</b></p> <table> <tr><td>T</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td></tr> <tr><td>S</td><td>0 rpm</td></tr> </table> <p><b>Remove chips</b></p> <p><b>Shank</b></p> <table> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>D</td><td>0.000</td></tr> <tr><td>DF</td><td>0.000 %</td></tr> <tr><td>V1</td><td>0.000</td></tr> <tr><td>V3</td><td>automatic</td></tr> <tr><td>DT</td><td>0.000 s</td></tr> </table> <p><b>Drilling centric</b></p> <table> <tr><td>T</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td></tr> <tr><td>S</td><td>0 rpm</td></tr> </table> <p><b>Chipbreaking</b></p> <p><b>Shank</b></p> <table> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>D</td><td>0.000</td></tr> <tr><td>DF</td><td>0.000 %</td></tr> <tr><td>V1</td><td>0.000</td></tr> <tr><td>V2</td><td>0.000</td></tr> <tr><td>DT</td><td>0.000 s</td></tr> </table>	T	D1	F	0.000 mm/min	S	0 rpm	ZB	0.000 abs	Z1	0.000 abs	D	0.000	DF	0.000 %	V1	0.000	V3	automatic	DT	0.000 s	T	D1	F	0.000 mm/min	S	0 rpm	ZB	0.000 abs	Z1	0.000 abs	D	0.000	DF	0.000 %	V1	0.000	V2	0.000	DT	0.000 s	<p>Chip clearance</p>  <p>Chip breaking</p> 			
T	D1																																												
F	0.000 mm/min																																												
S	0 rpm																																												
ZB	0.000 abs																																												
Z1	0.000 abs																																												
D	0.000																																												
DF	0.000 %																																												
V1	0.000																																												
V3	automatic																																												
DT	0.000 s																																												
T	D1																																												
F	0.000 mm/min																																												
S	0 rpm																																												
ZB	0.000 abs																																												
Z1	0.000 abs																																												
D	0.000																																												
DF	0.000 %																																												
V1	0.000																																												
V2	0.000																																												
DT	0.000 s																																												
Shank, Point	<p><b>Drilling centric</b></p> <table> <tr><td>T</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td></tr> <tr><td>S</td><td>0 rpm</td></tr> </table> <p><b>Chipbreaking</b></p> <p><b>Shank</b></p> <table> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>D</td><td>0.000</td></tr> <tr><td>DF</td><td>0.000 %</td></tr> <tr><td>V1</td><td>0.000</td></tr> <tr><td>V2</td><td>0.000</td></tr> <tr><td>DT</td><td>0.000 s</td></tr> </table> <p><b>Drilling centric</b></p> <table> <tr><td>T</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td></tr> <tr><td>S</td><td>0 rpm</td></tr> </table> <p><b>Chipbreaking</b></p> <p><b>Tip</b></p> <table> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>D</td><td>0.000</td></tr> <tr><td>DF</td><td>0.000 %</td></tr> <tr><td>V1</td><td>0.000</td></tr> <tr><td>V2</td><td>0.000</td></tr> <tr><td>DT</td><td>0.000 s</td></tr> <tr><td>XD</td><td>0.000</td></tr> </table>	T	D1	F	0.000 mm/min	S	0 rpm	ZB	0.000 abs	Z1	0.000 abs	D	0.000	DF	0.000 %	V1	0.000	V2	0.000	DT	0.000 s	T	D1	F	0.000 mm/min	S	0 rpm	ZB	0.000 abs	Z1	0.000 abs	D	0.000	DF	0.000 %	V1	0.000	V2	0.000	DT	0.000 s	XD	0.000	<p>Shank:</p> <p>The tool progresses until the drill shank programmed value Z1. For this the angle stated in the tool list is taken into account.</p> <p>Point:</p> <p>The tool progresses until the drill point has reached the programmed value Z1.</p>	
T	D1																																												
F	0.000 mm/min																																												
S	0 rpm																																												
ZB	0.000 abs																																												
Z1	0.000 abs																																												
D	0.000																																												
DF	0.000 %																																												
V1	0.000																																												
V2	0.000																																												
DT	0.000 s																																												
T	D1																																												
F	0.000 mm/min																																												
S	0 rpm																																												
ZB	0.000 abs																																												
Z1	0.000 abs																																												
D	0.000																																												
DF	0.000 %																																												
V1	0.000																																												
V2	0.000																																												
DT	0.000 s																																												
XD	0.000																																												
Z0	<p><b>Drilling centric</b></p> <table> <tr><td>T</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td></tr> <tr><td>S</td><td>0 rpm</td></tr> </table> <p><b>Remove chips</b></p> <p><b>Tip</b></p> <table> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>D</td><td>0.000</td></tr> <tr><td>DF</td><td>0.000 %</td></tr> <tr><td>V1</td><td>0.000</td></tr> <tr><td>V3</td><td>automatic</td></tr> <tr><td>DT</td><td>0.000 s</td></tr> <tr><td>XD</td><td>0.000</td></tr> </table>	T	D1	F	0.000 mm/min	S	0 rpm	ZB	0.000 abs	Z1	0.000 abs	D	0.000	DF	0.000 %	V1	0.000	V3	automatic	DT	0.000 s	XD	0.000	<p>Reference point</p> <p>[abs]</p>																					
T	D1																																												
F	0.000 mm/min																																												
S	0 rpm																																												
ZB	0.000 abs																																												
Z1	0.000 abs																																												
D	0.000																																												
DF	0.000 %																																												
V1	0.000																																												
V3	automatic																																												
DT	0.000 s																																												
XD	0.000																																												

## Section 3

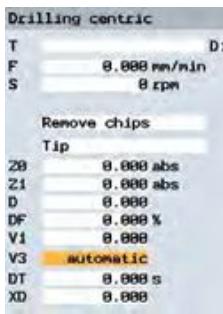
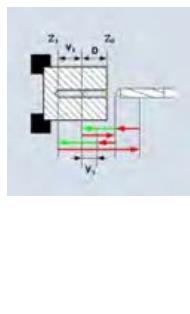
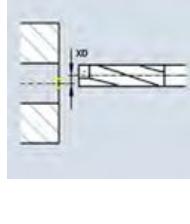
### Drilling centric

Notes

Parameter	Help picture	Description and hints
Z1		Entry depth for the drill point or the drill shank referred to Z0. [abs, inc]
D		Max. infeed (1 <sup>st</sup> cut)
DF		Percentage for every further infeed  DF = 100      Amount of infeed is constant. DF < 100      The amount of infeed is reduced by this amount for every new infeed.  Example: First infeed 4 mm, DF = 80% Next infeed: $4 \times 80\% = 3,2$ mm Next infeed: $3,2 \times 80\% = 2,56$ mm etc.
V1		Minimum infeed  The parameter V1 is available only if DF<100 was programmed. If the amount of infeed is very small, the parameter "V1" can be used to program a minimum infeed.  V1 < amount of infeed: The tool advances by the amount of infeed V1 > amount of infeed: The tool advances by the value programmed under V1.
V2		Distance of the retraction or specified by machine datum – only for chip breaking  Amount by which the drill is retracted for chip breaking.  V2=0: The tool is not retracted, instead it stops for the duration of one revolution.

## Section 3

### Drilling centric

Parameter	Help picture	Description and hints	Notes
V3		<p>Pre-stop – only for chip clearance</p> <p>Distance by which the drill stops prior to reaching the previous infeed depth when advancing at rapid traverse rate after chip clearance.</p> <p>Automatically: The pre-stopping distance is evaluated by ShopTurn.</p> <p>Only available with "Chip clearance".</p>	
DT		Dwell [s, U]	
XD		<p>Centre offset in X-direction</p> <p>The centre offset can be used for instance to produce a high accuracy bore A rotation drill (Type Rotation drill) must be used (Type Rotation drill) or a U-drill (Type Drill). A "normal" drill is not suitable. The maximum centre offset is stored in a machine datum.</p>	

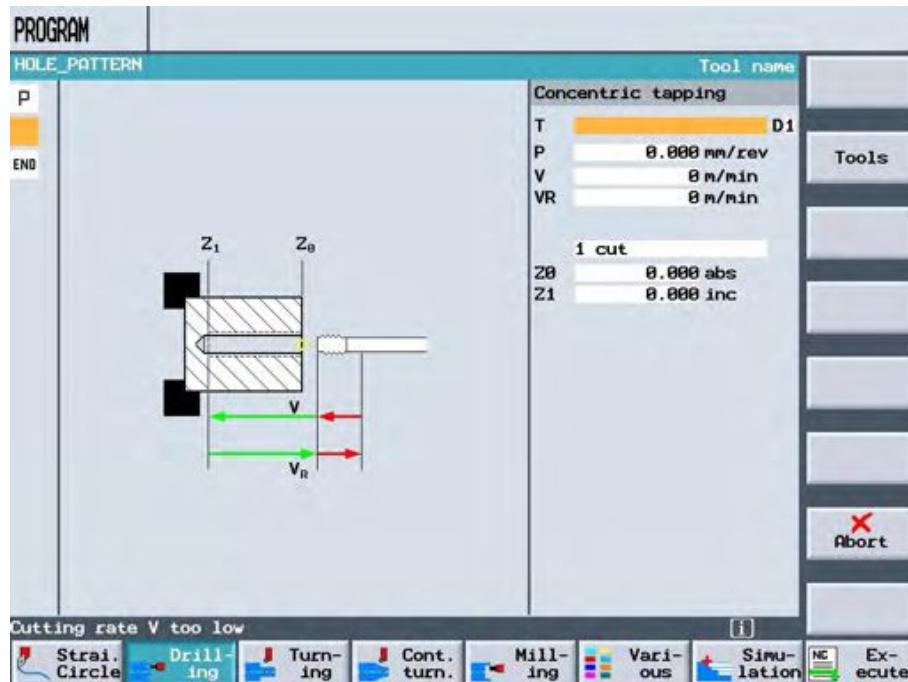
## Section 4

Notes

Thread centric

Thread  
centric

By pressing the VSK 2 "Thread centric" the window for the programming of concentric tapped holes will be called up..



### Approach / retraction to and from the work piece

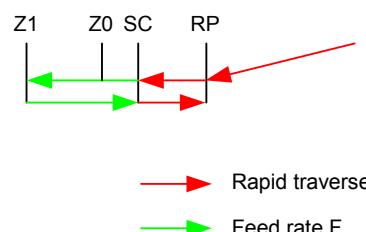
The tool traverses at rapid traverse rate to the programmed position taking into account the retraction plane and the safety distance. After that the tool enters the work piece with the pitch programmed under P until the depth Z1 is attained.

Three variants are available for machining:

#### Single cut

1. The tool cuts with the programmed spindle speed S or cutting speed V respectively to the tapping depth Z1.
2. The direction of rotation of the spindle changes and the tool is retracted with the programmed spindle speed SR or cutting speed VR respectively to the safety level and then with rapid traverse to the retraction plane.

Sequence diagram:



## Section 4

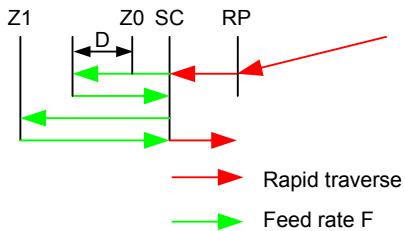
### Thread centric

#### Chip clearance

1. The tool cuts with the programmed spindle speed S or cutting speed V respectively as far as the 1<sup>st</sup> infeed depth (maximum infeed depth D).
2. The direction of rotation of the tool changes and the tool is retracted from the work piece for chip clearance at the spindle speed SR or cutting speed VR respectively as far as the safety level.
3. The direction of rotation is changed once more and the tool re-enters at the spindle speed S or cutting speed V respectively to the 1<sup>st</sup> infeed depth and continues to the next infeed depth.
4. Steps 2 and 3 are repeated until the programmed final depth Z1 is reached.
5. The direction of rotation of the spindle changes and the tool traverses with the spindle speed SR or cutting speed VR respectively as far as the safety level and then at rapid traverse rate to the retraction plane.

Notes

#### Sequence diagram:



#### Chip breaking

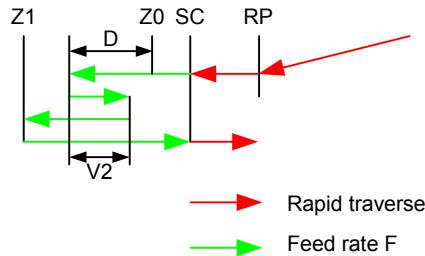
1. The tool cuts with the programmed spindle speed S or cutting speed V respectively to the 1<sup>st</sup> infeed depth (maximum infeed depth D).
2. The direction of rotation of the tools changes and the tool is retracted by the amount V2 in order to break the chips.
3. The direction of rotation changes once more and the tool continues at the spindle speed S or cutting speed V respectively to the next infeed depth.
4. Steps 2 and 3 are repeated until the final depth Z1 is reached.
5. The direction of rotation of the spindle changes and the tool is retracted with the spindle speed SR or cutting speed VR respectively back to the safety level.

## Section 4

Notes

Thread centric

Sequence diagram:



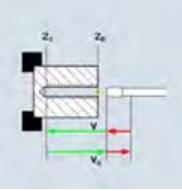
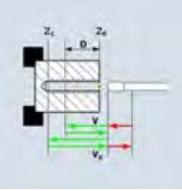
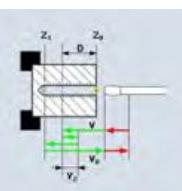
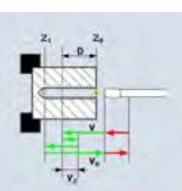
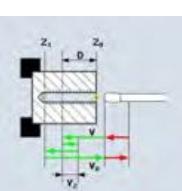
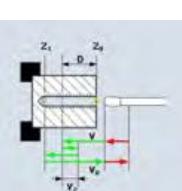
Hints:

The documentation by the machine manufacturer must be observed.

Parameter	Help picture	Description and hints														
T, D, S / V		See Section 3 "Drilling centric".														
P	<p><b>Concentric tapping</b></p> <table border="1"> <tr><td>T</td><td>D1</td></tr> <tr><td>P</td><td>0.000 mm/rev</td></tr> <tr><td>V</td><td>0 m/min</td></tr> <tr><td>VR</td><td>0 m/min</td></tr> <tr><td colspan="2">1 cut</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> </table>	T	D1	P	0.000 mm/rev	V	0 m/min	VR	0 m/min	1 cut		Z0	0.000 abs	Z1	0.000 inc	<p>Thread pitch</p> <p>The thread pitch depends on the used tool. [mm/rev, in/rev, MODULE, Threads/]</p> <p>MODULE: Used for example for worms that mesh with gearwheels.</p> <p>Threads/: Used for example for pipe threads.</p> <p>When entries in threads/" are made, the whole number preceding the decimal point must be entered into the first parameter field, while the number following the decimal point must be entered into the second and third field as a fraction.</p> <p>Example: 3,08 Threads/"</p> <p style="text-align: right;"><b>P 13 1/ 2 Thrd."/&gt;</b></p>
T	D1															
P	0.000 mm/rev															
V	0 m/min															
VR	0 m/min															
1 cut																
Z0	0.000 abs															
Z1	0.000 inc															
SR / VR	<p><b>Concentric tapping</b></p> <table border="1"> <tr><td>T</td><td>D1</td></tr> <tr><td>P</td><td>0.000 mm/rev</td></tr> <tr><td>V</td><td>0 m/min</td></tr> <tr><td>VR</td><td>0 m/min</td></tr> <tr><td colspan="2">1 cut</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> </table>	T	D1	P	0.000 mm/rev	V	0 m/min	VR	0 m/min	1 cut		Z0	0.000 abs	Z1	0.000 inc	<p>Spindle speed for retraction</p> <p>[m/min, U/min]</p>
T	D1															
P	0.000 mm/rev															
V	0 m/min															
VR	0 m/min															
1 cut																
Z0	0.000 abs															
Z1	0.000 inc															

## Section 4

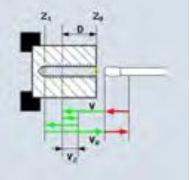
### Thread centric

Parameter	Help picture	Description and hints	Notes
Single cut, Chip clearance Chip breaking		<p>Single cut</p> <p>The thread is cut in one pass without an interruption.</p>	
<b>Concentric tapping</b>			
T D1 P 8.000 mm/rev V 0 m/min VR 0 m/min			
ZB 0.000 abs Z1 0.000 inc			
<b>Remove chips</b>			
ZB 0.000 abs Z1 0.000 inc D 0.000		<p>Chip clearance</p> <p>The tap is retracted from the work piece for chip clearance (not available when tapping with a compensating chuck).</p>	
<b>Chipbreaking</b>			
ZB 0.000 abs Z1 0.000 inc D 0.000 V2 automatic		<p>Chip breaking</p> <p>The tap is retracted by the retraction amount V2 for chip breaking (not available when tapping with a compensating chuck).</p>	
Z0		<p>Reference point</p> <p>[abs]</p>	
<b>Concentric tapping</b>			
T D1 P 8.000 mm/rev V 0 m/min VR 0 m/min			
Chipbreaking			
ZB 0.000 abs Z1 0.000 inc D 0.000 V2 automatic			
Z1		<p>Tapping depth referred to Z0</p> <p>[abs, inc]</p>	
<b>Concentric tapping</b>			
T D1 P 8.000 mm/rev V 0 m/min VR 0 m/min			
Chipbreaking			
ZB 0.000 abs Z1 0.000 inc D 0.000 V2 automatic			
D		<p>Maximum infeed (only for chip clearance or chip breaking)</p> <p>[mm]</p>	
<b>Concentric tapping</b>			
T D1 P 8.000 mm/rev V 0 m/min VR 0 m/min			
Chipbreaking			
ZB 0.000 abs Z1 0.000 inc D 0.000 V2 automatic			

## Section 4

### Thread centric

Notes

Parameter	Help picture	Description and hints
V2  <b>Concentric tapping</b> T D1 P 8.000 mm/rev V 8 m/min VR 8 m/min  <b>Chipbreaking</b> Z0 8.000 abs Z1 8.000 inc D 8.000 V2 automatic		<p>Retraction amount (only for chip breaking)</p> <p>Amount by which the drill is retracted for chip breaking.</p> <p>V2=automatically: The tool retracts for one revolution.</p> <p>Only available with "chip breaking".</p>

## Section 5

### Drilling / Reaming

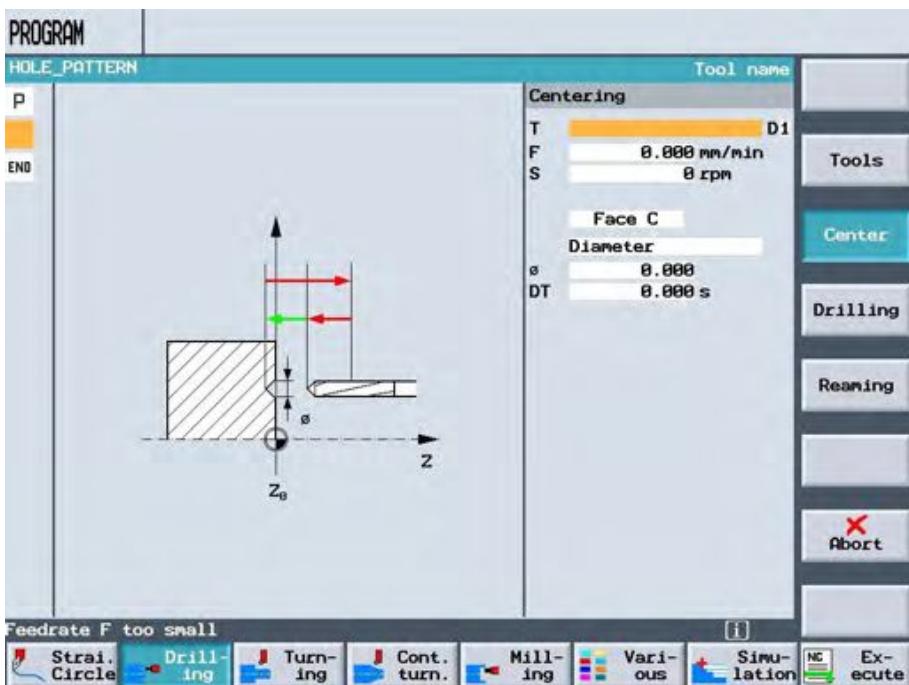
#### 5.1 Centring

Drilling  
Reaming

By pressing the VSK 3 "Drilling Reaming" the window for the programming of holes on the end face or on a perimeter face.

Center

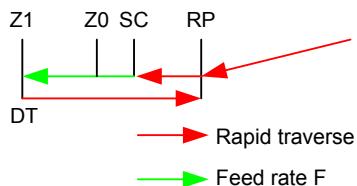
Pressing the VSK 3 "Centre" opens the window for the programming of centres.



#### Approach / retraction to and from the work piece

After Start the tool traverses at rapid traverse rate to the programmed position taking into account the retraction plane and the safety distance. Then the tool progresses into the work piece at the feed rate programmed under F, until the depth or the diameter is reached. After the dwell DT has elapsed the tool is retracted with rapid traverse to the safety level.

Sequence diagram:

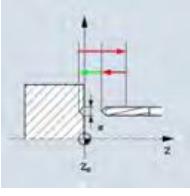
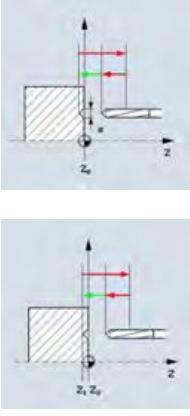
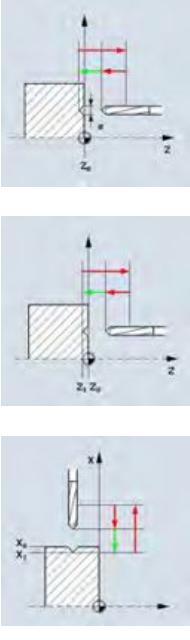
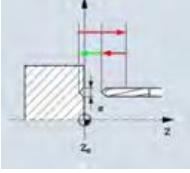


Notes

## Section 5

### Drilling / Reaming

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Section 3 "Drilling centric".
Position		Select between 4 possible positions : End face C - front End face Y - front Perimeter surface C - external Perimeter surface Y - external
Diameter, Tip		Diameter: The tool enters the work piece until the programmed diameter is attained on the surface. For this the angle of the tool entered in the tool list is taken into account.  Point: The tool progresses into the work piece until the programmed depth is reached.
Ø / Z1 / X1		Ø: Diameter of the centring Only available for "Centring – Diameter".  Z1: Infeed depth for the drill point or the drill shank referred to Z0 [abs, ink]  Only available for "Face C" and "Face Y".  X1: Infeed depth for the drill point or the drill shank referred to X0 [abs, ink]  Only available for „Perimeter surface C“ and „Perimeter surface Y“ .
DT		Dwell [s, U]

## Section 5

### Drilling / Reaming

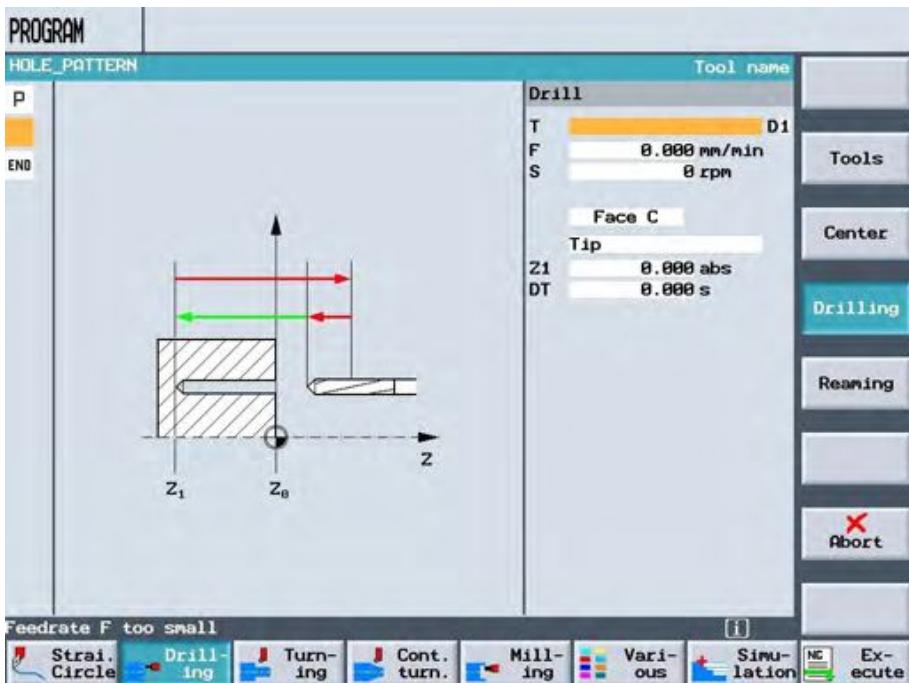
#### 5.2 Drilling

Drilling  
Reaming

By pressing the VSK 3 "Drilling Reaming" the window for the programming of holes on the face- or perimeter surface will be called up.

Drilling

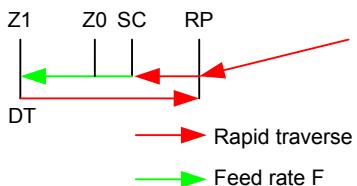
Pressing the VSK 3 "Drilling", for the drilling function.



#### Approach / retraction to and from the work piece

After Start the tool traverses at rapid traverse rate to the programmed position taking into account the retraction plane and the safety distance. Then the tool progresses into the work piece at the feed rate programmed under F until the final depth (X1 or Z1) is attained. After the dwell DT has elapsed, the tool will be retracted with rapid traverse to the safety level.

Sequence diagram:



Notes

## Section 5

### Drilling / Reaming

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Section 3 "Drilling centric".
Position		Selection between 4 different positions: Face C - front Face Y - front Perimeter surface C - external Perimeter surface Y - external
Shank, Tip	  	Point: The tool progresses until the drill point has reached the programmed value Z1.  Shank: The tool progresses until the drill shank has reached the programmed value Z1. For this the angle of the tool as stated in the tool list is taken into account.
Z1 / X1		Z1: Entry depth for the drill point or the drill shank referred to Z0. [abs, inc]  Only available for „Face C“ and „Face Y“ .  X1: Entry depth for the drill point or the drill shank referred to X0. [abs, inc]  Only available for „Perimeter surface C“ and „Perimeter surface Y“ .
DT		Dwell [s, U]

## Section 5

### Drilling / Reaming

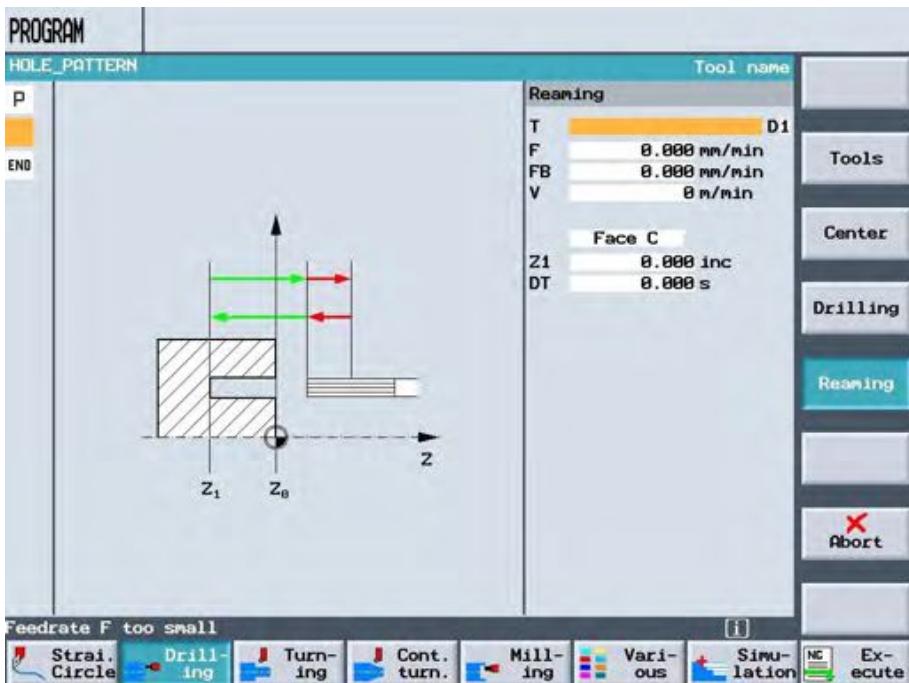
#### 5.3 Reaming

Drilling  
Reaming

By pressing the VSK 3 "Drilling Reaming" the window for the programming of holes on the face or perimeter surface will be called up.

Reaming

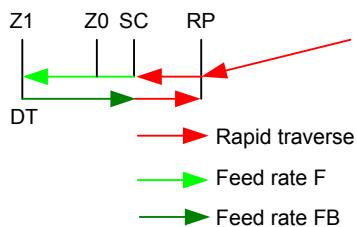
Pressing the VSK 4 "Reaming", for the reaming function.



#### Approach / retraction to and from the work piece

The tool traverses at rapid traverse rate to the programmed position taking into account the retraction plane and the safety distance. Then the tool progresses into the work piece at the feed rate programmed under F, until the final depth (X1 or Z1) is attained. After the dwell DT has elapsed, the tool will be retracted with rapid traverse to the safety level.

Sequence diagram:

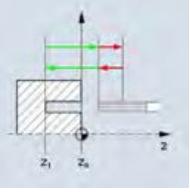
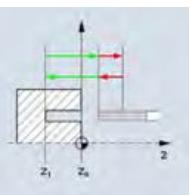
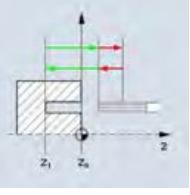
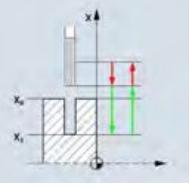
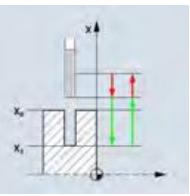


Notes

## Section 5

### Drilling / Reaming

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Section 3 "Drilling centric".
FB		Retraction feed rate [m/min, mm/rev]
Position		Selection between 4 different positions: Face C - front Face Y - front Perimeter surface C - external Perimeter surface Y - external
Z1 / X1		<p>Z1:</p> <p>Entry depth for the drill point or the drill shank referred to Z0.</p> <p>[abs, inc]</p> <p>Only available for „Face C“ and „Face Y“ .</p>
		<p>X1:</p> <p>Entry depth for the drill point or the drill shank referred to X0.</p> <p>[abs, inc]</p> <p>Only available for "Perimeter surface C" and "Perimeter surface Y" .</p>
DT		Dwell [s, U]

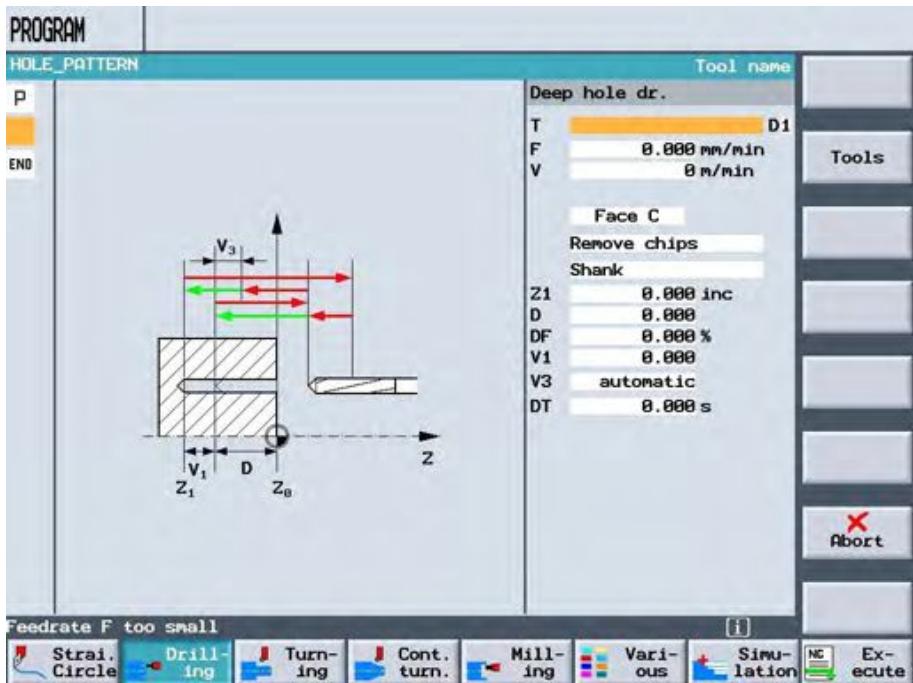
## Section 6

### Deep hole drilling

Deep hole  
drilling

By pressing the VSK 4 "Deep hole drilling" the window for the programming deep holes on the face or perimeter surface will be called up.

Notes



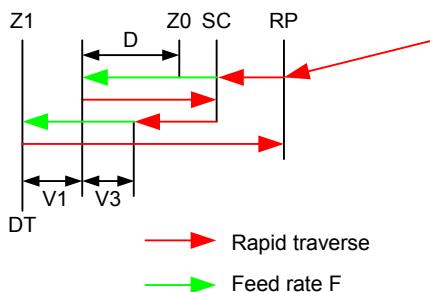
#### Approach / retraction to and from the work piece

The tool traverses at rapid traverse rate to the programmed position taking into account the retraction plane and the safety distance. Then the tool progresses into the work piece at the programmed feed rate.

#### Chip clearance:

The tool drills at the programmed feed rate (F) to the 1<sup>st</sup> depth of infeed. Once this is reached, the tool is retracted with rapid traverse for chip clearance and then returns to the 1<sup>st</sup> infeed depth minus a small measure (V3) (pre-stop). Drilling then continues to the next infeed depth followed by a retraction for chip clearance. This is repeated until the final programmed depth (Z1) is attained. When the dwell has elapsed, the tool is retracted with rapid traverse to the safety level.

#### Sequence diagram:



## Section 6

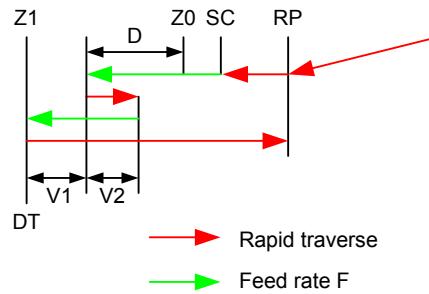
Notes

### Deep hole drilling

#### Chip breaking:

The tool progresses at the programmed feed rate ( $F$ ) to the 1<sup>st</sup> infeed depth. Once attained, the tool is retracted by a retraction amount ( $V_2$ ) to break the chip. Drilling then continues to the next infeed depth. This procedure is repeated until the final depth is reached. The retraction distance can be specified by either a machine datum or in the parameter mask. If the parameter is already specified by machine datum, it will not appear in the parameter mask.

#### Sequence diagram:

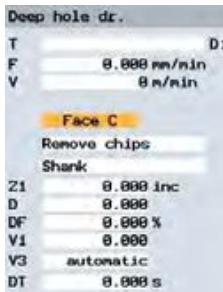
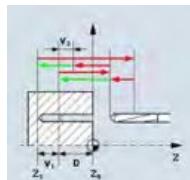
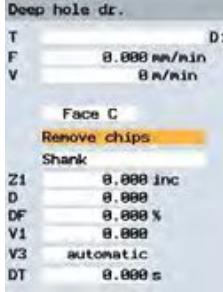
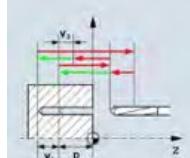
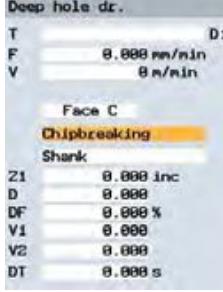
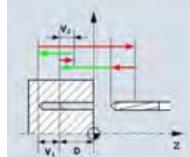
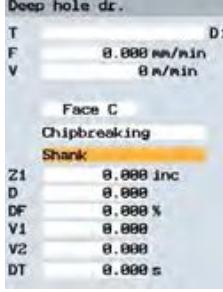
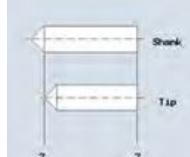
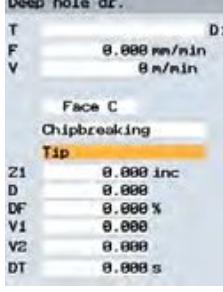
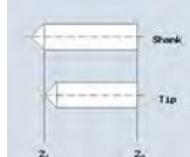


#### Hints:

The documentation by the machine manufacturer must be observed.

## Section 6

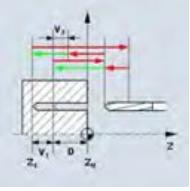
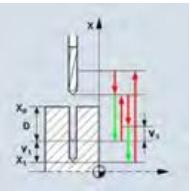
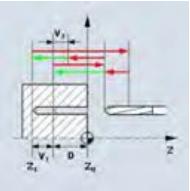
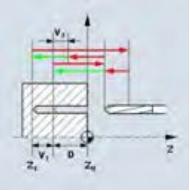
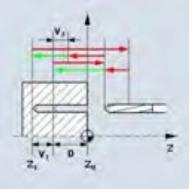
### Deep hole drilling

Parameter	Help picture	Description and hints	Notes
T, D, F, S / V		See Section 3 "Drilling centric".	
Position		Selection between 4 different positions: Face C - front Face Y - front Perimeter surface C - external Perimeter surface Y - external	
			
Chip clearance, Chip breaking		Chip clearance	
			
		Chip breaking	
Shank, Point		Shank:  The tool progresses until the drill shank programmed value Z1. For this the angle stated in the tool list is taken into account.	
			
		Point:  The tool progresses until the drill point has reached the programmed value Z1.	

## Section 6

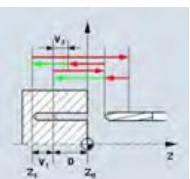
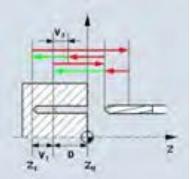
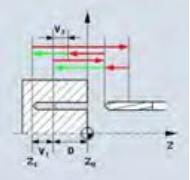
### Deep hole drilling

Notes

Parameter	Help picture	Description and hints
Z1 / X1	 	<p>Z1: Entry depth for the drill point or the drill shank referred to Z0. [abs, inc]</p> <p>X1: Entry depth for the drill point or the drill shank referred to X0. [abs, inc]</p> <p>Only available for "Face C" and "Face Y".</p>
D		Max. infeed (1 <sup>st</sup> cut)
DF		<p>Percentage for every further infeed</p> <p>DF = 100      Amount of infeed is constant.  DF &lt; 100      The amount of infeed is reduced by this amount for every new infeed.</p> <p>Example:  First infeed 4 mm, DF = 80%  Next infeed: <math>4 \times 80\% = 3,2</math> mm  Next infeed: <math>3,2 \times 80\% = 2,56</math> mm  etc.</p>
V1		<p>Minimum infeed</p> <p>The parameter V1 is available only if DF&lt;100 was programmed. If the amount of infeed is very small, the parameter "V1" can be used to program a minimum infeed.</p> <p>V1 &lt; amount of infeed: The tool advances by the amount of infeed  V1 &gt; amount of infeed: The tool advances by the value programmed under V1.</p>

## Section 6

### Deep hole drilling

Parameter	Help picture	Description and hints	Notes
V2  Deep hole dr. T Di F 0.000 mm/min V 0 m/min  Face C Chipbreaking Tip Z1 0.000 inc D 0.000 DF 0.000 % V1 0.000 V2 0.000 DT 0.000 s		Distance of the retraction or specified by machine datum  Amount by which the drill is retracted for chip breaking.  V2=0: The tool is not retracted, instead it stops for the duration of one revolution.  Only available for „Chip breaking“ .	
V3  Deep hole dr. T Di F 0.000 mm/min V 0 m/min  Face C Remove chips Tip Z1 0.000 inc D 0.000 DF 0.000 % V1 0.000 V3 automatic DT 0.000 s		Pre-stop – only for chip clearance  Distance by which the drill stops prior to reaching the previous infeed depth when advancing at rapid traverse rate after chip clearance.  Automatically: The pre-stopping distance is evaluated by ShopTurn.  Only available for "Chip clearance" .	
DT  Deep hole dr. T Di F 0.000 mm/min V 0 m/min  Face C Remove chips Tip Z1 0.000 inc D 0.000 DF 0.000 % V1 0.000 V3 automatic DT 0.000 s		Dwell [s, U]	

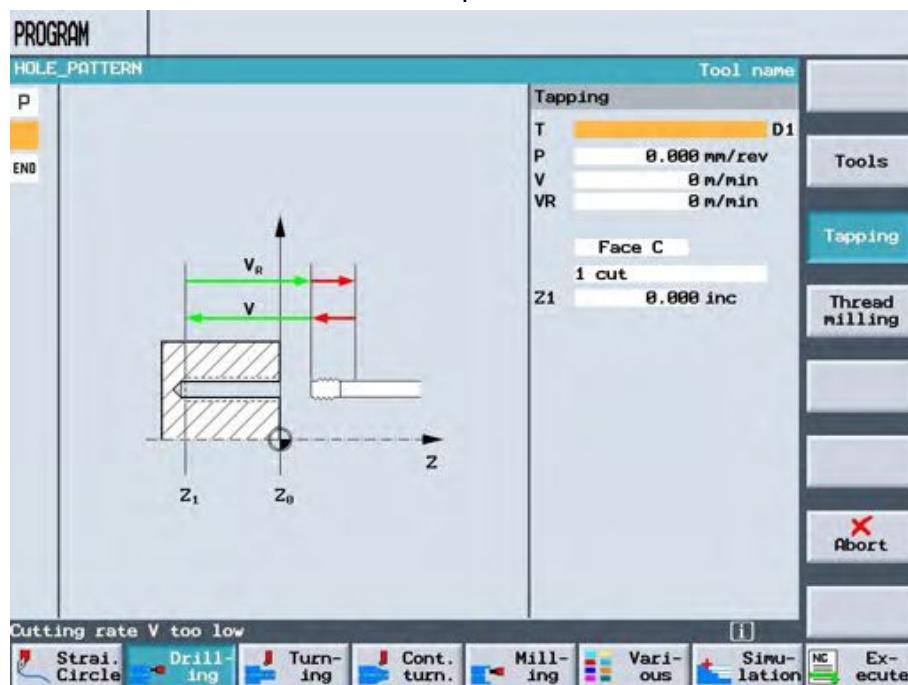
## Section 7

### Thread cutting

Notes

Tapping  
thread

By pressing the VSK 5 "Tapping thread" the window for the programming internal and external threads on face or perimeter surface will be called up.



#### 7.1 Vertical Softkey strip

Tapping

Pressing the VSK 3 "Tapping" the respective function "Tapping".

See Section 7.2 "Tapping".

Thread  
milling

Pressing the VSK 4 "Thread milling" calls up the functions for "Milling of internal and external threads".

See Section 7.3 "Thread milling".

## Section 7

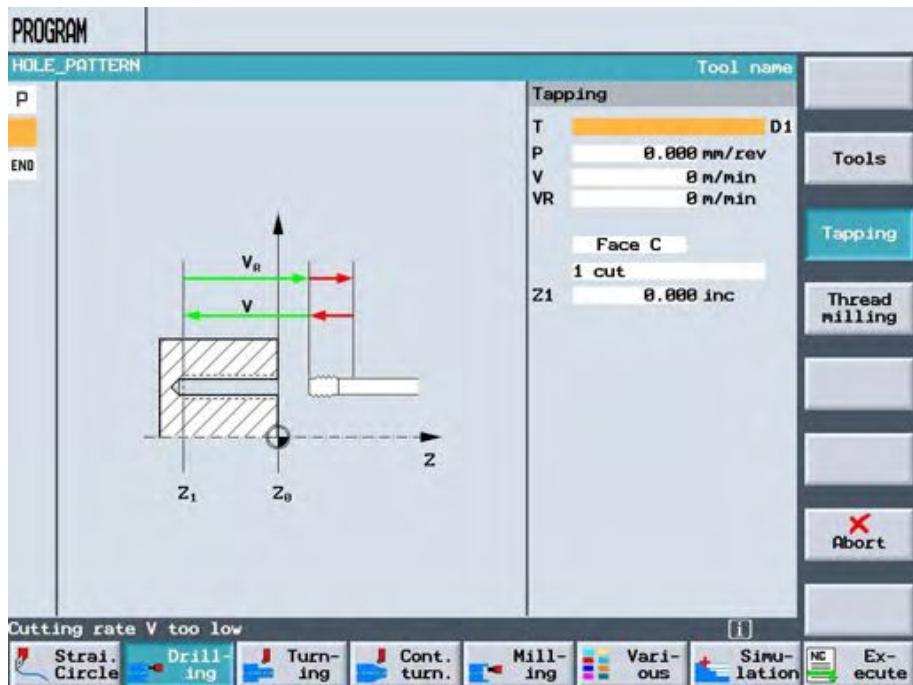
### Thread cutting

#### 7.2 Tapping

Tapping

Pressing the VSK 3 "Tapping" opens the window for the programming of tapped holes.

Notes



#### Approach / retraction to and from the work piece

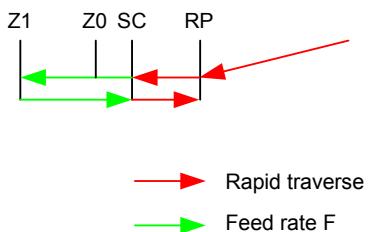
The tool traverses with the stopped spindle at rapid traverse rate to the retraction plane and then to the safety distance. There the spindle starts to rotate and the spindle speed and feed are synchronized.

Three variants are available for machining

#### Single cut

1. The tool cuts with the programmed spindle speed S or cutting speed V respectively to the tapping depth Z1.
2. The direction of rotation of the spindle changes and the tool is retracted with the programmed spindle speed SR or cutting speed VR respectively to the safety level and then with rapid traverse to the retraction plane.

Sequence diagram:



## Section 7

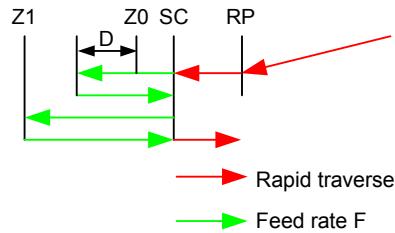
Notes

Thread cutting

### Chip clearance

1. The tool cuts with the programmed spindle speed S or cutting speed V respectively as far as the 1<sup>st</sup> infeed depth (maximum infeed depth D).
2. The direction of rotation of the tool changes and the tool is retracted from the work piece for chip clearance at the spindle speed SR or cutting speed VR respectively as far as the safety level.
3. The direction of rotation is changed once more and the tool re-enters at the spindle speed S or cutting speed V respectively to the 1<sup>st</sup> infeed depth and continues to the next infeed depth.
4. Steps 2 and 3 are repeated until the programmed final depth Z1 is reached.
5. The direction of rotation of the spindle changes and the tool traverses with the spindle speed SR or cutting speed VR respectively as far as the safety level and then at rapid traverse rate to the retraction plane.

Sequence diagram:



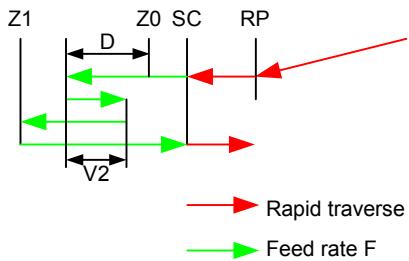
### Chip breaking

1. The tool cuts with the programmed spindle speed S or cutting speed V respectively to the 1<sup>st</sup> infeed depth (maximum infeed depth D).
2. The direction of rotation of the tools changes and the tool is retracted by the amount V2 in order to break the chips.
3. The direction of rotation changes once more and the tool continues at the spindle speed S or cutting speed V respectively to the next infeed depth.
4. Steps 2 and 3 are repeated until the final depth Z1 is reached.
5. The direction of rotation of the spindle changes and the tool is retracted with the spindle speed SR or cutting speed VR respectively back to the safety level.

## Section 7

### Thread cutting

Sequence diagram:



Notes

#### Hints:

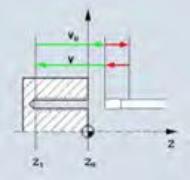
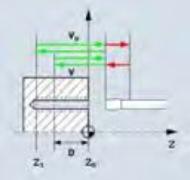
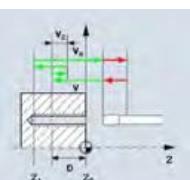
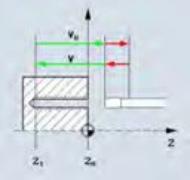
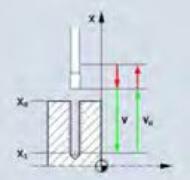
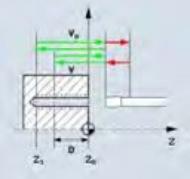
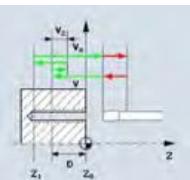
The documentation by the machine manufacturer must be observed.

Parameter	Help picture	Description and hints
T, D, S / V		See Section 3 "Drilling centric".
P		<p>Thread pitch</p> <p>The thread pitch depends on the used tool.</p> <p>[mm/rev, in/rev, MODULE, Threads/"]</p> <p>MODULE: Used for example for worms that mesh with gearwheels.</p> <p>Threads/": Used for example for pipe threads.</p> <p>When entries in threads/” are made, the whole number preceding the decimal point must be entered into the first parameter field, while the number following the decimal point must be entered into the second and third field as a fraction.</p> <p>Example: 3,08 Threads/”</p> <p style="text-align: center;"><b>P 13 1/ 2 Thrdls/”</b></p> <p>The thread pitches are converted when toggling.</p>
SR, VR		<p>Spindle speed for retraction</p> <p>[m/min, U/min]</p>
Position		<p>Selection between 4 different positions:</p> <ul style="list-style-type: none"> <li>Face C - front</li> <li>Face Y - front</li> <li>Perimeter surface C - external</li> <li>Perimeter surface Y - external</li> </ul>

## Section 7

### Thread cutting

Notes

Parameter	Help picture	Description and hints
Single cut, Chip clearance Chip breaking	  	<p>Single cut</p> <p>The thread is cut in one pass without an interruption.</p> <p>Chip clearance</p> <p>The tap is retracted from the work piece for chip clearance (not available when tapping with a compensating chuck).</p> <p>Chip breaking</p> <p>The tap is retracted by the retraction amount V2 for chip breaking (not available when tapping with a compensating chuck).</p>
Z1 / X1	 	<p>Tapping depth referred to Z0 [abs, inc]</p> <p>Only available for "Face C" and "Face Y".</p> <p>Tapping depth referred to X0 [abs, inc]</p> <p>Only available for "Perimeter surface C" and "Perimeter surface Y".</p>
D		<p>Maximum infeed [mm]</p> <p>Only available for "Chip clearance" or "Chip breaking".</p>
V2		<p>Retraction amount (only for chip breaking)</p> <p>Amount by which the drill is retracted for chip breaking.</p> <p>V2=automatically: The tool retracts for one revolution.</p> <p>[mm]</p> <p>Only available for "Chip breaking".</p>

## Section 7

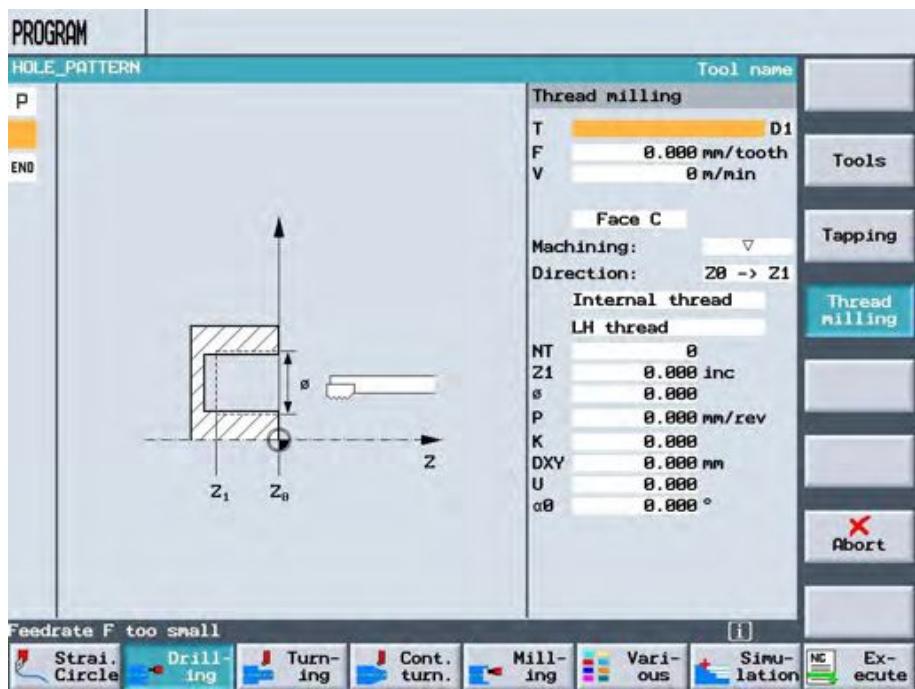
### Thread cutting

#### 7.3 Thread milling

Thread  
milling

Pressing the VSK 4 "thread milling" opens the window for programming external and internal thread milling.

Notes



#### Hints:

Using a thread milling cutter permits the machining of any internal or external thread.

The thread can be cut as a right or left-hand thread.

In case of metric threads (thread pitch P in mm/rev) ShopMill automatically presets the parameter thread depth K with a value evaluated from the thread pitch. This value can be changed. The pre-setting must be activated by a machine datum.

If the values from the tool list and the input mask differ, a fault message is put out.

The documentation by the machine manufacturer must be observed.

## Section 7

Notes

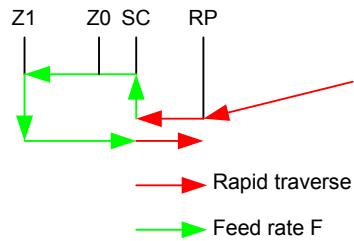
### Thread cutting

Different strategies are envisaged for the milling of internal and external threads:

#### Internal threads:

1. Positioning to the centre of the thread in the retraction plane at rapid traverse rate
2. Infeed to the reference plane, which is raised by the safety distance, with rapid traverse
3. Approach along an entry arc as evaluated internally by the control unit with the programmed feed rate
4. Entry motion to the thread diameter along a circular arc
5. Thread milling along a spiral path either clockwise or anti-clockwise (depending on whether a right-hand or left-hand thread respectively is being cut)
6. Lifting-off motion along a circular arc with the same direction of rotation and the programmed feed rate
7. Retraction from the thread followed by traverse to the retraction plane at rapid traverse rate

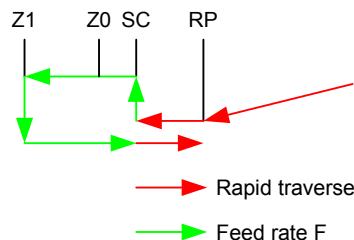
#### Sequence diagram:



#### External thread

1. Positioning to the starting point on the retraction plane with rapid traverse
2. Infeed to the reference plane, which is raised by the safety distance, with rapid traverse
3. Approach along an entry arc as evaluated internally by the control unit with the programmed feed rate
4. Entry motion to the thread diameter along a circular arc to the Z Ø
5. Thread milling along a spiral path either clockwise or anti-clockwise (depending on whether a right-hand or left-hand thread respectively is being cut)
6. Lifting-off motion along a circular arc with the opposite direction of rotation and the programmed feed rate
7. Retraction to the retraction plane with rapid traverse

#### Sequence diagram:



## Section 7

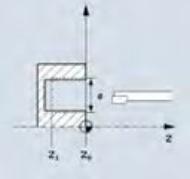
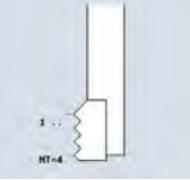
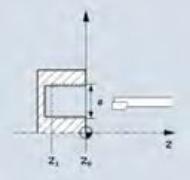
### Thread cutting

Parameter	Help picture	Description and hints	Notes
T, D, F, S / V		See Section 3 "Drilling Centric".	
Position		<p>Selection between 4 different positions:</p> <ul style="list-style-type: none"> <li>Face C - front</li> <li>Face Y - front</li> <li>Perimeter surface C - external</li> <li>Perimeter surface Y - external</li> </ul>	
<p><b>Thread milling</b>  T D1  F 8.000 mm/tooth  V 8 m/min  <b>Face C</b>  Machining: □  Direction: Z0 → Z1  Internal thread  LH thread  NT 8  Z1 0.000 inc  Ø 8.000  P 8.000 mm/zev  K 0.000  DXV 0.000 mm  U 0.000  αB 0.000 °</p>			
<p><b>Thread milling</b>  T D1  F 8.000 mm/tooth  V 8 m/min  <b>Face C</b>  Machining: □  Direction: Z0 → Z1  Internal thread  LH thread  NT 8  Z1 0.000 inc  Ø 8.000  P 8.000 mm/zev  K 0.000  DXV 0.000 mm  U 0.000  αB 0.000 °</p>		<p>Roughing:  Thread milling, leaving the programmed finishing allowance (U)</p> <p>Finishing: </p>	
<p><b>Thread milling</b>  T D1  F 8.000 mm/tooth  V 8 m/min  <b>Face C</b>  Machining: □  Direction: Z0 → Z1  Internal thread  LH thread  NT 8  Z1 0.000 inc  Ø 8.000  P 8.000 mm/zev  K 0.000  DXV 0.000 mm  U 0.000  αB 0.000 °</p>			
<p><b>Thread milling</b>  T D1  F 8.000 mm/tooth  V 8 m/min  <b>Face C</b>  Machining: □  Direction: Z0 → Z1  Internal thread  LH thread  NT 8  Z1 0.000 inc  Ø 8.000  P 8.000 mm/zev  K 0.000  DXV 0.000 mm  U 0.000  αB 0.000 °</p>		<p>If the direction is changed the machining direction of rotation will also be changed depending on the direction of rotation of the spindle (climb milling/conventional milling).</p> <p><b>Z0 → Z1</b> Machining commences at the surface of the work piece Z0.  Only available for "Face C" and "Face Y".</p> <p><b>Z1 → Z0</b> Machining starts from the bottom of the thread, e.g. in case of blind holes  Only available for "Face C" and "Face Y".</p> <p><b>X0 → X1</b> The machining starts at the surface of the work piece X0  Only available for „Perimeter surface Y“.</p> <p><b>X1 → X0</b> The machining starts at the surface of the work piece at the bottom of the thread  Only available for "Perimeter surface Y".</p>	

## Section 7

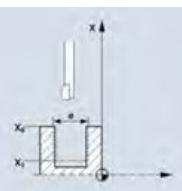
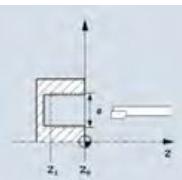
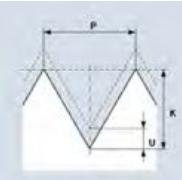
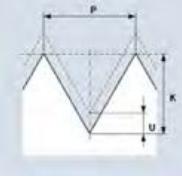
### Thread cutting

Notes

Parameter	Help picture	Description and hints
Internal thread External thread		<p>Internal thread: An internal thread is milled.</p> <p>External thread: An external thread is milled.</p>
Left-hand thread Right-hand thread		<p>Left-hand thread: A left-hand thread is milled.</p> <p>Right-hand thread: A right-hand thread is milled.</p>
NT		<p>Number of teeth of a tool tip.</p> <p>Single or multi-point tool tips can be used. The number of teeth is entered in the parameter NT. The required motions will be carried out in the cycle such that the lowest tooth will coincide with the thread end position. Depending on the tip geometry a suitable relief must be provided at the bottom of the thread.</p>
Z1		<p>Thread length</p> <p>[abs, inc]</p>

## Section 7

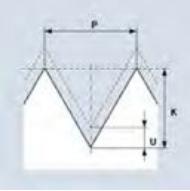
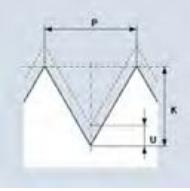
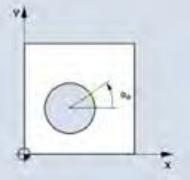
### Thread cutting

Parameter	Help picture	Description and hints	Notes
X1		Thread length [abs, inc]  Only available for "Perimeter surface Y".	
		Nominal diameter of the thread; Example: Nominal diameter of M12 = 12 mm	
P		Thread pitch [mm/rev, inch/U, threads/"]  If the tool tip features several teeth, the thread pitch depends on the used tool.  Threads/": When entries in threads/" are made, the whole number preceding the decimal point must be entered into the first parameter field, while the number following the decimal point must be entered into the second and third field as a fraction. Example: 13,5 Threads/"  <b>P 13 1/ 2 Thrd/s/"</b>  The thread pitches are converted when toggling.	
K		Thread depth [mm]  Hint: Is automatically evaluated if P is in mm/rev	

## Section 7

### Thread cutting

Notes

Parameter	Help picture	Description and hints
DXY		<p>Infeed per cut [mm, %]</p> <p>Only available for „roughing“.</p> <p>Alternatively the plane infeed can also be stated as a percentage of plane infeed (mm) as compared to the milling cutter diameter (mm).</p>
U		<p>Finishing allowance [mm]</p>
0		<p>Starting angle [degrees]</p>

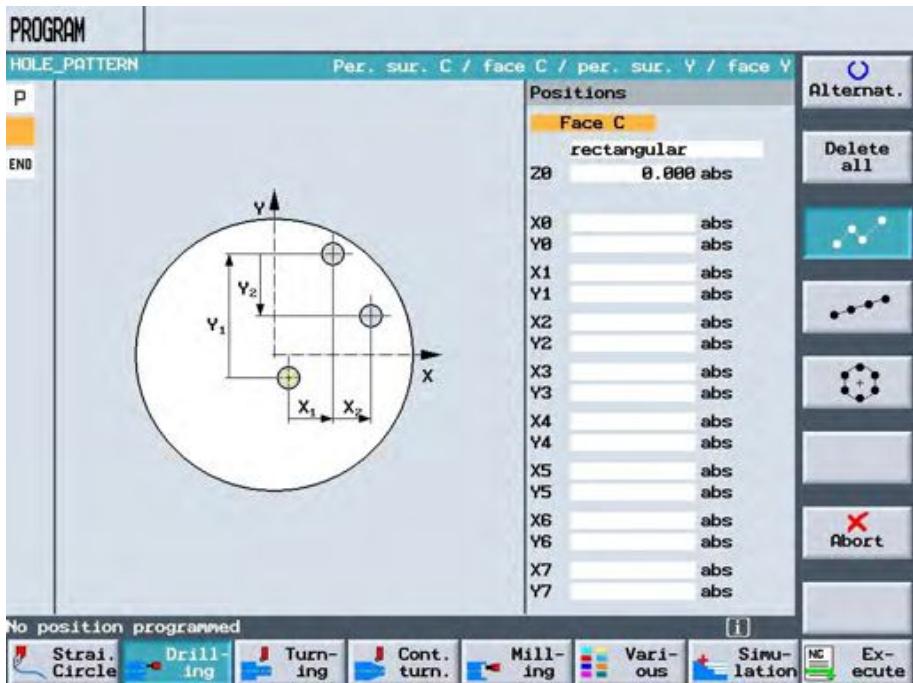
## Section 8

### Position patterns

Positions

Pressing the VSK 7 "Positions" opens the window for the programming of positions and position patterns.

Notes



After the technologies (e.g. centring, drilling, tapping) have been programmed it is now necessary to program the positions. ShopMill provides the following position patterns:

Any position

Positioning along a line, on a grid or on a frame

Positioning on a full or part circle

Several position patterns can be programmed one after the other (max. 20 technologies and position patterns altogether).

The position patterns will be executed in the programmed sequence.

The previously programmed technologies and position patterns will be interlinked.

## Section 8

Notes

### Position patterns

#### Machining sequence and tool traverses

1. Firstly all positions will be approached with the first tool, e.g. centring at all positions.
2. Machining always commences at the reference point.

Any position:  
as per the programmed position

Line and grid:

Starting from the reference point;  
in case of grid, traverse in direction of the  
1<sup>st</sup> axis and then in a meandering fashion

Full and partial circle:  
the positions are approached in anti-clockwise direction

3. Thereafter this sequence is repeated with the next programmed tool, e.g. for drilling.
4. This procedure is repeated until all machining technologies have been completed.

Within a position pattern as well as the change-over from one position pattern to the next, the optimized retraction takes place as far as the safety level, in all other cases to the retraction plane.

Following this the next position is approached at the rapid traverse rate.

If a position pattern entails but one position, the retraction after machining takes place to the retraction plane.

#### 8.1 Vertical Softkey strip

Delete all

Pressing the VSK 2 "Delete all", all programmed arbitrary drilling positions will be deleted.

This function is available only for the programming of arbitrary drilling positions (VSK 3).



Pressing the VSK 3 "Any positions" calls up the function for the programming of rectangular or polar dimensioned positions.

Up to 8 positions can be entered at a time.

*See Section 8.2 „Rectangular / polar positions“.*



Pressing the VSK 4 "Line, grid, frame" calls up the function for the programming of linear drilling patterns.

*See Section 8.3 "Line, grid, frame".*



Pressing the VSK 5 "Full circle, part circle" calls up the function for the programming of circular drilling patterns.

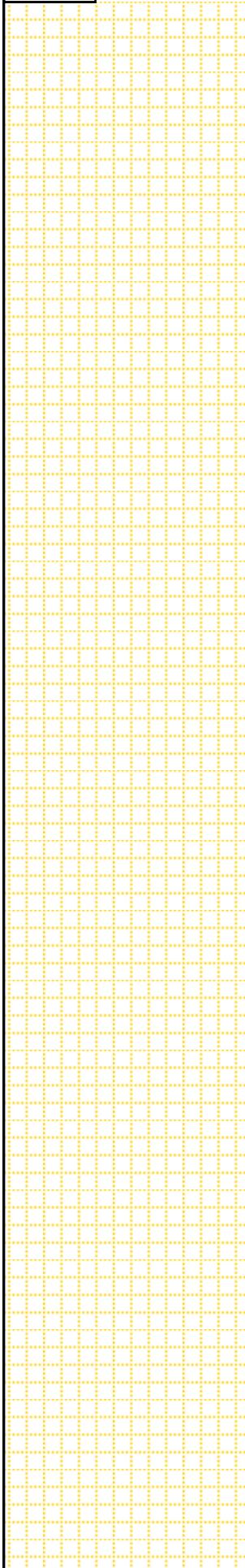
*See Section 8.4 "Full and partial circle".*

#### Hint:

If further positions or position patterns are required, these can be programmed directly following the 1<sup>st</sup> position pattern.

The parameter Z0 and the first position of the reference point can be dimensioned incrementally if the previous absolute position in the program was defined for instance by a straight line. If directly after the 1<sup>st</sup> position pattern a further position pattern is to be programmed, the first position or the reference point must be programmed with absolute dimensions.

Notes

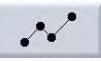


## Section 8

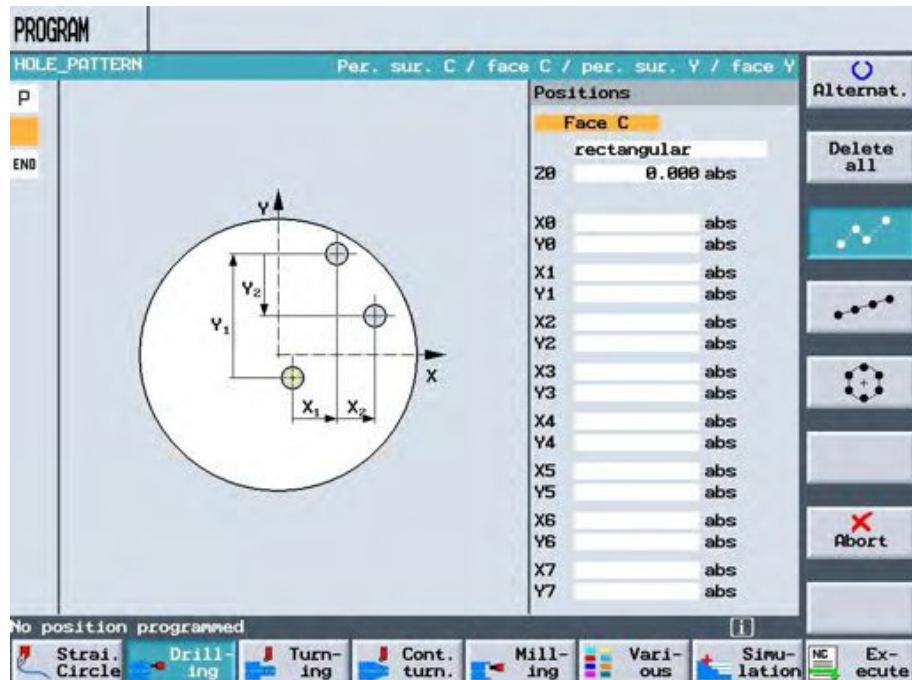
Notes

Position patterns

### 8.2 Rectangular / polar positions

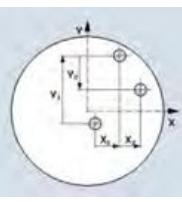
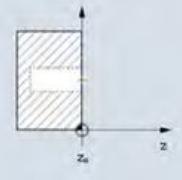
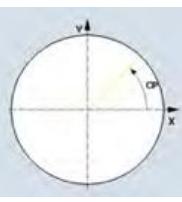
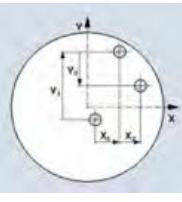


Pressing the VSK 3 "Any position" calls up the function for the programming of rectangular or polar dimensioned positions.



## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes
Position		Selection between 4 different positions: Face C - front Face Y - front Perimeter surface C - external Perimeter surface Y - external	
Rectangular, Polar		Selection of rectangular and polar dimensioning	
<b>Face C and Face Y - rectangular:</b>			
Z0		Z co-ordinate of the reference point [abs]	
CP		Positioning angle of the machining range [Grad] Only available for "Face Y".	
X0, Y0		1 <sup>st</sup> hole position in X 1 <sup>st</sup> hole position in Y [abs, inc]	

## Section 8

### Position patterns

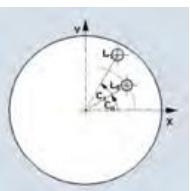
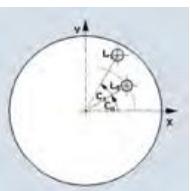
Notes

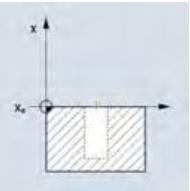
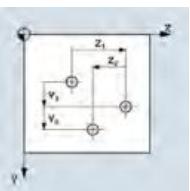
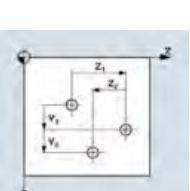
Parameter	Help picture	Description and hints																
X1 ... X7 Y1 ... Y7	<p><b>Positions</b></p> <table border="1"> <tr><td>Face C</td><td>rectangular</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>X8</td><td>abs</td></tr> <tr><td>Y8</td><td>abs</td></tr> <tr><td>X1</td><td>abs</td></tr> <tr><td>Y1</td><td>abs</td></tr> <tr><td>X2</td><td>abs</td></tr> <tr><td>Y2</td><td>abs</td></tr> </table>	Face C	rectangular	Z8	0.000 abs	X8	abs	Y8	abs	X1	abs	Y1	abs	X2	abs	Y2	abs	<p>Further positions in the X- and Y-axis [abs, inc]</p> <p>If further positions have to be programmed, the positions programmed so far must be saved first. Then re-open this input mask once more with the VSK 3 "Any position".</p>
Face C	rectangular																	
Z8	0.000 abs																	
X8	abs																	
Y8	abs																	
X1	abs																	
Y1	abs																	
X2	abs																	
Y2	abs																	

Face C and Face Y - polar:																				
Z0	<p><b>Positions</b></p> <table border="1"> <tr><td>Face C</td><td>Polar</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>C8</td><td>abs</td></tr> <tr><td>L8</td><td>abs</td></tr> <tr><td>C1</td><td>abs</td></tr> <tr><td>L1</td><td>abs</td></tr> <tr><td>C2</td><td>abs</td></tr> <tr><td>L2</td><td>abs</td></tr> </table>	Face C	Polar	Z8	0.000 abs	C8	abs	L8	abs	C1	abs	L1	abs	C2	abs	L2	abs	Z co-ordinate of the reference point [abs]		
Face C	Polar																			
Z8	0.000 abs																			
C8	abs																			
L8	abs																			
C1	abs																			
L1	abs																			
C2	abs																			
L2	abs																			
CP	<p><b>Positions</b></p> <table border="1"> <tr><td>Face Y</td><td>Polar</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>CP</td><td>0.000 °</td></tr> <tr><td>C8</td><td>abs</td></tr> <tr><td>L8</td><td>abs</td></tr> <tr><td>C1</td><td>abs</td></tr> <tr><td>L1</td><td>abs</td></tr> <tr><td>C2</td><td>abs</td></tr> <tr><td>L2</td><td>abs</td></tr> </table>	Face Y	Polar	Z8	0.000 abs	CP	0.000 °	C8	abs	L8	abs	C1	abs	L1	abs	C2	abs	L2	abs	<p>Positioning angle for the machining range [degrees]</p> <p>Only available for "Face Y".</p>
Face Y	Polar																			
Z8	0.000 abs																			
CP	0.000 °																			
C8	abs																			
L8	abs																			
C1	abs																			
L1	abs																			
C2	abs																			
L2	abs																			
C0	<p><b>Positions</b></p> <table border="1"> <tr><td>Face C</td><td>Polar</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>C8</td><td>abs</td></tr> <tr><td>L8</td><td>abs</td></tr> <tr><td>C1</td><td>abs</td></tr> <tr><td>L1</td><td>abs</td></tr> <tr><td>C2</td><td>abs</td></tr> <tr><td>L2</td><td>abs</td></tr> </table>	Face C	Polar	Z8	0.000 abs	C8	abs	L8	abs	C1	abs	L1	abs	C2	abs	L2	abs	C co-ordinate of the 1st position [abs]		
Face C	Polar																			
Z8	0.000 abs																			
C8	abs																			
L8	abs																			
C1	abs																			
L1	abs																			
C2	abs																			
L2	abs																			
L0	<p><b>Positions</b></p> <table border="1"> <tr><td>Face C</td><td>Polar</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>C8</td><td>abs</td></tr> <tr><td>L8</td><td>abs</td></tr> <tr><td>C1</td><td>abs</td></tr> <tr><td>L1</td><td>abs</td></tr> <tr><td>C2</td><td>abs</td></tr> <tr><td>L2</td><td>abs</td></tr> </table>	Face C	Polar	Z8	0.000 abs	C8	abs	L8	abs	C1	abs	L1	abs	C2	abs	L2	abs	1st position of the hole referred to the Y-axis [abs]		
Face C	Polar																			
Z8	0.000 abs																			
C8	abs																			
L8	abs																			
C1	abs																			
L1	abs																			
C2	abs																			
L2	abs																			

## Section 8

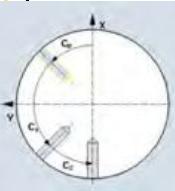
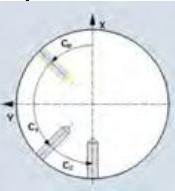
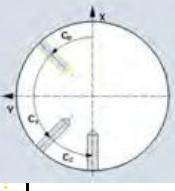
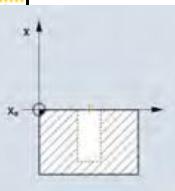
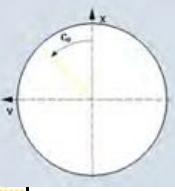
### Position patterns

Parameter	Help picture	Description and hints	Notes
C1 ... C7		<p>C co-ordinate of further positions [abs, inc]</p> <p>If further positions have to be programmed, the positions programmed so far must be saved first. Then re-open this input mask once more with the VSK 3 "Any position".</p>	
L1 ... L7		<p>Distance of the position [abs, inc]</p> <p>If further positions have to be programmed, the positions programmed so far must be saved first. Then re-open this input mask once more with the VSK 3 "Any position".</p>	

Perimeter surface C - rectangular:			
X0		Cylinder diameter $\phi$ [abs]	
Y0, Z0		1 <sup>st</sup> hole position in Y 1 <sup>st</sup> hole position in Z [abs]	
Y1 ... Y7, Z1 ... Z7		Further positions in the Y- and Z-axis [abs, inc]	

## Section 8

### Position patterns

Parameter	Help picture	Description and hints
Perimeter surface C - cylindrical:		
X0		See "Perimeter surface C - rectangular".
C0		C co-ordinate of the 1st position [abs]
Z0		1st position of the hole referred to the Z-axis [abs]
C1 ... C7, Z1 ... Z7		C co-ordinate of further positions, Further positions in the Z-axis [abs, inc]
Perimeter surface Y:		
X0		Reference point in X-direction [abs]
C0		Reference point

## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes																
Y0, Z0	<p><b>Positions</b> Per.surf.V</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td><b>Y0</b></td><td>abs</td></tr> <tr><td>Z0</td><td>abs</td></tr> <tr><td>Y1</td><td>abs</td></tr> <tr><td>Z1</td><td>abs</td></tr> <tr><td>Y2</td><td>abs</td></tr> <tr><td>Z2</td><td>abs</td></tr> </table>	X0	0.000 abs	C0	0.000 °	<b>Y0</b>	abs	Z0	abs	Y1	abs	Z1	abs	Y2	abs	Z2	abs	1 <sup>st</sup> hole position in Y 1 <sup>st</sup> hole position in Z [abs]	
X0	0.000 abs																		
C0	0.000 °																		
<b>Y0</b>	abs																		
Z0	abs																		
Y1	abs																		
Z1	abs																		
Y2	abs																		
Z2	abs																		
Y1 ... Y7, Z1 ... Z7	<p><b>Positions</b> Per.surf.V</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td><b>Y0</b></td><td>abs</td></tr> <tr><td><b>Z0</b></td><td>abs</td></tr> <tr><td>Y1</td><td>abs</td></tr> <tr><td>Z1</td><td>abs</td></tr> <tr><td>Y2</td><td>abs</td></tr> <tr><td>Z2</td><td>abs</td></tr> </table>	X0	0.000 abs	C0	0.000 °	<b>Y0</b>	abs	<b>Z0</b>	abs	Y1	abs	Z1	abs	Y2	abs	Z2	abs	Further positions in the Y- and Z-axis [abs, inc]	
X0	0.000 abs																		
C0	0.000 °																		
<b>Y0</b>	abs																		
<b>Z0</b>	abs																		
Y1	abs																		
Z1	abs																		
Y2	abs																		
Z2	abs																		

## Section 8

Notes

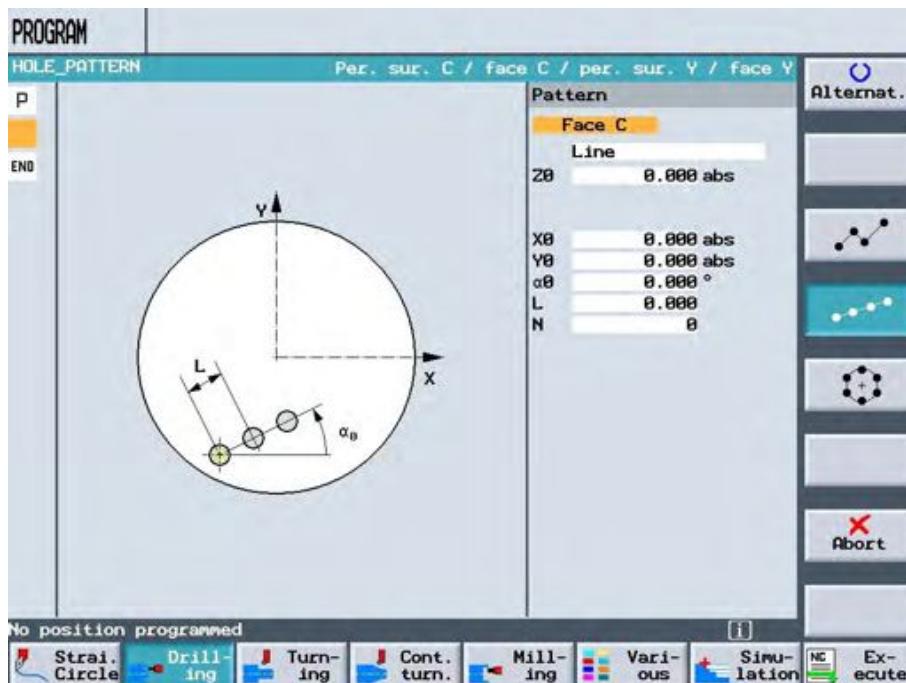
### Position patterns

#### 8.3 Line, grid



Pressing the VSK 4 "Line, grid" calls up the function for the programming of linear position patterns.

These positions can be arranged along a line, on a grid or a frame.



Parameter	Help picture	Description and hints
Position  Pattern Face C Line Z0 0.000 abs  X0 0.000 abs Y0 0.000 abs α0 0.000 ° L 0.000 N 0		Selection between 4 different positions: Face C - front Face Y - front Perimeter surface C - external Perimeter surface Y - external

Face C:	Line
 Pattern Face C Line Z0 0.000 abs  X0 0.000 abs Y0 0.000 abs α0 0.000 ° L 0.000 N 0	

## Section 8

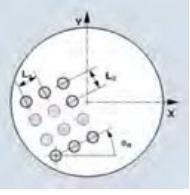
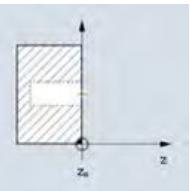
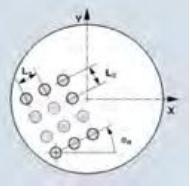
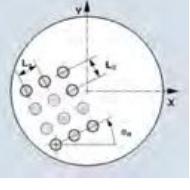
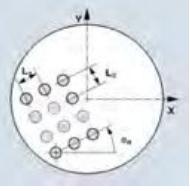
### Position patterns

Parameter	Help picture	Description and hints	Notes																
Z0	<p><b>Pattern</b></p> <table border="1"> <tr><td>Face C</td><td></td></tr> <tr><td>Line</td><td></td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	Face C		Line		Z0	0.000 abs	X0	0.000 abs	Y0	0.000 abs	α0	0.000 °	L	0.000	N	0	Z co-ordinate of the reference point [abs]	
Face C																			
Line																			
Z0	0.000 abs																		
X0	0.000 abs																		
Y0	0.000 abs																		
α0	0.000 °																		
L	0.000																		
N	0																		
X0, Y0	<p><b>Pattern</b></p> <table border="1"> <tr><td>Face C</td><td></td></tr> <tr><td>Line</td><td></td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	Face C		Line		Z0	0.000 abs	X0	0.000 abs	Y0	0.000 abs	α0	0.000 °	L	0.000	N	0	Reference point (first position) [abs, inc]  For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions.  [abs]	
Face C																			
Line																			
Z0	0.000 abs																		
X0	0.000 abs																		
Y0	0.000 abs																		
α0	0.000 °																		
L	0.000																		
N	0																		
	<p><b>Pattern</b></p> <table border="1"> <tr><td>Face C</td><td></td></tr> <tr><td>Line</td><td></td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	Face C		Line		Z0	0.000 abs	X0	0.000 abs	Y0	0.000 abs	α0	0.000 °	L	0.000	N	0	Angle of rotation of the line referred to the X-axis.  [Degrees]  Positive angle: The line rotates anti-clockwise.  Negative angle: The line rotates clockwise.	
Face C																			
Line																			
Z0	0.000 abs																		
X0	0.000 abs																		
Y0	0.000 abs																		
α0	0.000 °																		
L	0.000																		
N	0																		
L	<p><b>Pattern</b></p> <table border="1"> <tr><td>Face C</td><td></td></tr> <tr><td>Line</td><td></td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	Face C		Line		Z0	0.000 abs	X0	0.000 abs	Y0	0.000 abs	α0	0.000 °	L	0.000	N	0	Distance between the positions	
Face C																			
Line																			
Z0	0.000 abs																		
X0	0.000 abs																		
Y0	0.000 abs																		
α0	0.000 °																		
L	0.000																		
N	0																		
N	<p><b>Pattern</b></p> <table border="1"> <tr><td>Face C</td><td></td></tr> <tr><td>Line</td><td></td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	Face C		Line		Z0	0.000 abs	X0	0.000 abs	Y0	0.000 abs	α0	0.000 °	L	0.000	N	0	Number of positions	
Face C																			
Line																			
Z0	0.000 abs																		
X0	0.000 abs																		
Y0	0.000 abs																		
α0	0.000 °																		
L	0.000																		
N	0																		

## Section 8

Notes

### Position patterns

Parameter	Help picture	Description and hints
<b>Grid</b>		
<b>Pattern</b> Face C <b>Grid</b> Z0    0.000 abs X0    0.000 abs Y0    0.000 abs oθ    0.000 ° L1    0.000 L2    0.000 N1    0 N2    0		Grid
<b>Z0</b> <b>Pattern</b> Face C <b>Grid</b> <b>Z0</b> 0.000 abs X0    0.000 abs Y0    0.000 abs oθ    0.000 ° L1    0.000 L2    0.000 N1    0 N2    0		Z co-ordinate of the reference point [abs]
<b>X0, Y0</b> <b>Pattern</b> Face C <b>Grid</b> Z0    0.000 abs <b>X0</b> 0.000 abs <b>Y0</b> 0.000 abs oθ    0.000 ° L1    0.000 L2    0.000 N1    0 N2    0		Reference point (first position) [abs, inc]  For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions.
<b>o</b> <b>Pattern</b> Face C <b>Grid</b> Z0    0.000 abs X0    0.000 abs <b>oθ</b> 0.000 ° L1    0.000 L2    0.000 N1    0 N2    0		Angle of rotation of the grid/frame [Degrees]  Positive angle: The grid rotates anti-clockwise.  Negative angle: The grid rotates clockwise.
<b>L1, L2</b> <b>Pattern</b> Face C <b>Grid</b> Z0    0.000 abs X0    0.000 abs Y0    0.000 abs oθ    0.000 ° <b>L1</b> 0.000 L2    0.000 N1    0 N2    0		Distance of the positions in X-direction Distance of the positions in Y-direction [mm]

## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes											
N1, N2	<p><b>Pattern</b></p> <table border="1"> <tr><td>Face C</td></tr> <tr><td>Grid</td></tr> <tr><td>ZB 0.000 abs</td></tr> <tr><td> </td></tr> <tr><td>XB 0.000 abs</td></tr> <tr><td>YB 0.000 abs</td></tr> <tr><td>αB 0.000 °</td></tr> <tr><td>L1 0.000</td></tr> <tr><td>L2 0.000</td></tr> <tr><td>N1 0</td></tr> <tr><td>N2 0</td></tr> </table>	Face C	Grid	ZB 0.000 abs		XB 0.000 abs	YB 0.000 abs	αB 0.000 °	L1 0.000	L2 0.000	N1 0	N2 0	Number of positions in X-direction Number of positions in Y-direction	
Face C														
Grid														
ZB 0.000 abs														
XB 0.000 abs														
YB 0.000 abs														
αB 0.000 °														
L1 0.000														
L2 0.000														
N1 0														
N2 0														
Perimeter surface C:														
Line	<p><b>Pattern</b></p> <table border="1"> <tr><td>Per.surf.C</td></tr> <tr><td>Line</td></tr> <tr><td>XB 0.000 abs</td></tr> <tr><td> </td></tr> <tr><td>YB 0.000 abs</td></tr> <tr><td>ZB 0.000 abs</td></tr> <tr><td>αB 0.000 °</td></tr> <tr><td>L 0.000</td></tr> <tr><td>N 8</td></tr> </table>	Per.surf.C	Line	XB 0.000 abs		YB 0.000 abs	ZB 0.000 abs	αB 0.000 °	L 0.000	N 8	Line			
Per.surf.C														
Line														
XB 0.000 abs														
YB 0.000 abs														
ZB 0.000 abs														
αB 0.000 °														
L 0.000														
N 8														
X0	<p><b>Pattern</b></p> <table border="1"> <tr><td>Per.surf.C</td></tr> <tr><td>Line</td></tr> <tr><td>XB 0.000 abs</td></tr> <tr><td> </td></tr> <tr><td>YB 0.000 abs</td></tr> <tr><td>ZB 0.000 abs</td></tr> <tr><td>αB 0.000 °</td></tr> <tr><td>L 0.000</td></tr> <tr><td>N 8</td></tr> </table>	Per.surf.C	Line	XB 0.000 abs		YB 0.000 abs	ZB 0.000 abs	αB 0.000 °	L 0.000	N 8	Cylinder diameter $\emptyset$ [abs]			
Per.surf.C														
Line														
XB 0.000 abs														
YB 0.000 abs														
ZB 0.000 abs														
αB 0.000 °														
L 0.000														
N 8														
Y0, Z0	<p><b>Pattern</b></p> <table border="1"> <tr><td>Per.surf.C</td></tr> <tr><td>Line</td></tr> <tr><td>XB 0.000 abs</td></tr> <tr><td> </td></tr> <tr><td>YB 0.000 abs</td></tr> <tr><td>ZB 0.000 abs</td></tr> <tr><td>αB 0.000 °</td></tr> <tr><td>L 0.000</td></tr> <tr><td>N 8</td></tr> </table>	Per.surf.C	Line	XB 0.000 abs		YB 0.000 abs	ZB 0.000 abs	αB 0.000 °	L 0.000	N 8	Reference point (first position) [abs, inc]  For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions.			
Per.surf.C														
Line														
XB 0.000 abs														
YB 0.000 abs														
ZB 0.000 abs														
αB 0.000 °														
L 0.000														
N 8														
0	<p><b>Pattern</b></p> <table border="1"> <tr><td>Per.surf.C</td></tr> <tr><td>Line</td></tr> <tr><td>XB 0.000 abs</td></tr> <tr><td> </td></tr> <tr><td>YB 0.000 abs</td></tr> <tr><td>ZB 0.000 abs</td></tr> <tr><td>αB 0.000 °</td></tr> <tr><td>L 0.000</td></tr> <tr><td>N 8</td></tr> </table>	Per.surf.C	Line	XB 0.000 abs		YB 0.000 abs	ZB 0.000 abs	αB 0.000 °	L 0.000	N 8	Angle of rotation of the grid/frame [Degrees]  Positive angle: The grid rotates anti-clockwise.  Negative angle: The grid rotates clockwise.			
Per.surf.C														
Line														
XB 0.000 abs														
YB 0.000 abs														
ZB 0.000 abs														
αB 0.000 °														
L 0.000														
N 8														

## Section 8

### Position patterns

Notes

Parameter	Help picture	Description and hints
L	<p><b>Pattern</b> Per.surf.C Line X0 0.000 abs Y0 0.000 abs Z0 0.000 abs a0 0.000 ° L 0.000 N 0</p>	Distance of the positions

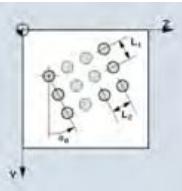
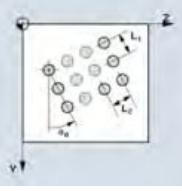
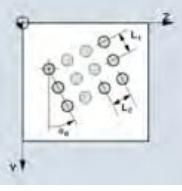
N	<p><b>Pattern</b> Per.surf.C Line X0 0.000 abs Y0 0.000 abs Z0 0.000 abs a0 0.000 ° L 0.000 N 0</p>	Number of positions
---	-------------------------------------------------------------------------------------------------------------------------------------	---------------------

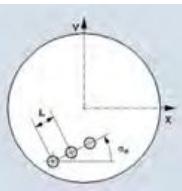
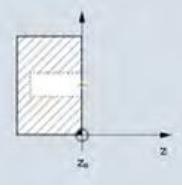
grid	<p><b>Pattern</b> Per.surf.C Grid X0 0.000 abs Y0 0.000 abs Z0 0.000 abs a0 0.000 ° L1 0.000 L2 0.000 N1 0 N2 0</p>	grid
X0	<p><b>Pattern</b> Per.surf.C Grid X0 0.000 abs Y0 0.000 abs Z0 0.000 abs a0 0.000 ° L1 0.000 L2 0.000 N1 0 N2 0</p>	Cylinder diameter $\emptyset$ [abs]
Y0, Z0	<p><b>Pattern</b> Per.surf.C Grid X0 0.000 abs Y0 0.000 abs Z0 0.000 abs a0 0.000 ° L1 0.000 L2 0.000 N1 0 N2 0</p>	Reference point (first position) [abs, inc] For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions.

## Section 8

### Position patterns

Notes

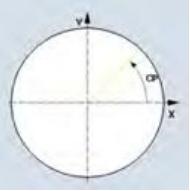
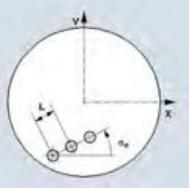
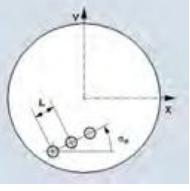
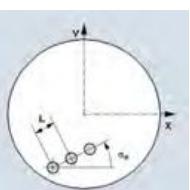
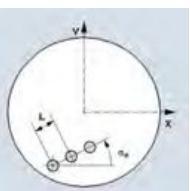
Parameter	Help picture	Description and hints
0		Angle of rotation of the grid/frame [Degrees] Positive angle: The grid rotates anti-clockwise. Negative angle: The grid rotates clockwise.
L1, L2		Distance of the positions in Y-direction Distance of the positions in Z-direction [mm]
N1, N2		Number of positions in Y-direction Number of positions in Z-direction

Face Y:		
Line		
<b>Pattern</b> Face Y Line ZB 0.000 abs CP 0.000 ° XB 0.000 abs YB 0.000 abs αB 0.000 ° L 0.000 N 0		Line
<b>Pattern</b> Face Y Line ZB 0.000 abs CP 0.000 ° XB 0.000 abs YB 0.000 abs αB 0.000 ° L 0.000 N 0		Z co-ordinate of the reference point [abs]

## Section 8

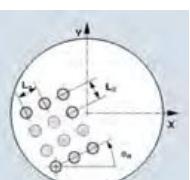
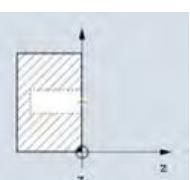
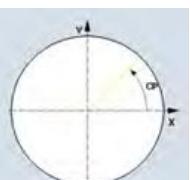
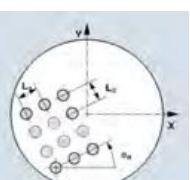
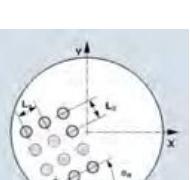
Notes

### Position patterns

Parameter	Help picture	Description and hints
CP		Positioning angle for machining range
X0, Y0		Reference point (first position) [abs, inc]  For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions.
0		Angle of rotation of the grid/frame [Degrees]  Positive angle: The grid rotates anti-clockwise.  Negative angle: The grid rotates clockwise.
L		Distance of the positions
N		Number of positions

## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes
Face Y:			
Grid			
<b>Pattern</b> Face Y Grid Z0 0.000 abs CP 0.000 ° X0 0.000 abs Y0 0.000 abs oθ 0.000 ° L1 0.000 L2 0.000 N1 0 N2 0		Grid	
Z0		Z co-ordinate of the reference point [abs]	
CP		Positioning angle for the machining range	
X0, Y0		Reference point (first position) [abs, inc]  For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions.	
0		Angle of rotation of the grid/frame [Degrees]  Positive angle: The grid rotates anti-clockwise.  Negative angle: The grid rotates clockwise.	

## Section 8

### Position patterns

Notes

Parameter	Help picture	Description and hints
L1, L2  <b>Pattern</b> Face Y Grid Z0 0.000 abs CP 0.000 ° X0 0.000 abs Y0 0.000 abs α0 0.000 ° L1 <b>0.000</b> L2 0.000 N1 0 N2 0		Distance of the positions in Y-direction Distance of the positions in Z-direction [mm]
N1, N2  <b>Pattern</b> Face Y Grid Z0 0.000 abs CP 0.000 ° X0 0.000 abs Y0 0.000 abs α0 0.000 ° L1 0.000 L2 0.000 N1 <b>0</b> N2 0		Number of positions in Y-direction Number of positions in Z-direction
Perimeter surface Y:		
Line  <b>Pattern</b> Per.surf.Y Line X0 0.000 abs C0 0.000 ° Y0 0.000 abs Z0 0.000 abs α0 0.000 ° L 0.000 H 0		Line
X0  <b>Pattern</b> Per.surf.Y Line X0 <b>0.000 abs</b> C0 0.000 ° Y0 0.000 abs Z0 0.000 abs α0 0.000 ° L 0.000 H 0		X co-ordinate of the reference point [abs]
C0  <b>Pattern</b> Per.surf.Y Line X0 0.000 abs C0 <b>0.000 °</b> Y0 0.000 abs Z0 0.000 abs α0 0.000 ° L 0.000 H 0		Reference point

## Section 8

### Position patterns

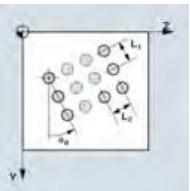
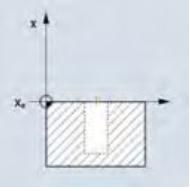
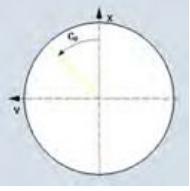
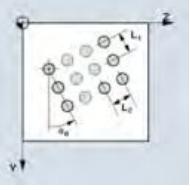
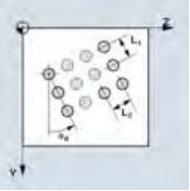
Notes

Parameter	Help picture	Description and hints														
Y0, Z0	<p><b>Pattern</b></p> <p>Per.surf.V</p> <p>Line</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td>V0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	X0	0.000 abs	C0	0.000 °	V0	0.000 abs	Z0	0.000 abs	α0	0.000 °	L	0.000	N	0	<p>Reference point (first position)</p> <p>[abs, inc]</p> <p>For the 1<sup>st</sup> call-up this position must be programmed with absolute dimensions.</p>
X0	0.000 abs															
C0	0.000 °															
V0	0.000 abs															
Z0	0.000 abs															
α0	0.000 °															
L	0.000															
N	0															
0	<p><b>Pattern</b></p> <p>Per.surf.V</p> <p>Line</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td>V0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	X0	0.000 abs	C0	0.000 °	V0	0.000 abs	Z0	0.000 abs	α0	0.000 °	L	0.000	N	0	<p>Angle of rotation of the grid/frame</p> <p>[Degrees]</p> <p>Positive angle: The grid rotates anti-clockwise.</p> <p>Negative angle: The grid rotates clockwise.</p>
X0	0.000 abs															
C0	0.000 °															
V0	0.000 abs															
Z0	0.000 abs															
α0	0.000 °															
L	0.000															
N	0															
L	<p><b>Pattern</b></p> <p>Per.surf.V</p> <p>Line</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td>V0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	X0	0.000 abs	C0	0.000 °	V0	0.000 abs	Z0	0.000 abs	α0	0.000 °	L	0.000	N	0	Distance of the positions
X0	0.000 abs															
C0	0.000 °															
V0	0.000 abs															
Z0	0.000 abs															
α0	0.000 °															
L	0.000															
N	0															
N	<p><b>Pattern</b></p> <p>Per.surf.V</p> <p>Line</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td>V0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>α0</td><td>0.000 °</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>N</td><td>0</td></tr> </table>	X0	0.000 abs	C0	0.000 °	V0	0.000 abs	Z0	0.000 abs	α0	0.000 °	L	0.000	N	0	Number of positions
X0	0.000 abs															
C0	0.000 °															
V0	0.000 abs															
Z0	0.000 abs															
α0	0.000 °															
L	0.000															
N	0															

## Section 8

### Position patterns

Notes

Parameter	Help picture	Description and hints
Perimeter surface Y:		
Grid		grid
X0		X co-ordinate of the reference point [abs]
C0		Reference point
Y0, Z0		Reference point (first position) [abs, inc]  For the 1 <sup>st</sup> call-up this position must be programmed with absolute dimensions
0		Angle of rotation of the grid/frame [Degrees]  Positive angle: The grid rotates anti-clockwise.  Negative angle: The grid rotates clockwise.

## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes																		
L1, L2	<p><b>Pattern</b></p> <p>Per.surf.Y</p> <p>Grid</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>o0</td><td>0.000 °</td></tr> <tr><td>L1</td><td>0.000</td></tr> <tr><td>L2</td><td>0.000</td></tr> <tr><td>N1</td><td>0</td></tr> <tr><td>N2</td><td>0</td></tr> </table>	X0	0.000 abs	C0	0.000 °	Y0	0.000 abs	Z0	0.000 abs	o0	0.000 °	L1	0.000	L2	0.000	N1	0	N2	0	Distance of the positions in Y-direction Distance of the positions in Z-direction [mm]	
X0	0.000 abs																				
C0	0.000 °																				
Y0	0.000 abs																				
Z0	0.000 abs																				
o0	0.000 °																				
L1	0.000																				
L2	0.000																				
N1	0																				
N2	0																				
N1, N2	<p><b>Pattern</b></p> <p>Per.surf.Y</p> <p>Grid</p> <table border="1"> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>C0</td><td>0.000 °</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>o0</td><td>0.000 °</td></tr> <tr><td>L1</td><td>0.000</td></tr> <tr><td>L2</td><td>0.000</td></tr> <tr><td>N1</td><td>0</td></tr> <tr><td>N2</td><td>0</td></tr> </table>	X0	0.000 abs	C0	0.000 °	Y0	0.000 abs	Z0	0.000 abs	o0	0.000 °	L1	0.000	L2	0.000	N1	0	N2	0	Number of positions in Y-direction Number of positions in Z-direction	
X0	0.000 abs																				
C0	0.000 °																				
Y0	0.000 abs																				
Z0	0.000 abs																				
o0	0.000 °																				
L1	0.000																				
L2	0.000																				
N1	0																				
N2	0																				

## Section 8

Notes

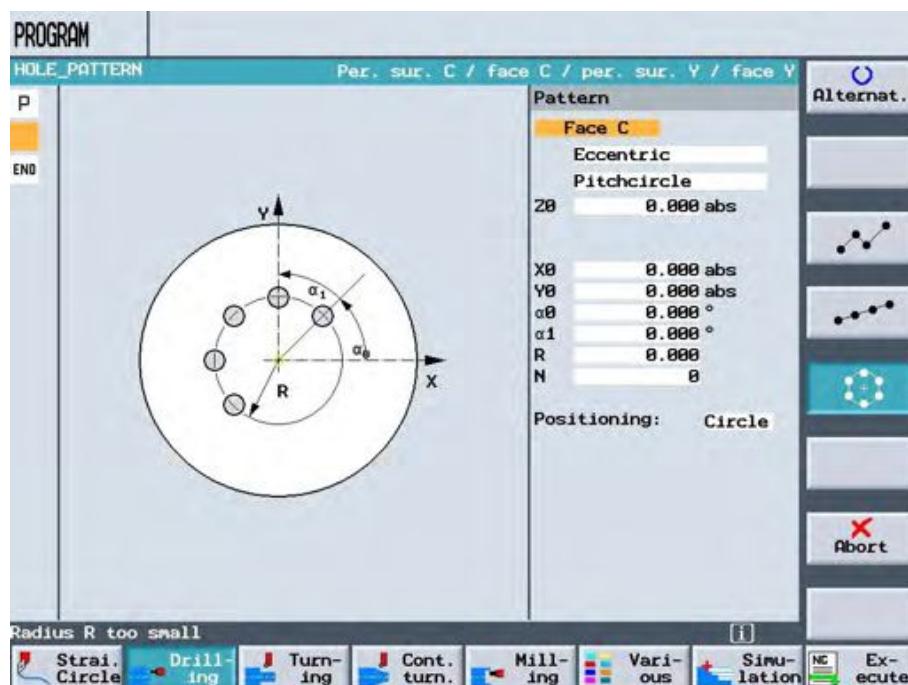
### Position patterns

#### 8.4 Full and partial circle

With this function holes can be programmed on a circle with a defined radius. The basic angle of rotation ( $\alpha_0$ ) for the 1<sup>st</sup> position is referred to the X-axis. The control unit progresses the axes by a constant angle depending on the number of holes. This angle is the same for all positions.

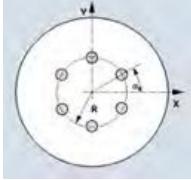
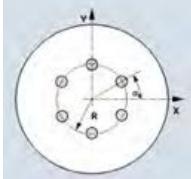
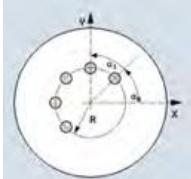
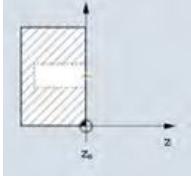
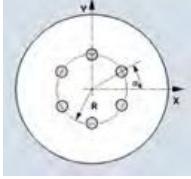


Pressing the VSK 5 "Full circle, partial circle" calls up the function for the programming of circular position patterns.



## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes
Face C:			
Concentric, eccentric	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face C</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB      0.000 abs</p> <p>XB      0.000 abs YB      0.000 abs αB      0.000 °</p> <p>R      0.000 N      8</p> <p>Positioning: Circle</p>	 <p>Full circle located concentrically on the face, Full circle located eccentrically on the face</p>	
Full circle, Partial circle	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face C</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB      0.000 abs</p> <p>XB      0.000 abs YB      0.000 abs αB      0.000 °</p> <p>R      0.000 N      8</p> <p>Positioning: Circle</p>	 <p>Full circle</p>  <p>Partial circle</p>	
Z0	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face C</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB      0.000 abs</p> <p>XB      0.000 abs YB      0.000 abs αB      0.000 °</p> <p>R      0.000 N      8</p> <p>Positioning: Circle</p>	 <p>Z co-ordinate of the reference point [abs]</p>	
X0, Y0	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face C</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB      0.000 abs</p> <p>XB      0.000 abs YB      0.000 abs αB      0.000 °</p> <p>R      0.000 N      8</p> <p>Positioning: Circle</p>	 <p>Reference point, position of the circle centre [abs]</p> <p>Only available for „Eccentric“.</p>	

## Section 8

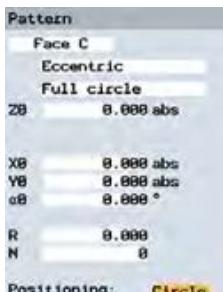
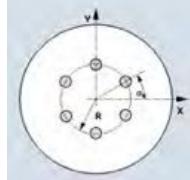
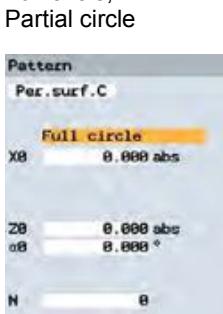
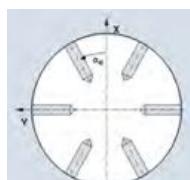
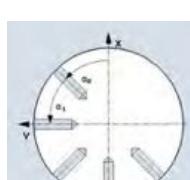
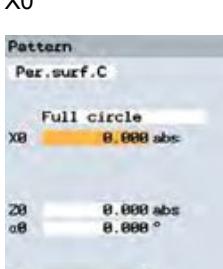
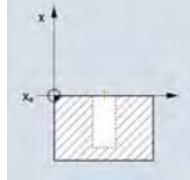
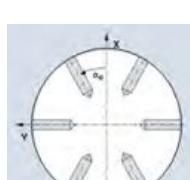
Notes

### Position patterns

Parameter	Help picture	Description and hints
0  <b>Pattern</b> Face C Eccentric Full circle Z0 0.000 abs  X0 0.000 abs Y0 0.000 abs a0 0.000 °  R 0.000 N 0  Positioning: Circle		<p>Start angle; angle of the 1<sup>st</sup> hole as referred to the X-axis.</p> <p>[Degrees]</p> <p>Positive angle: The circle is rotated anti-clockwise.</p> <p>Negative angle: The circle is rotated clockwise.</p>
1  <b>Pattern</b> Face C Eccentric Pitchcircle Z0 0.000 abs  X0 0.000 abs Y0 0.000 abs a0 0.000 ° c1 0.000 °  R 0.000 N 0  Positioning: Circle		<p>Progression angle</p> <p>[Degrees]</p> <p>After the first hole has been machined all further positions will be positioned onwards by this angle.</p> <p>Positive angle: The circle is rotated anti-clockwise.</p> <p>Negative angle: The circle is rotated clockwise.</p>
R  <b>Pattern</b> Face C Eccentric Full circle Z0 0.000 abs  X0 0.000 abs Y0 0.000 abs a0 0.000 °  R 0.000 N 0  Positioning: Circle		<p>Radius of the circle</p> <p>[mm]</p>
N  <b>Pattern</b> Face C Eccentric Full circle Z0 0.000 abs  X0 0.000 abs Y0 0.000 abs a0 0.000 °  R 0.000 N 0  Positioning: Circle		<p>Number of positions on the circle</p>

## Section 8

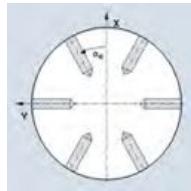
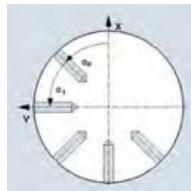
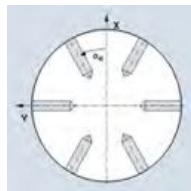
### Position patterns

Parameter	Help picture	Description and hints	Notes
<b>Positioning</b> 		<p>Only available for "Eccentric".</p> <p>Circle: The next position will be approached along the programmed circular path at the programmed feed rate FB. No contour collision will occur in a circular slot.</p> <p>Straight: The next position will be approached at rapid traverse rate. In a circular slot a contour collision will occur.</p> <p>It must be assured that the next position can be approached without danger of a collision</p>	
<b>Perimeter surface C:</b> 	 	<p>Full circle</p> <p>Partial circle</p>	
<b>X0</b> 		<p>Cylinder diameter <math>\emptyset</math> [abs]</p>	
<b>Z0</b> 		<p>Z co-ordinate of the reference point [abs]</p>	

## Section 8

### Position patterns

Notes

Parameter	Help picture	Description and hints
0	 <p><b>Pattern</b> Per.surf.C  Full circle X0 0.000 abs  Z0 0.000 abs α0 0.000 °  N 8</p>	<p>Start angle; angle of the 1<sup>st</sup> hole as referred to the X-axis.</p> <p>[Degrees]</p> <p>Positive angle: The circle is rotated anti-clockwise.</p> <p>Negative angle: The circle is rotated clockwise.</p>
1	 <p><b>Pattern</b> Per.surf.C  Pitchcircle X0 0.000 abs  Z0 0.000 abs α0 0.000 ° α1 0.000 °  N 8</p>	<p>Progression angle</p> <p>[Degrees]</p> <p>After the first hole has been machined all further positions will be positioned onwards by this angle.</p> <p>Positive angle: The circle is rotated anti-clockwise.</p> <p>Negative angle: The circle is rotated clockwise.</p> <p>Only available for "Pitch circle".</p>
N	 <p><b>Pattern</b> Per.surf.C  Full circle X0 0.000 abs  Z0 0.000 abs α0 0.000 °  N 8</p>	<p>Number of positions on the circle</p>

## Section 8

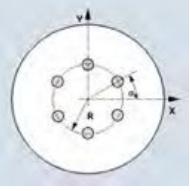
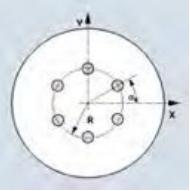
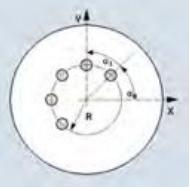
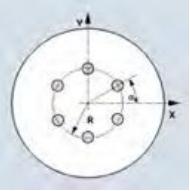
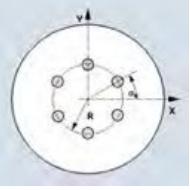
### Position patterns

Parameter	Help picture	Description and hints	Notes
Face Y:			
Concentric, eccentric	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face Y</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB 0.000 abs CP 0.000 °  XB 0.000 abs YB 0.000 abs dB 0.000 °  R 0.000 N 0</p> <p>Positioning: Circle</p>		Full circle located concentrically on the face, Full circle located eccentrically on the face
Full circle, Partial circle	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face Y</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB 0.000 abs CP 0.000 °  XB 0.000 abs YB 0.000 abs dB 0.000 °  R 0.000 N 0</p> <p>Positioning: Circle</p>		Full circle  Partial circle
ZB	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face Y</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p><b>ZB</b> 0.000 abs CP 0.000 °  XB 0.000 abs YB 0.000 abs dB 0.000 °  R 0.000 N 0</p> <p>Positioning: Circle</p>		Z co-ordinate of the reference point [abs]
CP	<p><b>Pattern</b></p> <ul style="list-style-type: none"> <li>Face Y</li> <li>Eccentric</li> <li>Full circle</li> </ul> <p>ZB 0.000 abs <b>CP</b> 0.000 °  XB 0.000 abs YB 0.000 abs dB 0.000 °  R 0.000 N 0</p> <p>Positioning: Circle</p>		Positioning angle for the machining range

## Section 8

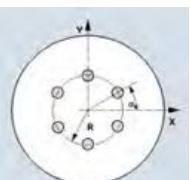
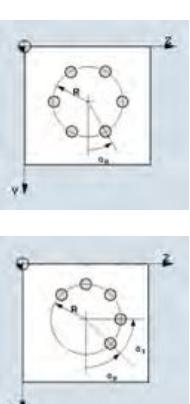
### Position patterns

Notes

Parameter	Help picture	Description and hints
X0, Y0		<p>Reference point, position of the circle centre [abs]</p> <p>Only available for "Eccentric".</p>
0		<p>Start angle; angle of the 1<sup>st</sup> hole as referred to the X-axis. [Degrees]</p> <p>Positive angle: The circle is rotated anti-clockwise.</p> <p>Negative angle: The circle is rotated clockwise.</p>
1		<p>Progression angle [Degrees]</p> <p>After the first hole has been machined all further positions will be positioned onwards by this angle.</p> <p>Positive angle: The circle is rotated anti-clockwise.</p> <p>Negative angle: The circle is rotated clockwise.</p> <p>Only available for "Pitch circle".</p>
R		<p>Radius of the circle [mm]</p>
N		<p>Number of positions on the circle</p>

## Section 8

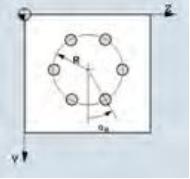
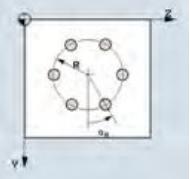
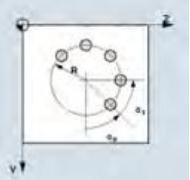
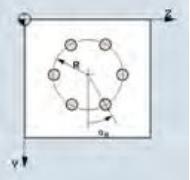
### Position patterns

Parameter	Help picture	Description and hints	Notes
<b>Positioning</b> <b>Pattern</b> Face V Eccentric Full circle ZB 0.000 abs CP 0.000 ° XB 0.000 abs YB 0.000 abs αB 0.000 ° R 0.000 N 0  Positioning: Circle		<p><b>Circle:</b> The next position will be approached along the programmed circular path at the programmed feed rate FB. No contour collision will occur in a circular slot.</p> <p><b>Straight:</b> The next position will be approached at rapid traverse rate. In a circular slot a contour collision will occur.</p> <p>It must be assured that the next position can be approached without danger of a collision occurring.</p>	
<b>Perimeter surface Y:</b> Full circle, Partial circle <b>Pattern</b> Per.surf.V Full circle XB 0.000 abs CB 0.000 ° YB 0.000 abs ZB 0.000 abs αB 0.000 ° R 0.000 N 0  Positioning: Circle		Full circle Partial circle	
X0	<b>Pattern</b> Per.surf.V Full circle XB 0.000 abs CB 0.000 ° YB 0.000 abs ZB 0.000 abs αB 0.000 ° R 0.000 N 0  Positioning: Circle	Cylinder diameter $\emptyset$ [abs]	
C0	<b>Pattern</b> Per.surf.V Full circle XB 0.000 abs CB 0.000 ° YB 0.000 abs ZB 0.000 abs αB 0.000 ° R 0.000 N 0  Positioning: Circle	Reference point	

## Section 8

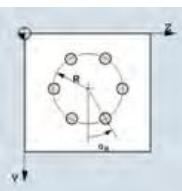
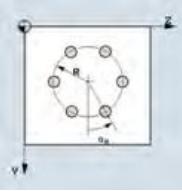
### Position patterns

Notes

Parameter	Help picture	Description and hints
Y0, Z0		Reference point, position of the circle centre [abs]
0		Start angle; angle of the 1 <sup>st</sup> hole as referred to the X-axis. [Degrees] Positive angle: The circle is rotated anti-clockwise. Negative angle: The circle is rotated clockwise.
1		Progression angle [Degrees] After the first hole has been machined all further positions will be positioned onwards by this angle. Positive angle: The circle is rotated anti-clockwise. Negative angle: The circle is rotated clockwise.
R		Radius of the circle [mm]

## Section 8

### Position patterns

Parameter	Help picture	Description and hints	Notes
N  <b>Pattern</b> Per.surf.Y  Full circle X0 0.000 abs C0 0.000 °  Y0 0.000 abs Z0 0.000 abs a0 0.000 °  R 0.000 N 0  Positioning: Circle		Number of positions on the circle	
Positioning  <b>Pattern</b> Per.surf.Y  Full circle X0 0.000 abs C0 0.000 °  Y0 0.000 abs Z0 0.000 abs a0 0.000 °  R 0.000 N 0  Positioning: Circle		<p>Circle: The next position will be approached along the programmed circular path at the programmed feed rate FB. No contour collision will occur in a circular slot.</p> <p>Straight: The next position will be approached at rapid traverse rate. In a circular slot a contour collision will occur.</p> <p>It must be assured that the next position can be approached without danger of a collision occurring.</p>	

## Section 9

### Repeat position

Notes

#### 9.1 Repeat position

If already programmed positions are to be approached again, this can be achieved simply and quickly by means of the function "Repeat position".

Repeat position

By pressing the VSK 8 "Repeat position" the function for repeating programmed position patterns will be called up.



ShopTurn attributes a number automatically for every position pattern and displays this in the work plan alongside the block number (1). The number that is to be repeated must be entered under Pos. (2).



## 1 Brief description

**Objective of the module:**

This module explains the programming of the functions of the standard turning cycles.

**Description of the module:**

In this module the functions and possibilities of ShopTurn, as referred to the standard milling cycles , will be generally described. Included in the standard milling cycles are the turning of:

- Steps (Stock removal)
- Grooves
- Undercuts (Form E, Form F, Thread DIN, Thread)
- Threads (Longitudinal, Taper, Planar)
- Cut-offs

**Content of the module:**

- Selection and general functions under "Turning"
  - Selection of the function "Turning"
  - Vertical Softkey strip
- Stock removal
  - Vertical Softkey strip
- Groove
- Undercut
  - Form E
  - Form F
  - Thread DIN
  - Thread
- Thread
  - Thread Longitudinal
  - Thread Planar
  - Synchronization point set
- Cut-off

# Operating Area Program editor - Programming Turning/Standardcycles

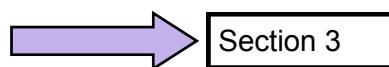
Selection and general functions  
under "Turning"

Selection of the function  
"Turning"  
Vertical Softkey strip



Stock removal

Vertical Softkey strip  
Stock removal 1  
Stock removal 2  
Stock removal 3



Groove

Groove 1  
Groove 2  
Groove 3



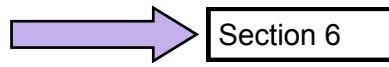
Undercut

Form E  
Form F  
Thread DIN  
Thread



Thread

Thread Longitudinal  
Thread Taper  
Thread Planar  
Synchronization point set



Cut-off



## Section 2

### Selection and general functions under "Turning"

#### 2.1 Selection of the function "Turning"

After producing the program header the program blocks for the machining, e.g. stock removal, turning of grooves, undercuts, threads or Cut-offs can be programmed.



By pressing the HSK 3 "Turning" select the range for the programming of standard milling cycles.

Notes



#### 2.2 Vertical Softkey strip



By pressing the VSK 1 "Stock removal" the respective function for the programming of Stock removal will be called up.

*See Section 3 "Stock removal".*



By pressing the VSK 2 "Groove" the respective function for the programming of Grooves will be called up.

*See Section 4 "Groove".*



By pressing the VSK 3 "Undercut" the respective function for the programming of Undercuts will be called up.

*See Section 5 "Undercut".*



By pressing the VSK 4 "Thread" the respective function for the programming of longitudinal,- taper and planar threads will be called up.

*See Section 6 „Thread“.*



By pressing the VSK 5 "Cut-off" the respective function for the programming of cut-offs will be called up.

*See Section 7 "Cut-off".*

## Section 3

Notes

Stock removal

### 3.1 Stock removal 1

With the cycle "Stock removal" any sort of work piece can be machined in longitudinal / planar direction.

Stock removal

By pressing the HSK 1 „Stock removal“ select the range for the programming of the stock removal cycle.

Stock removal 1 - straight

Stock removal 2 - straight with radii or chamfers

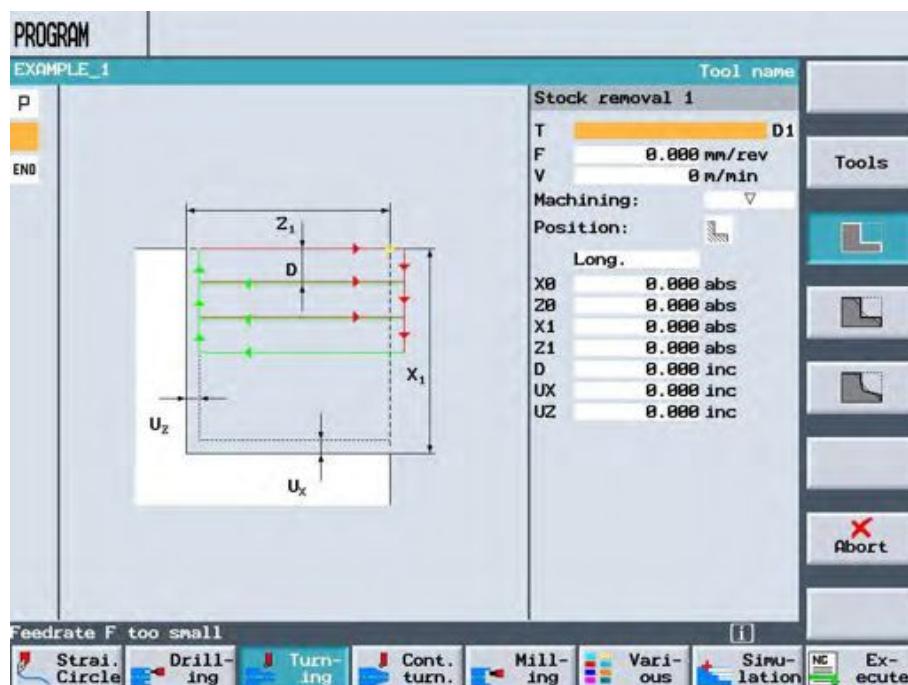
Stock removal 3 - tapers, radii or chamfers

L

By pressing the HSK 3 "Stock removal 1" select the range for the programming of the stock removal cycle.

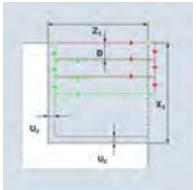
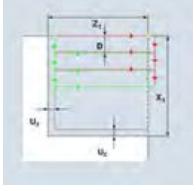
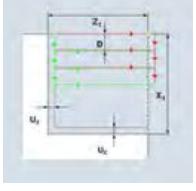
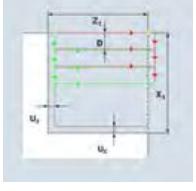
Under "Stock removal 1" the simple stock removal longitudinal or planar is possible.

The cycle "Stock removal1" is the preset cycle after selection of the Operation sector „Stock removal“.



## Section 3

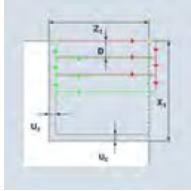
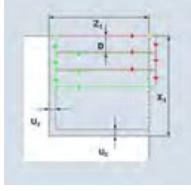
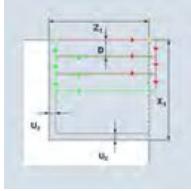
### Stock removal

Parameter	Help picture	Description and hints	Notes
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions“, Section 3 "Centring".	
Machining		Selection of the machining mode [roughing, finishing]  	
Location		Selection of the stock removal location    	
Stock removal direction,		Selection of the stock removal direction [Longitudinal, planar]	
X0, Z0,		Reference point in length and diameter [abs]	

## Section 3

### Stock removal

Notes

Parameter	Help picture	Description and hints
X1, Z1		End point in length and diameter [abs, inc]
D		Maximum Infeed [inc]
UX, UZ		Finishing allowance in X and Z [inc]

## Section 3

### Stock removal

#### 3.2 Stock removal 2

With the cycle "Stock removal 2" any sort of work piece can be turned longitudinally/planar with tapers, radii and/or chamfers.



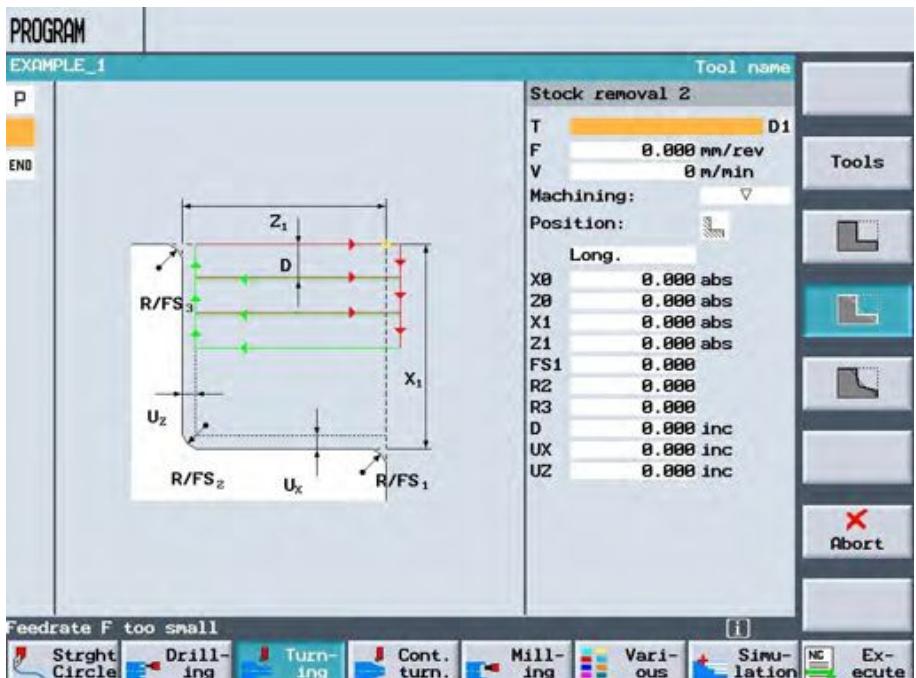
By pressing the HSK 1 "Stock removal" select the range for the programming of the stock removal cycle.



By pressing the HSK 4 "Stock removal 2" select the range for the programming of the stock removal cycle.

Under "Stock removal 2" the stock removal longitudinally or planar, straight with radii and/or chamfers is possible.

Notes



Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
Machining, Location, X0, Z0, X1, Z1		See Section „Stock removal 1“
FS1/R1 FS2/R2 FS3/R3	<p>chamfer / Radius</p> <p>[chamfer 1 / Radius 1] [chamfer 2 / Radius 2] [chamfer 3 / Radius 3]</p>	
D		See Section "Stock removal 1"
UX, UZ		See Section "Stock removal 1"

## Section 3

Notes

Stock removal

### 3.3 Stock removal 3

With the cycle "Stock removal 3" any sort of work piece can be turned longitudinally / planar with tapers.

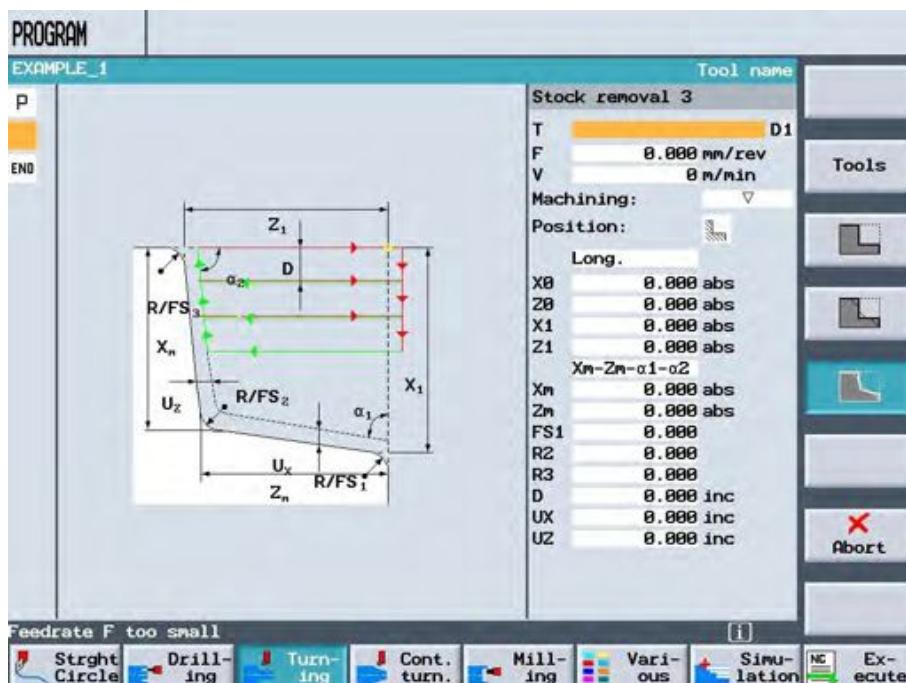
Stock removal

By pressing the HSK 1 "Stock removal" select the range for the programming of the stock removal cycle.



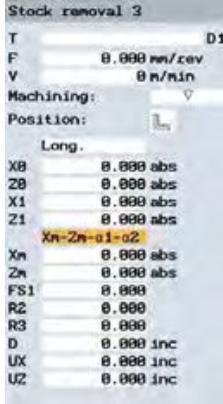
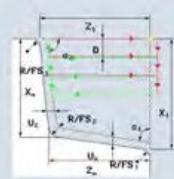
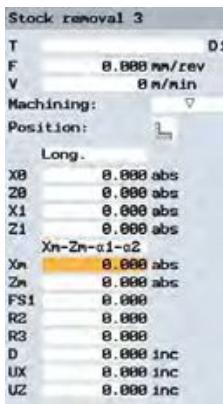
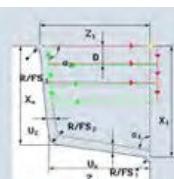
By pressing the HSK 5 "Stock removal 3" select the range for the programming of the stock removal cycle.

Under "Stock removal 3" simple stock removal longitudinally or planar with tapers incl. radii and/or chamfers is possible.



## Section 3

### Stock removal

Parameter	Help picture	Description and hints	Notes
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions“, Section 3 "Centring".	
Machining, Location, X0, Z0, X1, Z1		See Section „Stock removal 1“	
Intermediate point Xm - Zm -α1 - α2	<p><b>Stock removal 3</b></p>  <p>[Xm / Zm] [Xm / α1] [Xm / α2] [Zm / α1] [Zm / α2] [α1 / α2]</p> 	Selection	
Intermediate point	<p><b>Stock removal 3</b></p>  <p>Xm / Zm Xm / α1 Xm / α2 Zm / α1 Zm / α2 α1 / α2</p> <p>Hint: For the input in mm there is the choice of: [abs / inc].</p> 	Intermediate point selection in mm / Angle of the path.  Xm - Zm -α1 - α2  Xm / Zm Xm / α1 Xm / α2 Zm / α1 Zm / α2 α1 / α2  Hint: For the input in mm there is the choice of: [abs / inc].	
FS1, R1, FS2, R2, FS3, R3		See Section "Stock removal 2"	
D		See Section "Stock removal 1"	
UX, UZ,		See Section "Stock removal 1"	

## Section 4

Notes

### Groove

#### 4.1 Groove

With the cycle "Groove" a groove can be cut into the work piece.

**Groove**

By pressing the HSK 2 "Groove" select the range for the programming of the grooving cycle.

Groove 1 - simple grooving cycle

Groove 2 - grooving cycle with tapers, radii or chamfers

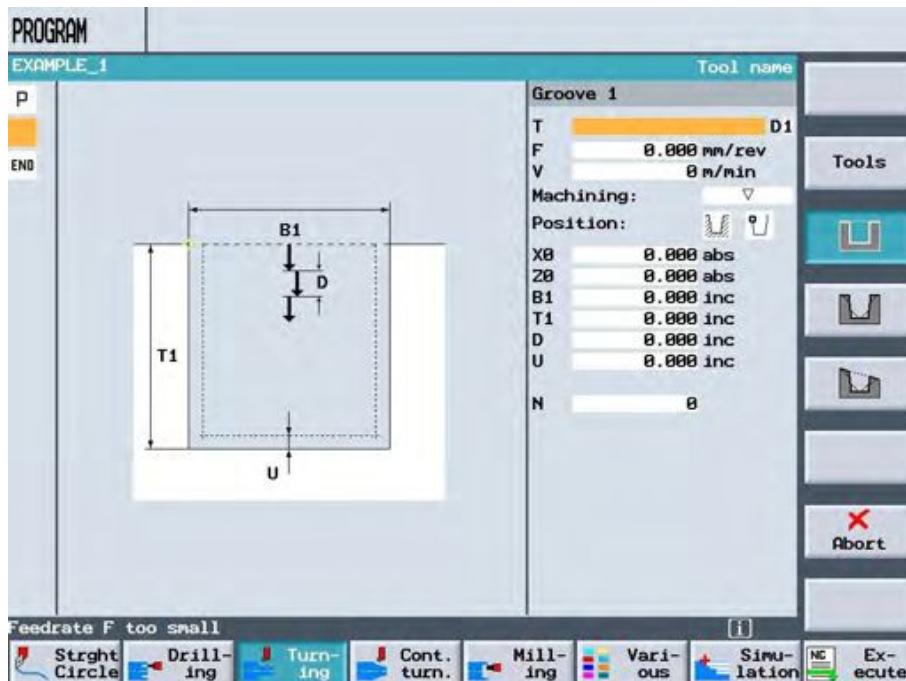
Groove 3 - grooving cycle on a taper with tapers, radii or chamfers



By pressing the HSK 3 "Groove" select the range for the programming of the grooving cycle.

Under "Groove" simple grooving can be carried out.

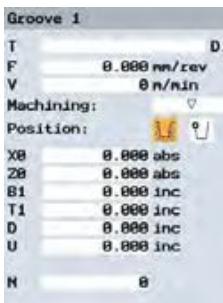
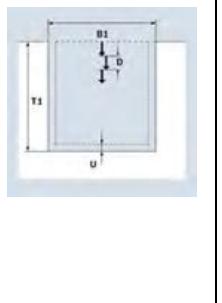
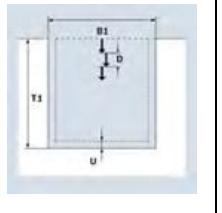
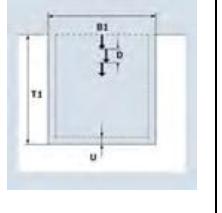
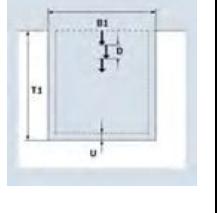
Cycle "Groove 1" is the default cycle after selection of the Operation sector Groove



Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
Machining		Selection of the machining mode [roughing, finishing, roughing + finishing]

## Section 4

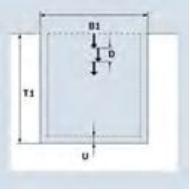
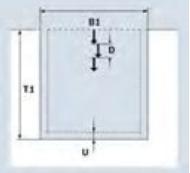
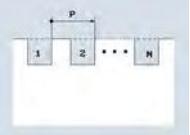
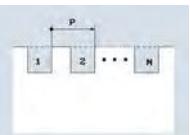
### Groove

Parameter	Help picture	Description and hints	Notes
Location		Selection of the groove location	
Location		Lage des Reference pointes	
X0, Z0		Reference point [abs / inc].	
B1		Width of groove [inc]	
T1		Depth of groove at the reference point [abs / inc]	

## Section 4

### Groove

Notes

Parameter	Help picture	Description and hints
D		Maximum Infeed [inc]
U, UX, UZ		Finishing allowance Alternatively: UX, UZ [inc]
N		Number of grooves
P		Distance between grooves Available only for N > 1

## Section 4

### Groove

#### 4.2 Groove 2

With the cycle "Groove 2" a groove with tapers, radii and/or chamfers can be machined into the work piece.

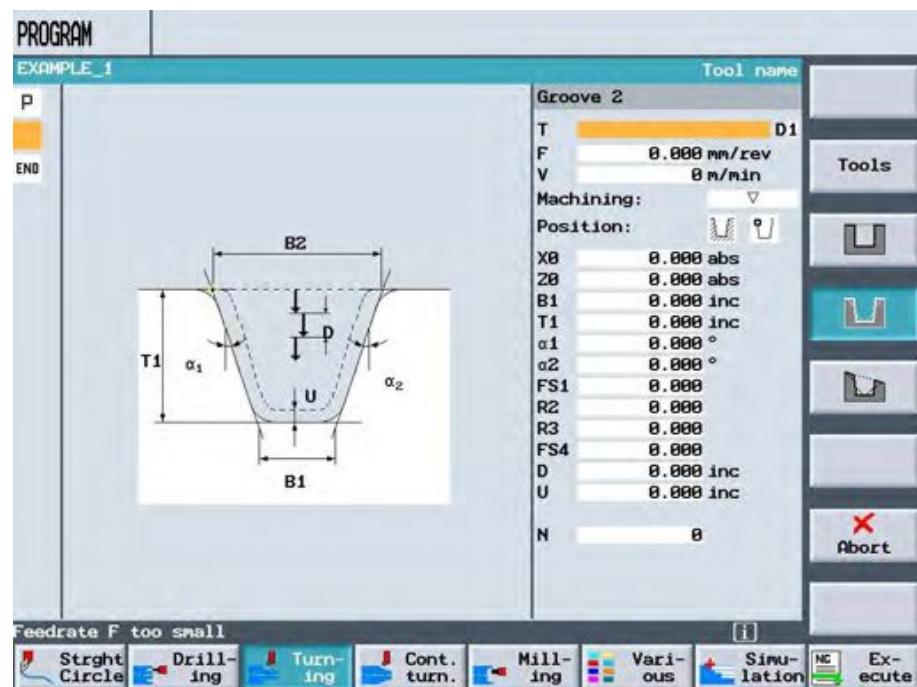
Groove

By pressing the HSK 2 "Groove" select the range for the programming of the grooving cycle.



By pressing the HSK 3 "Groove 2" select the range for the programming of the grooving cycle.

Under "Groove 2" the grooving with tapers incl. radii and/or chamfers is possible.

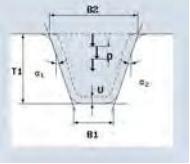
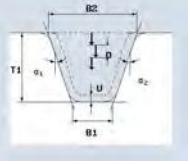
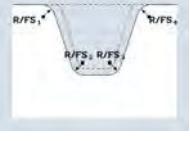


Notes

## Section 4

### Groove

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
Machining, Location, X0, Z0		See Section "Groove 1"
B1 / B2		Width of groove [inc] [B1, B2]
T1		See Section "Groove 1"
alpha1, alpha2		Flank angle [Degrees]
FS1, R1, FS2, R2, FS3, R3, FS4, R4		See Section "Groove 1"
D, U, N		See Section "Groove 1"

## Section 4

### Groove

#### 4.3 Groove 3

With the cycle "Groove 3" a groove can be cut on a taper with tapers, radii and/or chamfers.

Groove

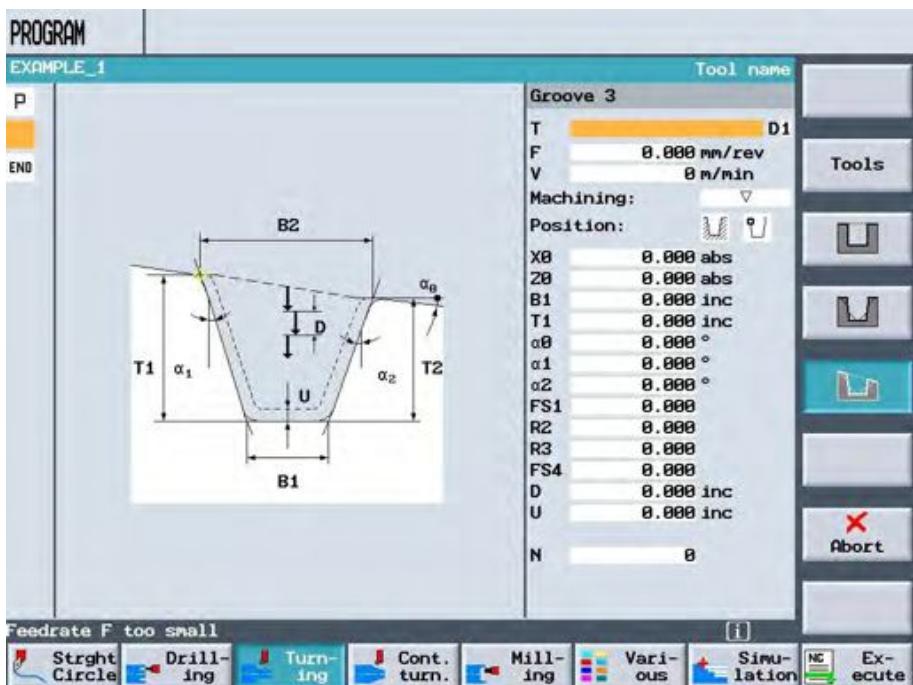
By pressing the HSK 2 "Groove" select the range for the programming of the grooving cycle.



By pressing the HSK 4 "Groove 3" select the range for the programming of the grooving cycle.

Under "Groove 3" the grooving on a taper with tapers, radii and/or chamfers is possible.

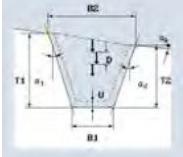
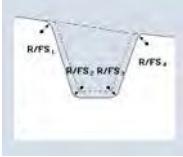
Notes



## Section 4

Notes

### Groove

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions“, Section 3 "Centring".
Machining, Location, X0, Z0, B1, T1		See Section "Groove 2"
a0	 <p>The diagram shows a cross-section of a tapered groove. It features two parallel horizontal lines at the top and bottom labeled B2 and B1 respectively. Between them are two vertical lines labeled T1 and T2. The angle between the top and bottom lines is labeled a0. A point labeled U is shown on the left side of the groove.</p>	Angle of the taper [Degrees]
FS1, R1, FS2, R2, FS3, R3, FS4, R4	 <p>The diagram shows a cross-section of a groove with four radii labeled R1, R2, R3, and R4. Below the groove, four finish codes are listed: R/FS1, R/FS2, R/FS3, and R/FS4.</p>	See Section "Groove 2"
D, U, N		See Section "Groove 2"

## Section 5

### Undercut

#### 5.1 Undercut Form E

With the cycle "Undercut Form E" any sort of undercut can be machined on the work piece.

Notes

Undercut

By pressing the HSK 3 "Undercut" select the range for the programming of Undercut cycles.

Undercut Form E as per DIN509

Undercut Form F as per DIN 509

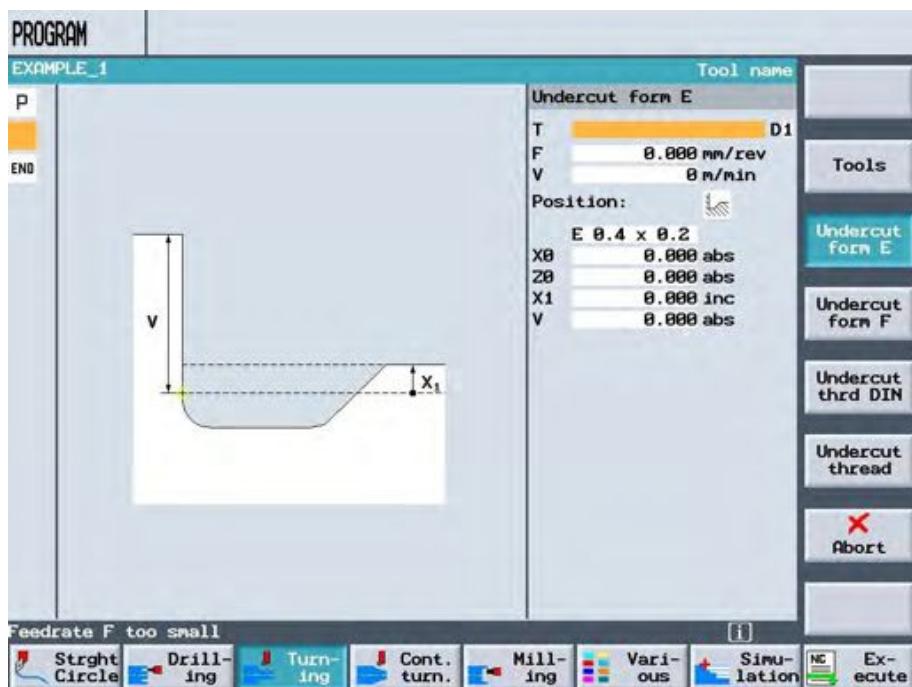
Undercut Thread DIN as per DIN76

Undercut Thread - freely definable undercut.

Undercut  
form E

By pressing the HSK 3 "Undercut Form E" select the range for the programming of the Undercut cycle Form E as per DIN509.

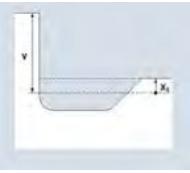
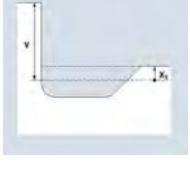
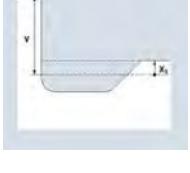
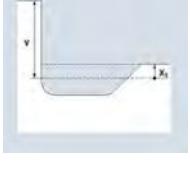
Cycle "Undercut Form E" is the default cycle after selection of the Operation sector Undercut.



## Section 5

### Undercut

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
Location		Selection of the stock removal location
Undercut size		Selection of the Undercut size <b>E 0.4 x 0.2</b> <b>E 0.6 x 0.2</b> <b>E 0.6 x 0.3</b> <b>E 1.0 x 0.4</b> <b>E 1.0 x 0.2</b> <b>E 1.6 x 0.3</b> <b>E 2.5 x 0.4</b> <b>E 4.0 x 0.5</b>
X0, Z0		Reference point
X1		Allowance in X [abs, inc]
V1		Parallel path [abs, inc]

## Section 5

### Undercut

#### 5.2 Undercut Form F

With the cycle "Undercut Form F" an Undercut can be machined on the work piece.

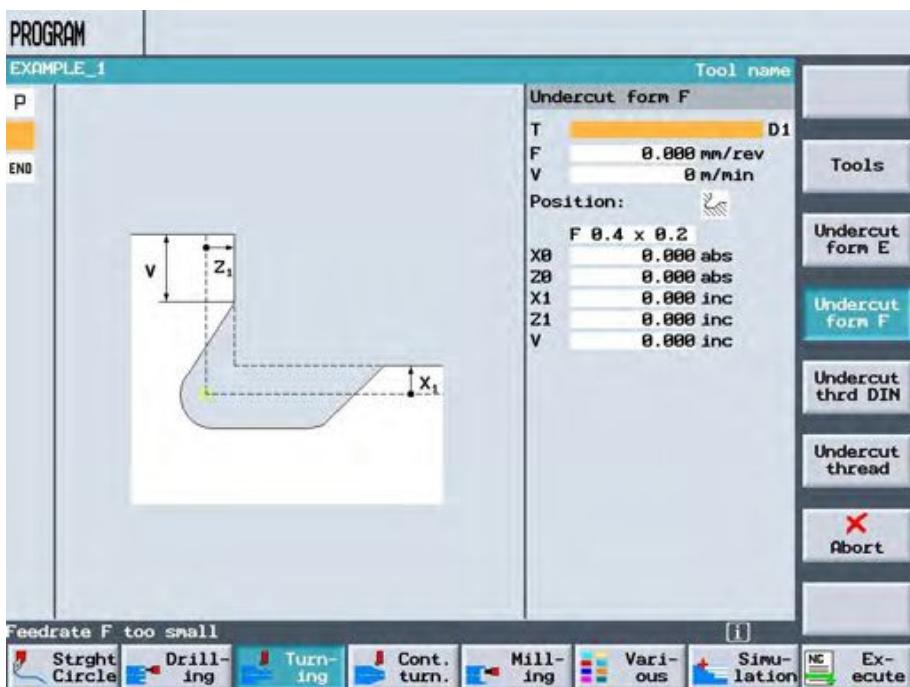
Undercut 5

By pressing the HSK 3 "Undercut" select the range for the programming of the Undercut cycles.

Undercut form F

By pressing the HSK 3 "Undercut Form F" select the range for the programming of the Undercut cycle.

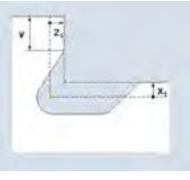
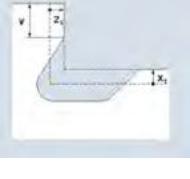
Notes



## Section 5

### Undercut

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
Location		Selection of the stock removal location
Undercut size		Selection of the Undercut size <b>F 0.4 x 0.2</b> <b>F 0.6 x 0.2</b> <b>F 0.6 x 0.3</b> <b>F 1.0 x 0.2</b> <b>F 1.0 x 0.4</b> <b>F 1.6 x 0.3</b> <b>F 2.5 x 0.4</b> <b>F 4.0 x 0.5</b>
X0, Z0		Reference point See Section "Undercut Form E"
X1, Z1		Allowance [abs, inc]
V		Parallel path See Section "Undercut Form E"

## Section 5

### Undercut Thread DIN

#### 5.3 Undercut Thread DIN

With the cycle "Undercut Thread DIN" a Thread-undercut as per DIN can be machined on the work piece.

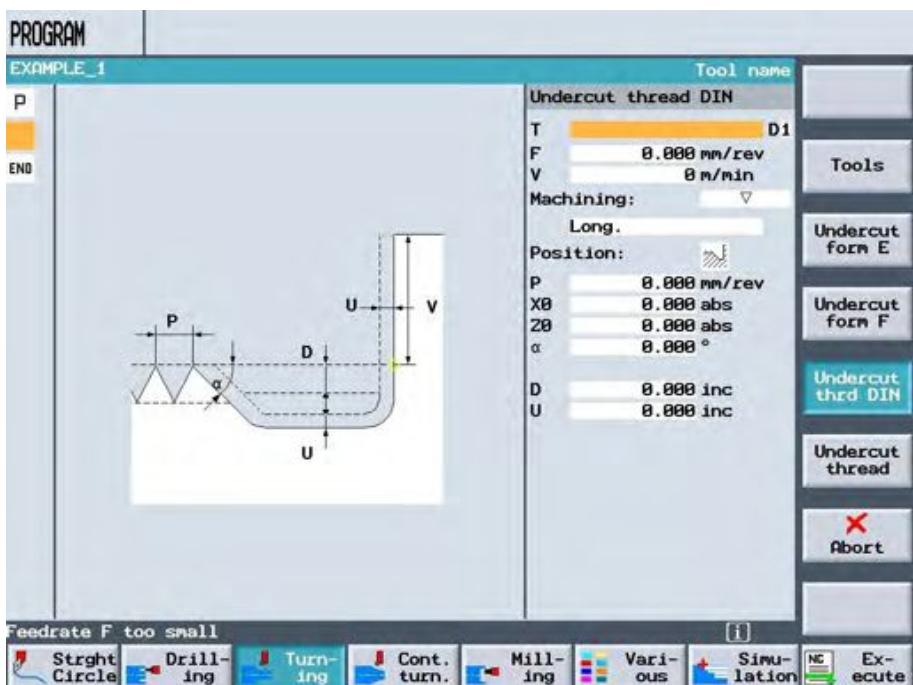
Undercut

By pressing the HSK 3 "Undercut" select the range for the programming of the Undercut cycles.

Undercut  
thrd DIN

By pressing the HSK 5 "Undercut thrd DIN" select the range for the programming of the Undercut cycle as per DIN 76.

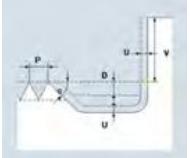
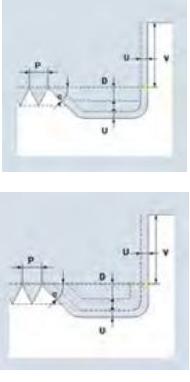
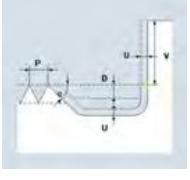
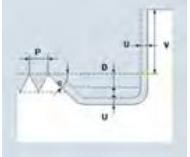
Notes



## Section 5

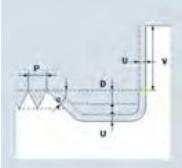
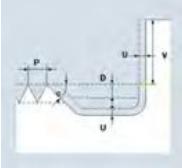
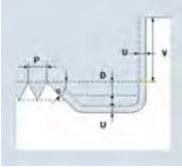
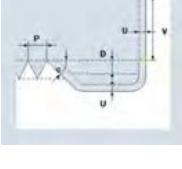
### Undercut Thread DIN

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
Machining		Selection of the machining mode [roughing, finishing, roughing + finishing]
Machining direction		Selection of the machining direction [Longitudinal]  [Parallel to the contour]
Location		Selection of the stock removal location
P		<p>Pitch [mm/rev]</p> <p>The pitch can be freely programmed.</p> <p>Alternatively:</p> <p>[0,25 mm/rev, - 0,30 mm/rev, - 0,35 mm/rev]  [0,40 mm/rev, - 0,45 mm/rev, - 0,50 mm/rev]  [0,60 mm/rev, - 0,70 mm/rev, - 0,75 mm/rev]  [0,80 mm/rev, - 1,00 mm/rev, - 1,25 mm/rev]  [1,50 mm/rev, - 1,75 mm/rev, - 2,00 mm/rev]  [2,50 mm/rev, - 3,00 mm/rev, - 3,50 mm/rev]  [4,00 mm/rev, - 4,50 mm/rev, - 5,00 mm/rev]  [5,50 mm/rev, - 6,00 mm/rev]</p>

## Section 5

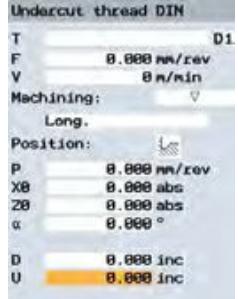
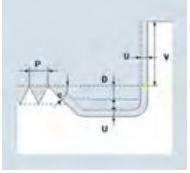
### Undercut Thread DIN

Parameter	Help picture	Description and hints	Notes
X0, Z0		Reference point [abs]	
$\alpha$		Plunging angle [Degrees]	
V		Parallel path [abs, inc]  Hint:  Available only for machining "Finishing" or "Roughing + Finishing"	
D		Infeed [inc]	

## Section 5

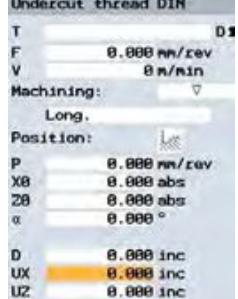
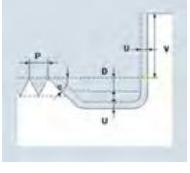
### Undercut Thread DIN

Notes

Parameter	Help picture	Description and hints
<b>U</b> 		Finishing allowance [inc]



By pressing the HSK 1 "Alternatively" the parameters change as follows:

Parameter	Help picture	Description and hints
<b>UX, UZ</b> 		Finishing allowance in X / in Z' [inc]

## Section 5

### Undercut Thread

#### 5.4 Undercut Thread

With the cycle "Undercut Thread" a freely definable Thread undercut can be machined on the work piece.

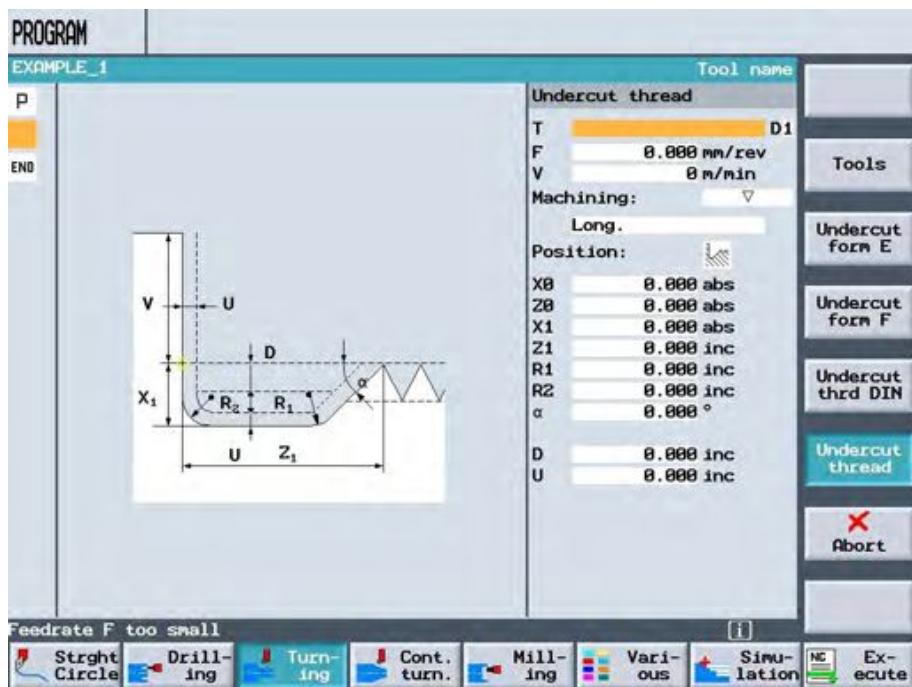
Notes

Undercut

By pressing the HSK 3 "Undercut" select the range for the programming of the Undercut cycles.

Undercut  
thread

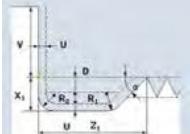
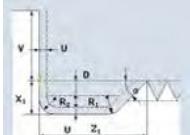
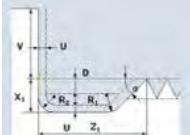
By pressing the HSK 6 "Undercut Thread" select the range for the programming of the freely definable Undercut cycle.



## Section 5

### Undercut Thread

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 „Centring“.
Machining		See Section „Undercut Thread DIN“
Machining direction	 <b>Undercut thread</b> D1 T 0.000 mm/rev F 0.000 abs V 0 m/min Machining: Long. Position: X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 inc R1 0.000 inc R2 0.000 inc α 0.000 ° D 0.000 inc U 0.000 inc	Selection of the machining direction [Longitudinal]  [Parallel to the contour]
Machining, Location, X0, Z0		See Section "Undercut Thread DIN"
X1, Z1	 <b>Undercut thread</b> D1 T 0.000 mm/rev F 0.000 abs V 0 m/min Machining: Long. Position: X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 inc R1 0.000 inc R2 0.000 inc α 0.000 ° D 0.000 inc U 0.000 inc	Allowance in X / in Z [abs, inc]
R1, R2	 <b>Undercut thread</b> D1 T 0.000 mm/rev F 0.000 abs V 0 m/min Machining: Long. Position: X0 0.000 abs Z0 0.000 abs X1 0.000 abs Z1 0.000 inc R1 0.000 inc R2 0.000 inc α 0.000 ° D 0.000 inc U 0.000 inc	Radius 1 / Radius 2 [inc]
α, V, D, U, UX, UZ		See Section "Undercut Thread DIN"

## Section 6

### Thread

#### 6.1 Thread Longitudinal

Notes

With the cycle "Thread" a thread can be cut on the work piece.

Thread

By pressing the HSK 4 "Thread" select the range for the programming of the thread cutting cycles.

Thread Longitudinal

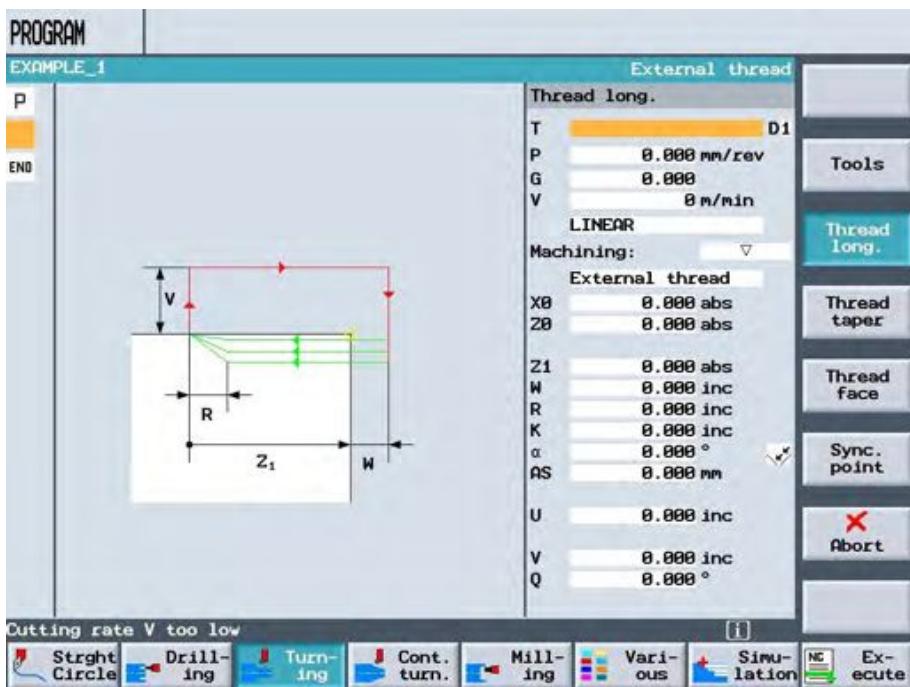
Thread Taper

Thread Planar

Set synchronization point - (Thread cutting)

Thread long.

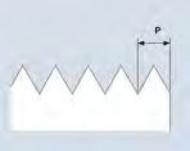
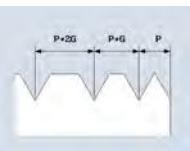
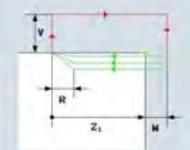
By pressing the HSK 3 "Thread long." select the range for the programming of the longitudinal thread cycle.



## Section 6

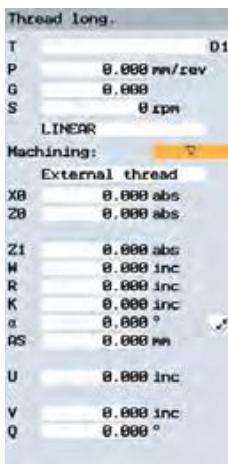
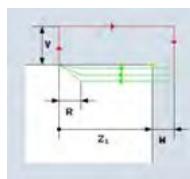
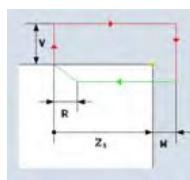
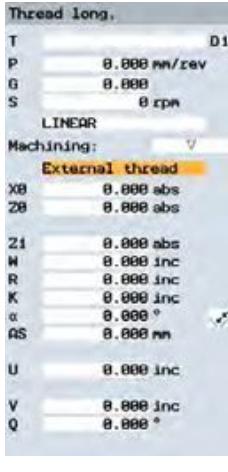
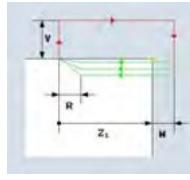
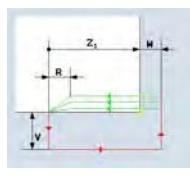
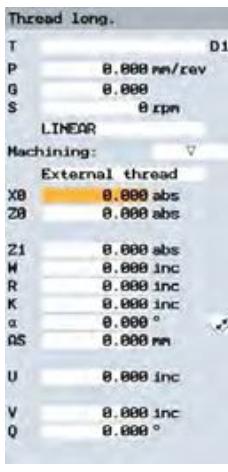
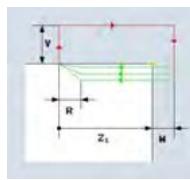
### Thread

Notes

Parameter	Help picture	Description and hints
T		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
P	 <b>Thread long.</b> T D1 P 0.000 mm/rev G 0.000 V 0 m/min LINEAR Machining: External thread X8 0.000 abs Z8 0.000 abs  Z1 0.000 abs W 0.000 inc R 0.000 inc K 0.000 inc α 0.000 ° AS 0.000 mm  U 0.000 inc  V 0.000 inc Q 0.000 °	Selection of the pitch Alternatively: 0.000 mm/rev 0.000 in/rev 0 / 100 Thrcds/" 0.000 MODULE
G	 <b>Thread long.</b> T D1 P 0.000 mm/rev G 0.000 V 0 m/min LINEAR Machining: External thread X8 0.000 abs Z8 0.000 abs  Z1 0.000 abs W 0.000 inc R 0.000 inc K 0.000 inc α 0.000 ° AS 0.000 mm  U 0.000 inc  V 0.000 inc Q 0.000 °	Pitch change Parameter "G" Available only for mm/rev and in/rev.
V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3  Centring .
Infeed	 <b>Thread long.</b> T D1 P 0.000 mm/rev G 0.000 V 0 m/min LINEAR Machining: External thread X8 0.000 abs Z8 0.000 abs  Z1 0.000 abs W 0.000 inc R 0.000 inc K 0.000 inc α 0.000 ° AS 0.000 mm  U 0.000 inc  V 0.000 inc Q 0.000 °	Infeed: Constant cutting depth / Constant chip cross section Alternatively: LINEAR DEGRESSIVE

## Section 6

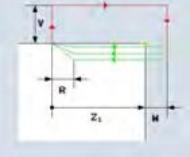
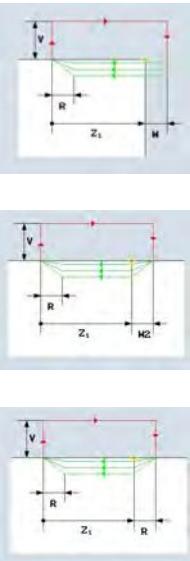
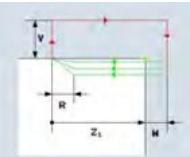
### Thread

Parameter	Help picture	Description and hints	Notes
Machining   <p><b>Thread long:</b> T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: ▽ External thread X0 0.000 abs Z0 0.000  Z1 0.000 abs H 0.000 inc R 0.000 inc K 0.000 inc α 0.000 ° AS 0.000 mm  U 0.000 inc V 0.000 inc Q 0.000 °</p>	  	Selection of the machining mode [roughing, finishing, roughing + finishing]  ▽ ▽▽▽ ▽+▽▽▽	
Machining direction   <p><b>Thread long:</b> T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: ▽ External thread X0 0.000 abs Z0 0.000  Z1 0.000 abs H 0.000 inc R 0.000 inc K 0.000 inc α 0.000 ° AS 0.000 mm  U 0.000 inc V 0.000 inc Q 0.000 °</p>	  	Selection of the machining direction Alternatively:  <b>External thread</b>  <b>Internal thread</b>	
X0, Z0   <p><b>Thread long:</b> T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: ▽ External thread X0 0.000 abs Z0 0.000  Z1 0.000 abs H 0.000 inc R 0.000 inc K 0.000 inc α 0.000 ° AS 0.000 mm  U 0.000 inc V 0.000 inc Q 0.000 °</p>		Reference point [abs]	

## Section 6

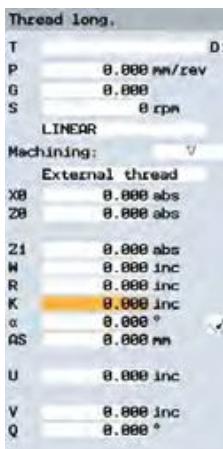
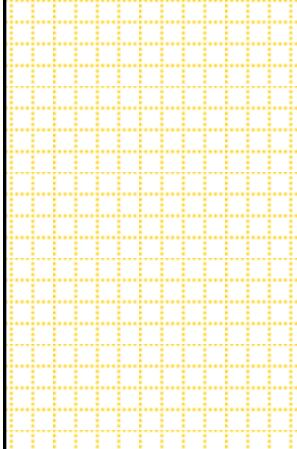
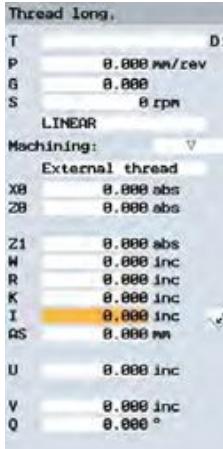
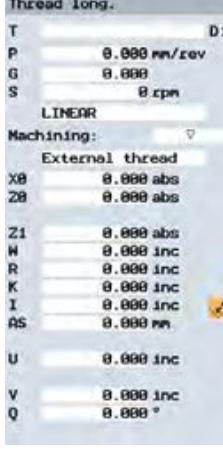
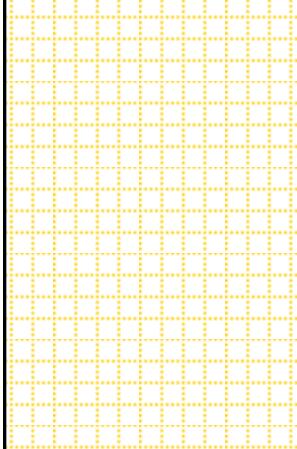
### Thread

Notes

Parameter	Help picture	Description and hints
Z1		Thread length [abs, inc]
W, W2, W2 =R		Thread run-in / Thread entry / Thread entry = Thread exit  W: Thread run-in  Alternatively: [W, W2, W2=R]  W2: Thread entry  W2=R: Thread entry = Thread exit
R		Thread exit [inc]

## Section 6

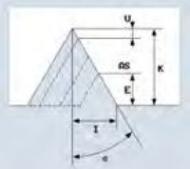
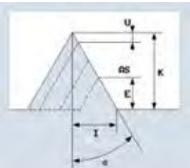
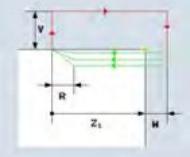
### Thread

Parameter	Help picture	Description and hints	Notes
K		<p>Thread depth</p> <p>[inc]</p> <p>Hint: By input of the thread pitch "P" the value will be evaluated by ShopTurn. If the value has been evaluated by ShopTurn, the field appears with a grey background. . The value can, however, be altered in spite and the field appears white once again.</p>	
I		<p>Infeed slope (flank) / (angle)</p> <p>Alternatively: [Degrees, inc]</p>	
Infeed		<p>Selection of the infeed mode</p>  <p>[Infeed along a flank, - Infeed with changing flank]</p>	

## Section 6

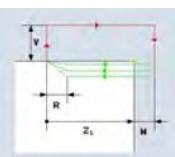
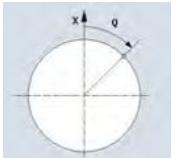
### Thread

Notes

Parameter	Help picture	Description and hints
AS  Thread long. T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: External thread X0 0.000 abs Z0 0.000 abs Z1 0.000 abs H 0.000 inc R 0.000 inc K 0.000 inc I 0.000 inc RS 0.000 mm U 0.000 inc V 0.000 inc Q 0.000 °		First infeed / number of roughing cuts [mm]
U  Thread long. T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: External thread X0 0.000 abs Z0 0.000 abs Z1 0.000 abs H 0.000 inc R 0.000 inc K 0.000 inc I 0.000 inc RS 0.000 mm U 0.000 inc V 0.000 inc Q 0.000 °		Finishing allowance [inc]
NN  Thread long. T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: VVV External thread X0 0.000 abs Z0 0.000 abs Z1 0.000 abs H 0.000 inc R 0.000 inc K 0.000 inc I 0.000 inc RS 0.000 mm U 0.000 inc NN 0 V 0.000 inc Q 0.000 °		Dummy passes

## Section 6

### Thread

Parameter	Help picture	Description and hints	Notes																																		
V	<p>Thread long:</p> <table border="1"> <tr><td>T</td><td>D1</td></tr> <tr><td>P</td><td>0.000 mm/rev</td></tr> <tr><td>G</td><td>0.000</td></tr> <tr><td>S</td><td>0 rpm</td></tr> <tr><td colspan="2">LINEAR</td></tr> <tr><td>Machining:</td><td>External thread</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>H</td><td>0.000 inc</td></tr> <tr><td>R</td><td>0.000 inc</td></tr> <tr><td>K</td><td>0.000 inc</td></tr> <tr><td>I</td><td>0.000 inc</td></tr> <tr><td>AS</td><td>0.000 mm</td></tr> <tr><td>U</td><td>0.000 inc</td></tr> <tr><td>V</td><td>0.000 inc</td></tr> <tr><td>Q</td><td>0.000 °</td></tr> </table> 	T	D1	P	0.000 mm/rev	G	0.000	S	0 rpm	LINEAR		Machining:	External thread	X0	0.000 abs	Z0	0.000 abs	Z1	0.000 abs	H	0.000 inc	R	0.000 inc	K	0.000 inc	I	0.000 inc	AS	0.000 mm	U	0.000 inc	V	0.000 inc	Q	0.000 °	Return run lift-off [inc]	
T	D1																																				
P	0.000 mm/rev																																				
G	0.000																																				
S	0 rpm																																				
LINEAR																																					
Machining:	External thread																																				
X0	0.000 abs																																				
Z0	0.000 abs																																				
Z1	0.000 abs																																				
H	0.000 inc																																				
R	0.000 inc																																				
K	0.000 inc																																				
I	0.000 inc																																				
AS	0.000 mm																																				
U	0.000 inc																																				
V	0.000 inc																																				
Q	0.000 °																																				
Q	<p>Thread long:</p> <table border="1"> <tr><td>T</td><td>D1</td></tr> <tr><td>P</td><td>0.000 mm/rev</td></tr> <tr><td>G</td><td>0.000</td></tr> <tr><td>S</td><td>0 rpm</td></tr> <tr><td colspan="2">LINEAR</td></tr> <tr><td>Machining:</td><td>External thread</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>Z1</td><td>0.000 abs</td></tr> <tr><td>H</td><td>0.000 inc</td></tr> <tr><td>R</td><td>0.000 inc</td></tr> <tr><td>K</td><td>0.000 inc</td></tr> <tr><td>I</td><td>0.000 inc</td></tr> <tr><td>AS</td><td>0.000 mm</td></tr> <tr><td>U</td><td>0.000 inc</td></tr> <tr><td>V</td><td>0.000 inc</td></tr> <tr><td>Q</td><td>0.000 °</td></tr> </table> 	T	D1	P	0.000 mm/rev	G	0.000	S	0 rpm	LINEAR		Machining:	External thread	X0	0.000 abs	Z0	0.000 abs	Z1	0.000 abs	H	0.000 inc	R	0.000 inc	K	0.000 inc	I	0.000 inc	AS	0.000 mm	U	0.000 inc	V	0.000 inc	Q	0.000 °	Start angle offset [Degrees]	
T	D1																																				
P	0.000 mm/rev																																				
G	0.000																																				
S	0 rpm																																				
LINEAR																																					
Machining:	External thread																																				
X0	0.000 abs																																				
Z0	0.000 abs																																				
Z1	0.000 abs																																				
H	0.000 inc																																				
R	0.000 inc																																				
K	0.000 inc																																				
I	0.000 inc																																				
AS	0.000 mm																																				
U	0.000 inc																																				
V	0.000 inc																																				
Q	0.000 °																																				

## Section 5

### Undercut

Notes

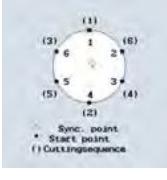


By pressing the VSK 1 "Alternatively" the input parameters for a multi-start thread is called up.

Parameter	Help picture	Description and hints
L	 	Number of starts (max. 6)
N	 	1 of L starts Hint: N ≠ 0: "machine only start N' parameter "P" is not shown N = 0: "machine all starts' parameter "P" is shown
A	 	Start changing depth [inc] Hint: Firstly machine all thread-starts one after the other to the start changing depth A , then machine all thread-starts one after the other to the depth 2A etc. until the final depth is reached. A=0: Start changing depth will not be taken into account, i.e. each thread start is machined completely before the thread start is machined.

## Section 6

### Thread

Parameter	Help picture	Description and hints	Notes
P  <b>Thread long.</b> T D1 P 0.000 mm/rev G 0.000 S 0 rpm LINEAR Machining: External thread X0 0.000 abs Z0 0.000 abs Z1 0.000 abs W 0.000 inc R 0.000 inc K 0.000 inc I 0.000 inc AS 0.000 mm U 0.000 inc V 0.000 inc L G N 0 A 0.000 inc P 1		<p>Start f thread-start P=1.....L</p> <p>Hint: Available only for N=0 If P &gt; 1 the previously machined starts are not taken into account.</p>	

## Section 6

Notes

Thread

### 6.2 Thread taper

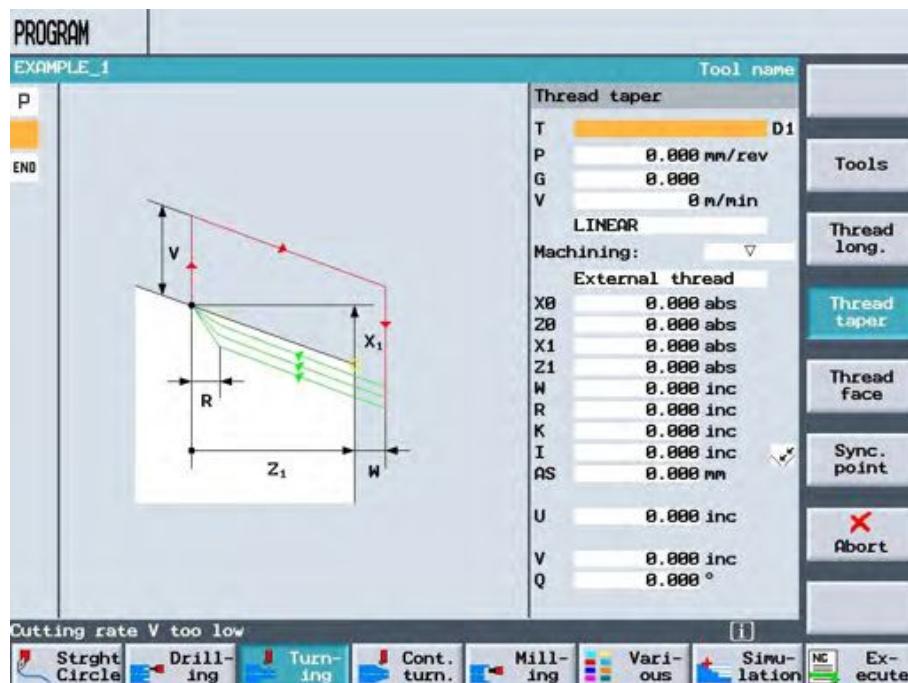
With the cycle "Thread taper" a taper thread can be cut on the work piece.

Thread

By pressing the HSK 4 "Thread" select the range for the programming of the thread cutting cycles.

Thread  
taper

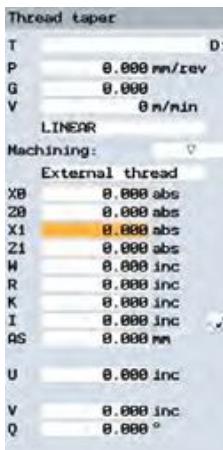
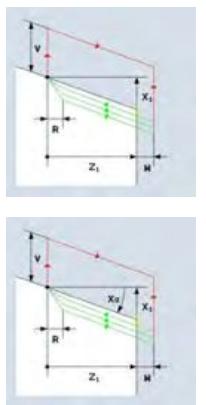
By pressing the HSK 4 "Thread taper" select the range for the programming of the cycle "Taper threads".



## Section 6

### Thread

Notes

Parameter	Help picture	Description and hints
T		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
P,G,V; Machining, X0, Z0		See Section "Thread Longitudinal".
X1	 	Thread taper [abs, inc, Degrees]
Z1, W (W2, W2=R), R, K, I, AS, U, NN, V, Q;(L, N, A, P)		See Section "Thread longitudinal".

## Section 6

Notes

Thread

### 6.3 Thread planar

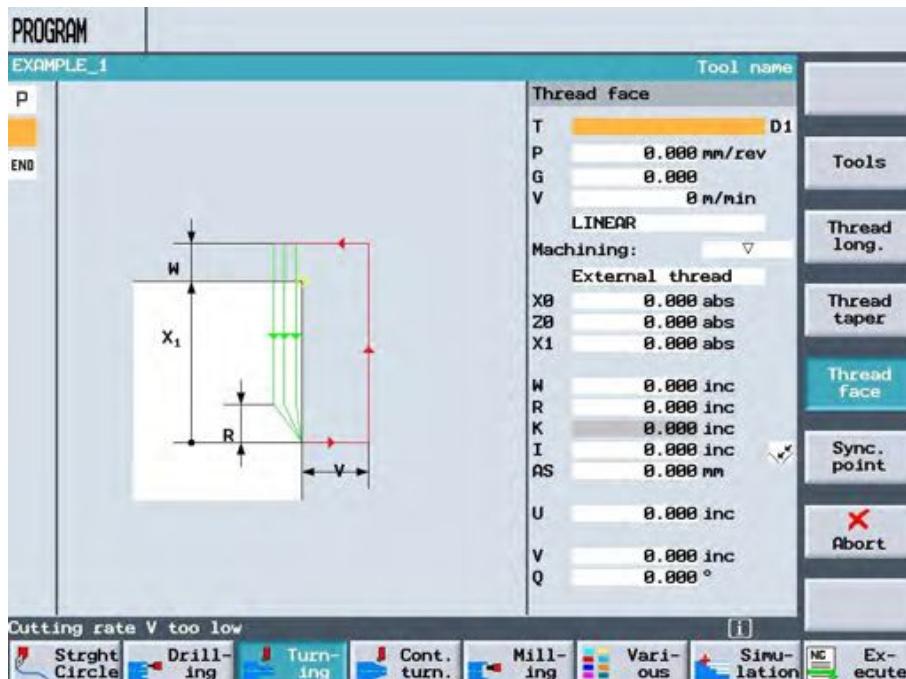
With the cycle "Thread Face" a thread can be cut on the end face of the work piece.

Thread

By pressing the HSK 4 "Thread" select the range for the programming of the thread cutting cycles.

Thread  
face

By pressing the HSK 5 "Thread face" select the range for the programming of the cycle „Planar thread“.



Parameter	Help picture	Description and hints
T		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".
P,G,V; Machining, X0, Z0, X1		See Section "Thread longitudinal".
W (W2, W2=R) R, K, I, AS, U, NN, V, Q;(L, N, A, P)		See Section "Thread longitudinal".

## Section 6

### Thread

#### 6.4 Set synchronization point

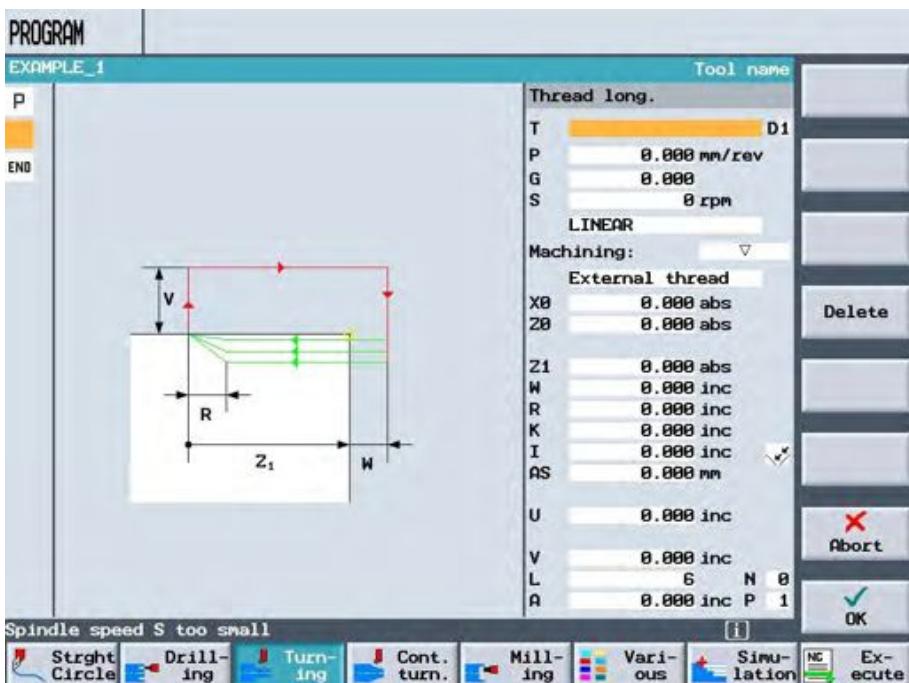
If a thread is to be re-machined - e.g. if during thread cutting the tool broke - the function "Synchronization point" can be used'. ShopTurn in this case takes into account the angular offset of a thread start that results by the renewed clamping of the work piece.

Thread

By pressing the HSK 4 "Thread" select the range for the programming of the thread cutting cycles.

Sync.  
point

By pressing the HSK 6 "Synchronization point" determine the synchronization point in the previously selected thread cutting cycle ["Thread longitudinal", "Thread taper", "Thread planar"].



Hint:

The function Synchronization point is suitable for the:

- re-machining of threads
- start point determination in case of multi-start threads
- re-approach of a thread start

Notes

## Section 7

Notes

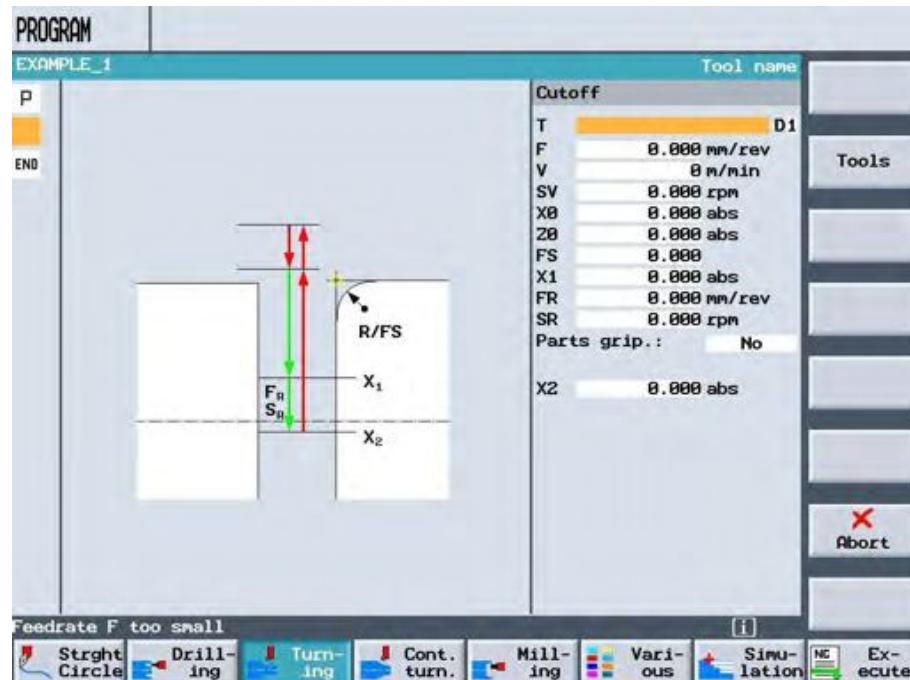
Cut-off

### 7.1 Cut-off

With the cycle "Cut-off" any sort of work piece can be parted off

Cutoff

By pressing the HSK 5 "Cut-off" select the range for the programming of the Cut-off cycle.



## Section 7

### Cut-off

Parameter	Help picture	Description and hints	Notes																								
T,F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Thread, Positions", Section 3 "Centring".																									
SV	<p><b>Cutoff</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/rev</td></tr> <tr><td>F</td><td>0 m/min</td></tr> <tr><td>V</td><td>0.000 rpm</td></tr> <tr><td>SV</td><td>8.000 rpm</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>FS</td><td>0.000</td></tr> <tr><td>X1</td><td>0.000 abs</td></tr> <tr><td>FR</td><td>0.000 mm/rev</td></tr> <tr><td>SR</td><td>0.000 rpm</td></tr> <tr><td>Parts grip.:</td><td>No</td></tr> <tr><td>X2</td><td>0.000 abs</td></tr> </table>	T	0.000 mm/rev	F	0 m/min	V	0.000 rpm	SV	8.000 rpm	X0	0.000 abs	Z0	0.000 abs	FS	0.000	X1	0.000 abs	FR	0.000 mm/rev	SR	0.000 rpm	Parts grip.:	No	X2	0.000 abs	Speed limit for constant cutting speed. [rev/min]  If only the parameter "V" appears - Cutting speed.	
T	0.000 mm/rev																										
F	0 m/min																										
V	0.000 rpm																										
SV	8.000 rpm																										
X0	0.000 abs																										
Z0	0.000 abs																										
FS	0.000																										
X1	0.000 abs																										
FR	0.000 mm/rev																										
SR	0.000 rpm																										
Parts grip.:	No																										
X2	0.000 abs																										
X0, Z0	<p><b>Cutoff</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/rev</td></tr> <tr><td>F</td><td>0 m/min</td></tr> <tr><td>V</td><td>0.000 rpm</td></tr> <tr><td>SV</td><td>8.000 rpm</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>FS</td><td>0.000</td></tr> <tr><td>X1</td><td>0.000 abs</td></tr> <tr><td>FR</td><td>0.000 mm/rev</td></tr> <tr><td>SR</td><td>0.000 rpm</td></tr> <tr><td>Parts grip.:</td><td>No</td></tr> <tr><td>X2</td><td>0.000 abs</td></tr> </table>	T	0.000 mm/rev	F	0 m/min	V	0.000 rpm	SV	8.000 rpm	X0	0.000 abs	Z0	0.000 abs	FS	0.000	X1	0.000 abs	FR	0.000 mm/rev	SR	0.000 rpm	Parts grip.:	No	X2	0.000 abs	Reference point [abs, inc]	
T	0.000 mm/rev																										
F	0 m/min																										
V	0.000 rpm																										
SV	8.000 rpm																										
X0	0.000 abs																										
Z0	0.000 abs																										
FS	0.000																										
X1	0.000 abs																										
FR	0.000 mm/rev																										
SR	0.000 rpm																										
Parts grip.:	No																										
X2	0.000 abs																										
FS / R	<p><b>Cutoff</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/rev</td></tr> <tr><td>F</td><td>0 m/min</td></tr> <tr><td>V</td><td>0.000 rpm</td></tr> <tr><td>SV</td><td>8.000 rpm</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>FS</td><td>8.000</td></tr> <tr><td>X1</td><td>0.000 abs</td></tr> <tr><td>FR</td><td>0.000 mm/rev</td></tr> <tr><td>SR</td><td>0.000 rpm</td></tr> <tr><td>Parts grip.:</td><td>No</td></tr> <tr><td>X2</td><td>0.000 abs</td></tr> </table>	T	0.000 mm/rev	F	0 m/min	V	0.000 rpm	SV	8.000 rpm	X0	0.000 abs	Z0	0.000 abs	FS	8.000	X1	0.000 abs	FR	0.000 mm/rev	SR	0.000 rpm	Parts grip.:	No	X2	0.000 abs	Selection chamfer / radius [FS, R]	
T	0.000 mm/rev																										
F	0 m/min																										
V	0.000 rpm																										
SV	8.000 rpm																										
X0	0.000 abs																										
Z0	0.000 abs																										
FS	8.000																										
X1	0.000 abs																										
FR	0.000 mm/rev																										
SR	0.000 rpm																										
Parts grip.:	No																										
X2	0.000 abs																										
X1	<p><b>Cutoff</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/rev</td></tr> <tr><td>F</td><td>0 m/min</td></tr> <tr><td>V</td><td>0.000 rpm</td></tr> <tr><td>SV</td><td>8.000 rpm</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>FS</td><td>0.000</td></tr> <tr><td>X1</td><td>0.000 abs</td></tr> <tr><td>FR</td><td>0.000 mm/rev</td></tr> <tr><td>SR</td><td>0.000 rpm</td></tr> <tr><td>Parts grip.:</td><td>No</td></tr> <tr><td>X2</td><td>0.000 abs</td></tr> </table>	T	0.000 mm/rev	F	0 m/min	V	0.000 rpm	SV	8.000 rpm	X0	0.000 abs	Z0	0.000 abs	FS	0.000	X1	0.000 abs	FR	0.000 mm/rev	SR	0.000 rpm	Parts grip.:	No	X2	0.000 abs	Depth for feed reduction [abs, inc]	
T	0.000 mm/rev																										
F	0 m/min																										
V	0.000 rpm																										
SV	8.000 rpm																										
X0	0.000 abs																										
Z0	0.000 abs																										
FS	0.000																										
X1	0.000 abs																										
FR	0.000 mm/rev																										
SR	0.000 rpm																										
Parts grip.:	No																										
X2	0.000 abs																										

## Section 7

### Cut-off

Notes

Parameter	Help picture	Description and hints
FR  Cutoff T D1 F 0.000 mm/rev V 0 m/min SV 0.000 rpm X0 0.000 abs Z0 0.000 abs FS 0.000 X1 0.000 abs FR 0.000 mm/rev SR 0.000 rpm Parts grip.: No X2 0.000 abs		Reduced feed rate [mm/rev]
SR  Cutoff T D1 F 0.000 mm/rev V 0 m/min SV 0.000 rpm X0 0.000 abs Z0 0.000 abs FS 0.000 X1 0.000 abs FR 0.000 mm/rev SR 0.000 rpm Parts grip.: No X2 0.000 abs		Reduced speed [rev/min]
Parts grip  Cutoff T D1 F 0.000 mm/rev V 0 m/min SV 0.000 rpm X0 0.000 abs Z0 0.000 abs FS 0.000 X1 0.000 abs FR 0.000 mm/rev SR 0.000 rpm Parts grip.: Yes XM 0.000 abs X2 0.000 abs		YES: Extend the parts grip NO: Do not extend the parts grip Hint: The data by the machine manufacturer must be observed.
XM  Cutoff T D1 F 0.000 mm/rev V 0 m/min SV 0.000 rpm X0 0.000 abs Z0 0.000 abs FS 0.000 X1 0.000 abs FR 0.000 mm/rev SR 0.000 rpm Parts grip.: Yes XM 0.000 abs X2 0.000 abs		Depth at which the drawer is extended [abs]
X2  Cutoff T D1 F 0.000 mm/rev V 0 m/min SV 0.000 rpm X0 0.000 abs Z0 0.000 abs FS 0.000 X1 0.000 abs FR 0.000 mm/rev SR 0.000 rpm Parts grip.: No X2 0.000 abs		Final depth [abs, inc]

## 1 Brief description

**Objective of the module:**

This module explains the programming of contours and their machining under ShopTurn.

**Description of the module:**

Apart from the described standard turning functions, ShopTurn also provides for the programming and machining of complex contours. A contour consists of single contour elements, at least two of which up to a max. of 250 make up a defined contour. Additionally chamfers, radii, undercuts or tangential transitions can be programmed between the individual elements.

**Content of the module:**

Selection and general functions under „Contour turning“

- Selection of the function „Contour turning“
- Vertical Softkey strip
- Basic hints

- Depiction of the contour

Creating a contour

Input of contour elements

Alteration of contour elements

- Adding a contour element
- Alteration of a contour element
- Changing the dialogue
- Inserting a contour element
- Deleting a contour element

Contour turning

- Stock removal

- Residue stock removal

- Grooving

- Residue grooving

- Parting

- Residue parting

# Operation Area Program editor - Programming Contour turning

Selection and general functions  
under "Contour turning"

Selection of the function  
"Contour turning"

Vertical Softkey strip

Basic hints

Depiction of the contour

Section 2

Creating a contour

Section 3

Input of contour elements

Section 4

Alteration of contour elements

Adding a contour element

Alteration of a contour ele-  
ment

Changing the dialogue

Inserting a contour element

Deleting a contour element

Section 5

Stock removal and residue stock  
removal

Stock removal

Residue stock removal

Section 6

Grooving and residue grooving

Grooving

Residue grooving

Section 7

Parting and residue parting

Parting

Residue parting

Section 8

## Section 2

### Selection and general functions under "Contour turning"

#### 2.1 Selection of the function "Contour turning"

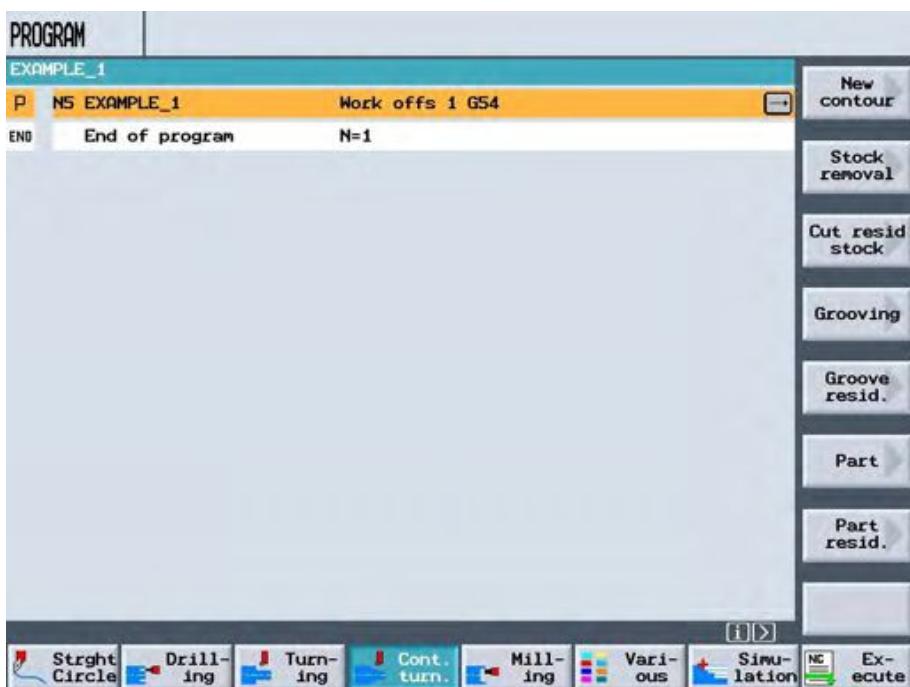
After producing the program header the program blocks for the machining, e.g. the contour turning.

In this module it is assumed that the XY-plane is the machining plane and the Z-axis the tool axis. The selection of the tool axis is made in the program header, [See Module A112 - "Operating Area Program editor - Program header"](#).



By pressing the HSK 4 "Contour turning" select the range for the programming of contours and their machining.

Notes



## Section 2

Notes

Selection and general functions under "Contour turning"

### 2.2 Vertical Softkey strip



By pressing the VSK 1 "New contour" a new contour will be initiated. Following this the contour elements can be programmed.

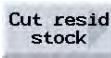
See

- Section 3 "Creating a contour"*,
- Section 4 "Input of contour elements"*
- Section 5 "Hints regarding the contour"*



By pressing the VSK 2 "Stock removal" the respective function will be called up.

See *Section 6.1 "Stock removal"*.



By pressing the VSK 3 "Cut resid. stock" the function Residue stock removal will be called up.

See *Section 6.2 "Cut resid. stock"*.



By pressing the VSK 4 "Grooving" the respective function will be called up.

See *Section 7.1 "Grooving"*.



By pressing the VSK 5 "Groove resid." the function „Residue groove material“ will be called up.

See *Section 7.2 "Residue groove material"*.



By pressing the VSK 6 "Part" the respective function will be called up.

See *Section 8.1 "Part"*.



By pressing the VSK 7 "Part resid." the function „Residue parting material“ will be called up.

See *Section 8.2 "Residue parting material"*.

## Section 2

### Selection and general functions under “Contour turning”

#### 2.3 Basic hints

Notes

**Types of contours:** The following contours can be programmed:  
open contours and  
closed contours

**Number of contour elements:** A contour consists of single contour-elements. A contour must consist of at least 2 elements and at most 250 elements  
Additionally  
chamfers,  
radii or  
tangential transitions  
can be programmed between the contour elements.

**Contour calculator:** The integrated contour calculator evaluates the points of intersection of adjoining contour elements taking into account the geometrical aspects and permits .

**Programming-sequence:** For contour turning the contour must always be programmed first followed by the technology.  
The contours can be machined  
with Stock removal / Residue stock removal  
with Grooving / Residue grooving  
with Parting / Residue parting

#### Examples for the programming sequence of contours

**Any sort of contour:**

1. Enter the contour  
Assemble the contour by entering the various contour elements one after the other.
2. Stock removal (roughing)  
The contour is machined taking into account the approach and leaving strategies.
3. Stock removal (finishing)  
If for roughing a finishing allowance was programmed, the contour will be finished as well.

## Section 2

Notes

Selection and general functions under “Contour turning“

### 2.4 Depiction of the contour



ShopTurn shows a contour in the work plan as a program block (with an open square bracket).



If this block is opened by pressing the key “Cursor right“, The individual contour elements will be depicted symbolically alongside the graphics window.

Contour element	Symbol	Meaning
Start point		Start point of the contour
Straight upwards		Straight in a 90°-grid
Straight downwards		Straight in a 90°-grid
Straight to the right		Straight in a 90°-grid
Straight to the left		Straight in a 90°-grid
Straight any (Taper)		Straight with any angle
Circular arc to the right (clockwise)		Circle (radius)
Circular arc to the left (Anti-clockwise)		Circle (radius)
Contour end		End of contour description

#### Status of the contour-elements

The various colours of the symbols refer to their status:

Symbol	Foreground	Background	Meaning
	black	orange	Cursor points to a new or. current element
	black	white	Normal element
	red	white	Element is presently not being considered

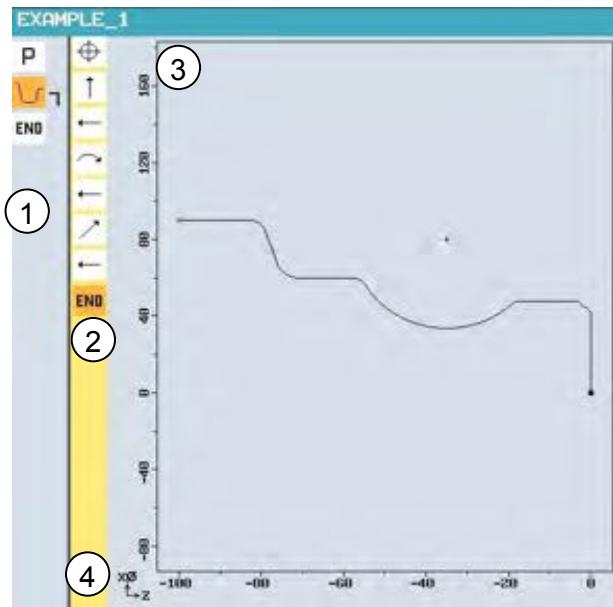
## Section 2

### Selection and general functions under “Contour turning“

#### Graphical depiction

Parallel to the sequential input of the contour elements the progress contour programming is shown in the graphics window as a line graphic.

Notes



- ① Symbols of the work plan
- ② Symbols of the contour elements
- ③ Graphics window with scaling
- ④ Co-ordinate system

#### Depiction of the contour elements

During programming the produced contour elements can take on a variety of line-types and colours.

Black	End of program contour-elements
Orange	Current contour element
Green dotted	Alternative contour element
Blue dotted	Partially defined contour-element

#### Scaling

The scaling of the co-ordinate system changes to suit the changes of the entire contour.

#### Co-ordinate system

The position of the co-ordinate system is shown in the graphics window.

## Section 3

### Creating a contour

Notes

A separate contour must be created for each contour that is to be rotated.

The contours are stored at the end of the program.

If a new contour is to be created, the Start point of the contour must be specified first of all.

ShopTurn automatically defines the end of the contour.

If the tool axis is changed, ShopTurn will automatically change the Start point to suit the axes.

Any number of auxiliary commands (max. 40 characters) can be entered in the form of G-Codes for the Start point.

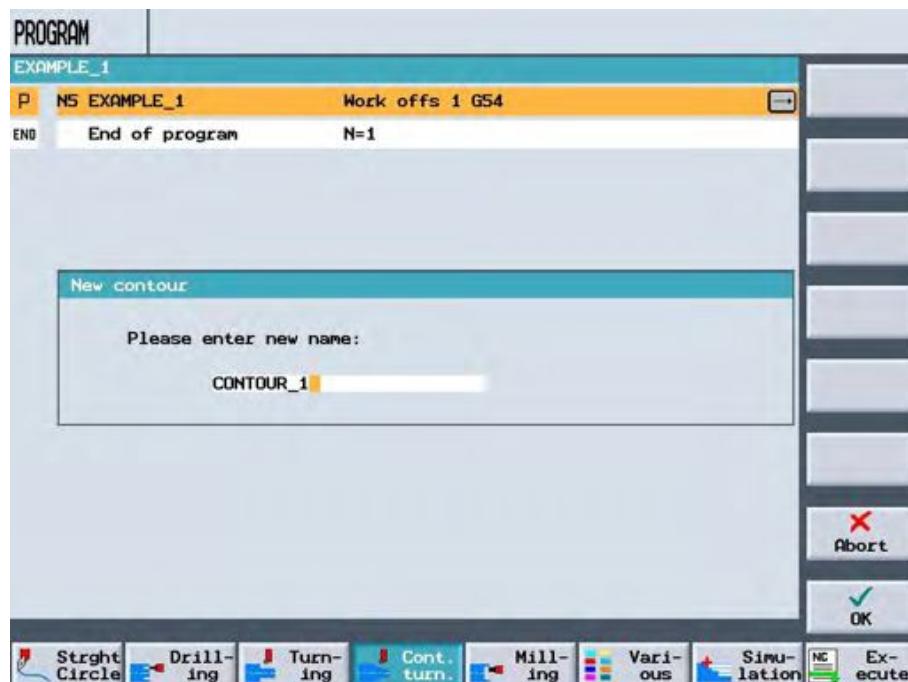
Hint:

If a contour is to be created, which is similar in shape to an existing contour, it is possible to copy the existing contour, rename it and to alter the required contour elements.

If, however, an identical contour is to be used again at another part of the program, the copy must not be renamed. Alterations then made in one of the contours will automatically also be made in the other contour by the same name.

New contour

By pressing the VSK 1 "New contour" the window for the input of the contour name will be opened.



Enter the name for the new contour.  
The contour name must be explicit.

## Section 3

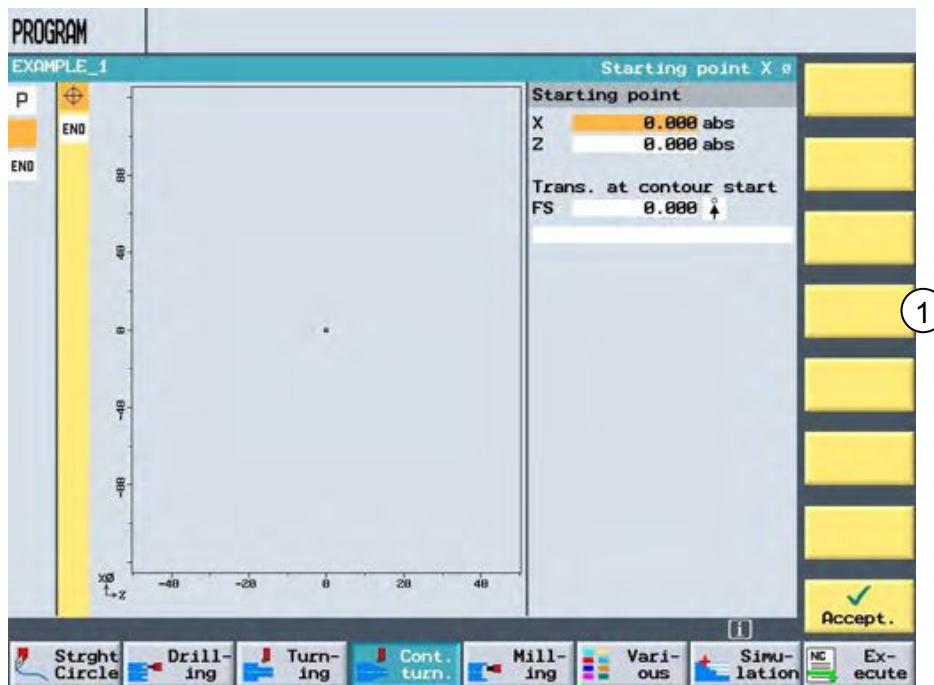
### Creating a contour



By pressing the VSK 8 "OK" a new contour will be initiated.

The window for the input of the contour Start point opens.

Notes



**Start point  
Cartesian:**

Enter the following parameters in the window:

Tool axis

Absolute Start point in X- and Z-axis  
(depending on the selected tool axis)

Any auxiliary commands (max. 40 characters)



By pressing the VSK 8 "Accept." this Start point will be transferred to the contour.

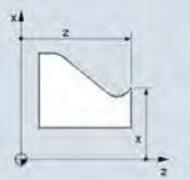
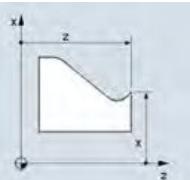
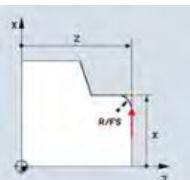
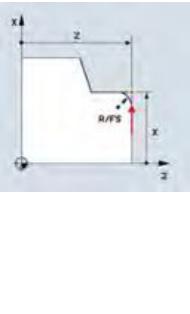
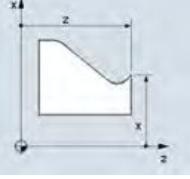
See also Module A100 - "Basic Principles Turning Machine".

## Section 3

Notes

### Creating a contour

#### Parameter “Start point contour”

Parameter	Help picture	Description and hints
X		Start point in X-direction [abs]
Z		Start point in Z-direction [abs]
FS / R		Transition at the contour start
Direction before the contour		Position of the transition element in relation to the Start point of the contour.  Alternatively:    
Auxiliary commands		Any sort of auxiliary command in the form of G-Code (max. 40 characters)

## Section 4

### Input of contour elements

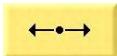
After a new contour has been started and the Start point has been defined, the individual contour elements, which make up the contour, can be defined as well.

Notes

The following contour elements are available for the definition of a contour.



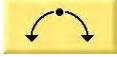
Straight vertical



Straight horizontal



Straight diagonal



Circle/Circular arc

#### Parameter input

If some of the fields are left blank, ShopTurn automatically assumes that these values are unknown and attempts to determine them from other parameters.  
In case of contours, for which more parameters are entered than absolutely necessary, this can lead to conflicts. In such a case try to enter fewer parameters and leave as many as possible for ShopTurn to evaluate.

## Section 4

Notes

Input of contour elements

### Contour transition elements

Between any two contour elements a transition element can be chosen:

- Chamfer
- Radius
- Undercut

The transition element is always added at the end of a contour element. The selection of a contour transition element is made in the parameter mask of the contour element. The contour transition Undercut is an exception to this. This can only be programmed if the next element has been defined and the transition angle is  $90^\circ \pm 2,99^\circ$ .

A contour transition element can always be inserted if there is a point of intersection between the two adjoining elements, which can be calculated from the entered values. Otherwise the contour elements Straight/Circle must be used; e.g. at the start or end of a contour).

Hence in case of a closed contour a transition element can also be programmed between the first and last element. The Start point after programming will then lie outside the contour.

### Auxiliary commands

For each contour element any auxiliary commands in the form of G-Code can be entered; for instance it would be possible to programme a speed reduction or a defined stop for the contour element Circle "G9".

The auxiliary commands (max. 40 characters) can be entered in the extended parameter mask (Softkey „All parameters“).

### Further functions

For the programming of a contour the following further functions are available:

Tangential to previous element  
The transition to the preceding element can be programmed as a tangent.

Dialogue selection  
If from the parameter values entered so far there are two possible contour solutions, one of them must be chosen.

Closing a contour  
From the present position the contour can be closed with a straight to the Start point.

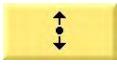
## Section 4

### Input of contour elements

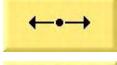
#### Input of contour elements

After the transfer of the Start point, the input of the contour elements can commence. For this the respective vertical Softkeys are available.

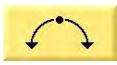
Notes



Select a contour element by pressing the respective vertical Softkey.



Enter all data in the input mask that are available from the shop drawing (e.g. length of a straight, end positions, transition to the next element, pitch angle etc.).

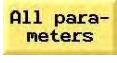


By pressing the VSK 8 "Accept.", transfer the contour element to the contour.

This process is to be repeated until the contour has been completed.



By pressing the VSK 8 "Accept.", transfer the contour to the work plan.



By pressing the VSK 5 "All parameters" it is possible to show further parameters, for instance to add auxiliary commands.

## Section 4

Notes

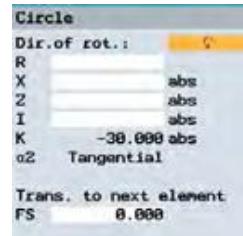
Input of contour elements

### Tangent to the preceding element

Tangent  
prev.elem

By pressing the VSK 2 "Tangent prev. elem" the transition to the preceding element can be programmed as a tangent.

The angle to the preceding element 2 is set to "0". In the input field of the parameters appears the selection „tangential“.



### Dialogue selection

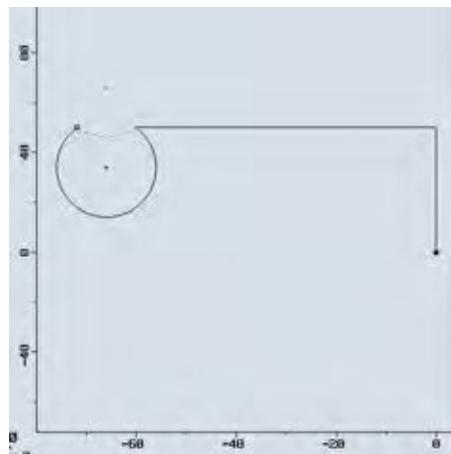
Dialog  
select

During the input for a contour element it is possible that two contour solutions result, one of which must be selected.

By pressing the VSK 1 "Dialog select" it is possible to choose between the two different contour possibilities.

①

②



Example: Circle: Alternative contour possibilities

- ① Alternative 1 (alternative contour element)
- ② Alternative 2 (selected contour element)

Dialog  
accept

By pressing the VSK 8 "Dialog accept" the selected contour possibility (black contour element) will be transferred.

Following this further data for the contour element can be entered or the contour element can be transferred to the contour.

## Section 4

### Input of contour elements

#### Closing a contour

A contour needs to be closed only in case of a raw material description. If you do not wish to define all elements from Start to Start, the contour can be closed from the respective present position to the Start.

Notes

More

Close contour

By pressing the VSK 6 "More" followed by the VSK 6 „Close contour“ the contour will be closed automatically by straights:

From the present position to the turning centre

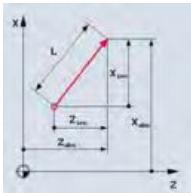
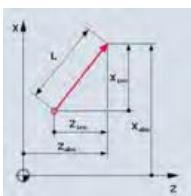
Straight in Z, to the Start position of the Z-axis.

Straight in X to the Start point

A transition element (chamfer/radius) may be added to this straight.

Listed in the following table are the parameters for the contour elements „Straight in X-direction“, „Straight in Z-direction“ and „Straight diagonal“. However, only those parameters relevant for the respective straight will be shown.

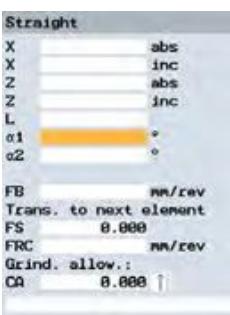
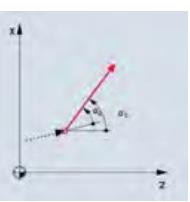
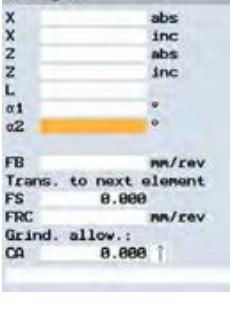
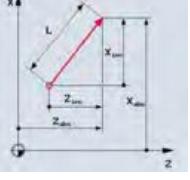
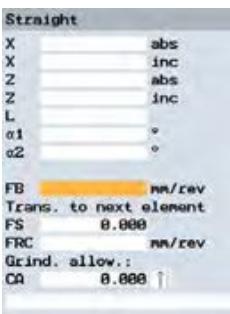
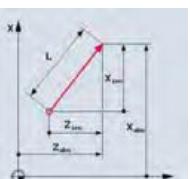
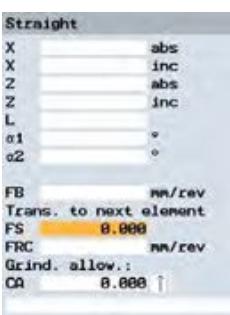
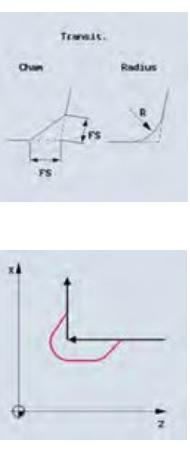
Parameter for the contour element "Straight diagonal"; the VSK "All parameters" is pressed:

Parameter	Help picture	Description and hints														
X, Z	 <b>Straight</b> <table border="1"> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td>*</td></tr> <tr><td><math>\alpha_1</math></td><td>*</td></tr> <tr><td><math>\alpha_2</math></td><td>*</td></tr> </table> <p>FB mm/rev Trans. to next element FS 0.000 FRC mm/rev Grind. allow.: CA 0.000 ↑</p>	X	abs	X	inc	Z	abs	Z	inc	L	*	$\alpha_1$	*	$\alpha_2$	*	End point in X - and Z - Direction [abs, inc]  Incremental dimension: The sign will be taken into account.
X	abs															
X	inc															
Z	abs															
Z	inc															
L	*															
$\alpha_1$	*															
$\alpha_2$	*															
L	 <b>Straight</b> <table border="1"> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td>*</td></tr> <tr><td><math>\alpha_1</math></td><td>*</td></tr> <tr><td><math>\alpha_2</math></td><td>*</td></tr> </table> <p>FB mm/rev Trans. to next element FS 0.000 FRC mm/rev Grind. allow.: CA 0.000 ↑</p>	X	abs	X	inc	Z	abs	Z	inc	L	*	$\alpha_1$	*	$\alpha_2$	*	Length
X	abs															
X	inc															
Z	abs															
Z	inc															
L	*															
$\alpha_1$	*															
$\alpha_2$	*															

## Section 4

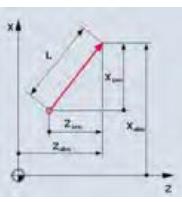
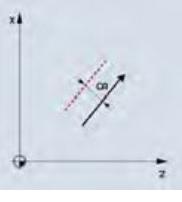
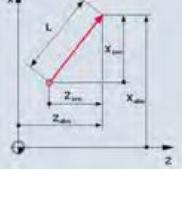
Notes

### Input of contour elements

Parameter	Help picture	Description and hints
<b>a1</b>  		Start angle to the Z-axis [Degrees]
<b>a2</b>  		Angle to the preceding element [Degrees]
<b>FB</b>  		Element referred feed rate [mm/rev]
<b>FS / R / Undercut</b>  		Transition to the next element [Chamfer / radius]  [Undercut] Hint: This can only be programmed if the next element has been defined and the transition angle is 90° +/- 2,99°. See also Module A117- "Program editor - Turning standard cycles - Undercut"

## Section 4

### Input of contour elements

Parameter	Help picture	Description and hints	Notes
FRC		Element referred feed rate Chamfer / Rounding / Undercut [mm/rev]	
CA		Grinding allowance left / right of the contour	
Grinding allowance		Grinding allowance left / right of the contour Alternatively:  	
Auxiliary commands		For each contour element any auxiliary commands in the Form of G-Code can be entered. The auxiliary commands (max. 40 characters) can be selected in the extended parameter mask.	

## Section 4

Notes

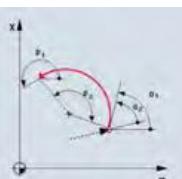
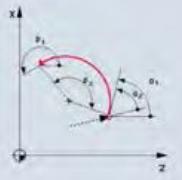
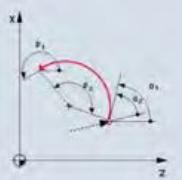
### Input of contour elements

Parameter for the contour element "Circle"; VSK "All parameters" pressed:

Parameter	Help picture	Description and hints
Turning direction		Selection of the turning direction Rotation clockwise Rotation anti-clockwise
R		Radius of the circle [mm]
X, Z		End point X / Z See Range "Straight"
I, K		Circle centre point [abs, inc] Incremental dimension: The sign will be taken into account. Hint: The help picture alongside is valid only for „inc“. In the input field „abs“ the position of the centre point is programmed with reference to the diameter of the work piece.
a1, a2		See section „Straight“

## Section 4

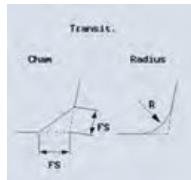
### Input of contour elements

Parameter	Help picture	Description and hints	Notes
$\beta_1$		End angle to the Z-axis [Degrees]	
$\beta_2$		Opening angle [Degrees]	
FB		Element referred feed rate [mm/rev]	

## Section 4

### Input of contour elements

Notes

Parameter	Help picture	Description and hints
FS / R  <b>Circle</b> Dir. of rot.: ? R      abs X      inc Z      abs Z      inc I      abs I      inc K      abs K      inc a1      ° a2      ° p1      ° p2      °  FB      mm/rev Trans. to next element FS      0.000 FRC      mm/rev Grind. allow.: CR      0.000		Chamfer / Rounding  Alternatively: [Chamfer, Radius]
FRC, CA, Auxiliary commands		See section „Straight“

## Section 5

### Alteration of contour elements

An existing contour can be changed in retrospect. Individual contour elements can be:

- added
- altered
- inserted or
- deleted

Hint:

If in one program there are two contours defined by the same name, any alterations made in the one contour will automatically taken over in the other one with the same name..

#### 5.1 Adding a contour element



Selection within the contour.

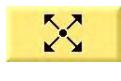


Open the contour.

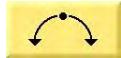
The individual contour elements will be listed.



Set the cursor on the last contour element before the end.



Select the respective contour element on the vertical Softkey strip; e.g. "Straight" or "Circle/Circular arc".



Enter the parameter in the input mask.



Press the VSK "Accept.".

The required contour element will be added to the contour.

Notes

## Section 5

Notes

### Alteration of contour elements

#### 5.2 Alteration of a contour element



Select the contour in the work plan.



Open the contour.

The individual contour elements will be listed.



Set the cursor on the last contour element before the end.



By pressing the key "Cursor right" the associated input mask will open and the selected element will be depicted enlarged in the programming graphics.

Enter the required alterations.



Press the VSK "Accept".

The current values of the contour element will be transferred and the alterations will be depicted in the programming graphics.

#### 5.3 Changing the dialogue

If during the data input for a contour element two different solutions resulted and the wrong alternative was selected, the selection can be changed again afterwards. If an explicit contour resulted from other parameters, no further dialogue will appear.



Open the input mask for the contour element.



Press the VSK "Change selection".

Both contour variants will be shown.



By pressing the VSK "Dialog select" permits the toggling between the two alternatives.



By pressing the VSK "Dialog accept" the selected contour alternative will be transferred to the contour.

## Section 5

### Alteration of contour elements

#### 5.4 Inserting a contour element

Notes



Select the contour in the work plan.

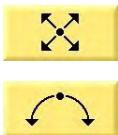


Open the contour.

The individual contour elements will be listed.



Set the cursor on the last contour element, behind which the contour element is to be inserted.



Select the new contour element via the vertical Softkey strip; e.g. "Straight" or "Circle/Circular arc".

Enter the parameter in the input mask.



Press the VSK "Accept.".

The required contour element will be added to the contour.



By selection of the succeeding contour element all successive will be re-evaluated and depicted accordingly in the graphics window.

The status of these contour elements changes from a red to a black foreground.

It may be possible that the end point of the inserted element does not suit the Start point of the following element. In this case ShopTurn displays the error message "Geometry values controversial". If this is to be rectified, a taper without any parameter values must be inserted.

## Section 5

Notes

### Alteration of contour elements

#### 5.4 Deleting a contour element



Select the contour in the work plan.



Open the contour.

The individual contour elements will be listed.



Set the cursor on the contour element that is to be deleted.

Delete element

Press the VSK 1 "Delete element".

OK

Press the VSK 8 "OK".

The contour element will be deleted and the contour will be shown according to the new situation in the graphics window.

## Section 6

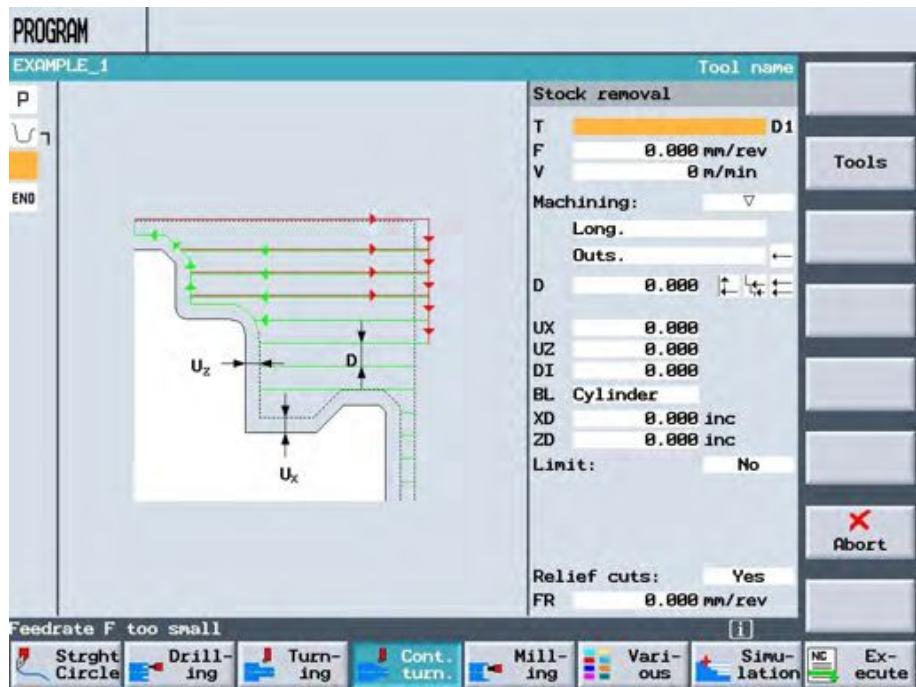
### Stock removal / Residue stock removal

#### 6.1 Stock removal

Stock removal

By pressing the VSK 2 "Stock removal" the respective function for the machining will be selected.

Notes



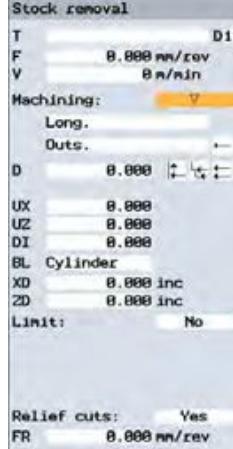
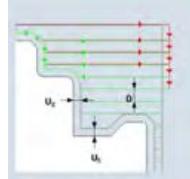
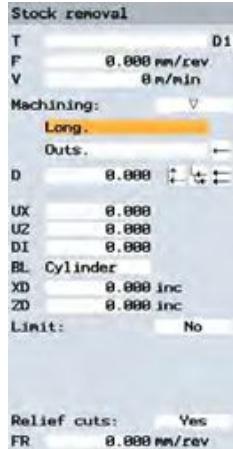
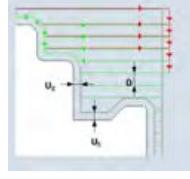
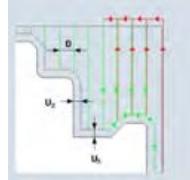
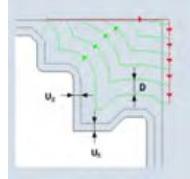
With the function "Stock removal" "Cut resid. stock" machining along any programmed contour both longitudinally or planer is possible.

## Section 6

Notes

### Stock removal / Residue stock removal

#### Parameter der Funktion "Stock removal" and "Cut resid. stock"

Parameter	Help picture	Description and hints
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Threads, Positions", Section 3 "Centring".
Machining	 	Selection of the machining Alternatively:   Hint: For finishing the following input fields are available: Allowance NO / Yes - Input parameter "U1" Limitation NO / YES – See Section Limitation in this Module Undercuts NO / YES
Direction of stock removal	   	Selection of direction of stock removal <b>Long.</b> [External, Internal] Selection of machining direction   <b>Face</b> [Front end, Rear end] Selection of machining direction   <b>Contour parallel</b> [External, Internal, Front end, Rear end] Selection of machining direction   Front end, Rear end   External, Internal

## Section 6

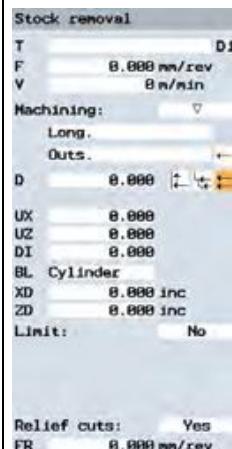
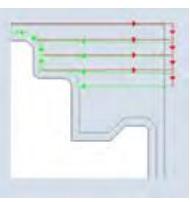
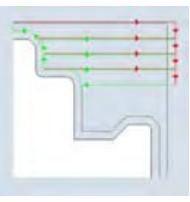
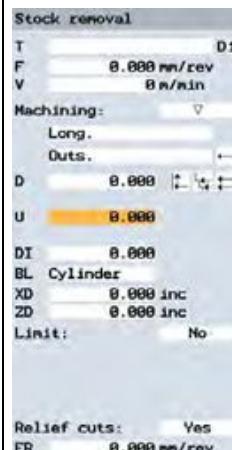
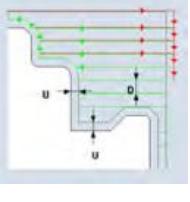
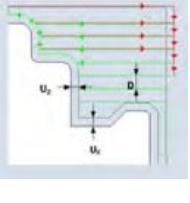
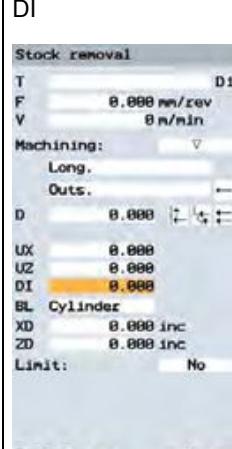
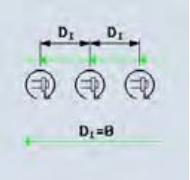
### Stock removal / Residue stock removal

Parameter	Help picture	Description and hints	Notes
D  Stock removal T D1 F 0.000 mm/rev V 0 m/min Machining: Long. Outs. D 0.000 UX 0.000 UZ 0.000 DI 0.000 BL Cylinder XD 0.000 inc ZD 0.000 inc Limit: No  Relief cuts: Yes FR 0.000 mm/rev		Maximum infeed  For longitudinal and planar [D]  For parallel to contour [D, Dx, Dz]	
Contour smoothing  Stock removal T D1 F 0.000 mm/rev V 0 m/min Machining: Long. Outs. D 0.000 UX 0.000 UZ 0.000 DI 0.000 BL Cylinder XD 0.000 inc ZD 0.000 inc Limit: No  Relief cuts: Yes FR 0.000 mm/rev		Contour smoothing  Automatic smoothing of the contour  Smoothing of the contour - always  Smoothing of the contour - never  Hint: only available for longitudinal and planar	
Cut segmentation  Stock removal T D1 F 0.000 mm/rev V 0 m/min Machining: Long. Outs. D 0.000 UX 0.000 UZ 0.000 DI 0.000 BL Cylinder XD 0.000 inc ZD 0.000 inc Limit: No  Relief cuts: Yes FR 0.000 mm/rev		Selection of cut segmentation  Align cut segmentation with edges  Equal cut segmentation  Hint: only available for longitudinal and planar	

## Section 6

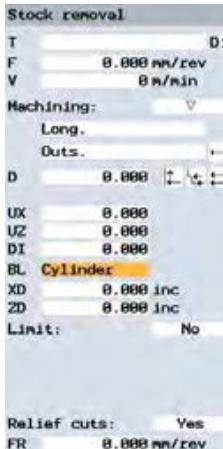
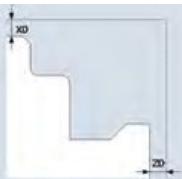
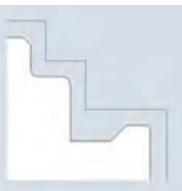
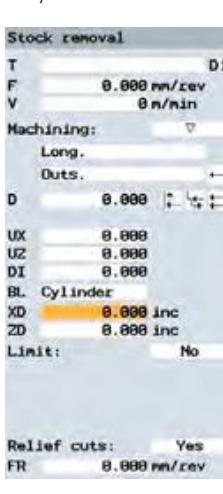
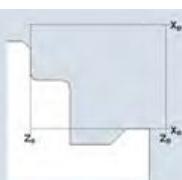
### Stock removal / Residue stock removal

Notes

Parameter	Help picture	Description and hints
Cutting depth	  	<p>Selection of cutting depth</p>  <p>Constant cutting depth</p> <p>Hint: Only available for "Align cut segmentation with edges".</p> <p>Selection of cutting depth</p>  <p>Changing cutting depth</p> <p>Hint: Only available for longitudinal and planar</p> <p>Hint: Only available for "Align cut segmentation with edges".</p>
U, UX, UZ	  	<p>Finishing allowance</p> <p>[U]</p> <p>[UX, UZ]</p>
DI	 	<p>Distance feed interruption</p>

## Section 6

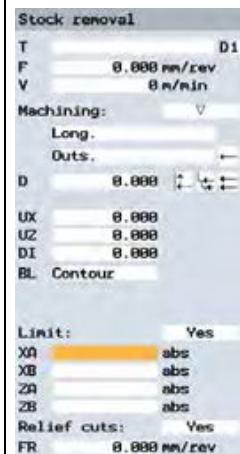
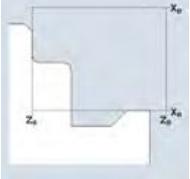
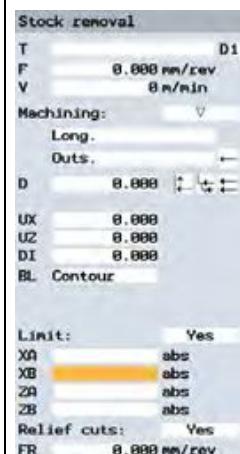
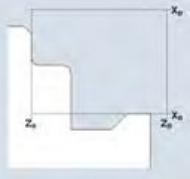
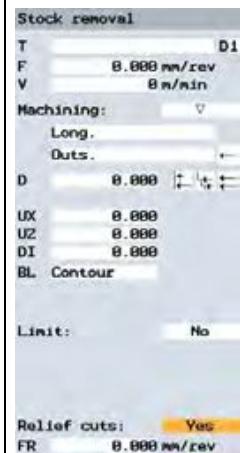
### Stock removal / Residue stock removal

Parameter	Help picture	Description and hints	Notes
BL		Raw material description <b>Cylinder</b>	
	  	<b>Cylinder</b> <b>Allowance</b> <b>Contour</b>	
XD, ZD		Allowance or Cylinder dimension [abs, inc] Not available for "Contour"	
		Limitation of machining range [NO, YES]	

## Section 6

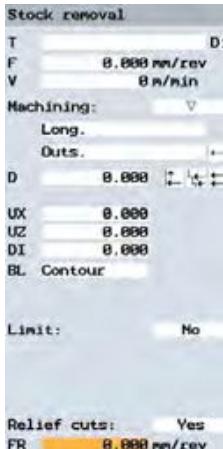
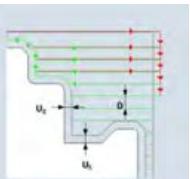
### Stock removal / Residue stock removal

Notes

Parameter	Help picture	Description and hints
<b>XA, ZA</b> 		Limit X, Z [abs]
<b>XB, ZB</b> 		Limit X, Z [abs, inc]
<b>Undercuts</b> 		Undercuts <span style="background-color: orange; color: white; padding: 2px;">Yes</span> <span style="background-color: orange; color: white; padding: 2px;">No</span>

## Section 6

### Stock removal / Residue stock removal

Parameter	Help picture	Description and hints	Notes
FR  		Plunging feed rate for undercuts [mm/rev]  Available only with the selection "YES"	

## 6.2 Residue stock removal

If the material, that remained during Stock removal against the contour, is to be removed, the function "Residue stock removal" can be used. In case of Stock removal against the contour ShopTurn recognizes automatically whether any residue material has remained and generates an up-dated raw material contour. Material that remains due to a finishing allowance is no residue material. With the function "Residue stock removal" the excess material is machined with a further tool. The function "Residue material removal" is a Software-Option.

**Cut resid stock**

By pressing the VSK 3 "Cut resid. stock" the respective function for the machining is selected.

For the parameter see the Section "Stock removal".

## Section 7

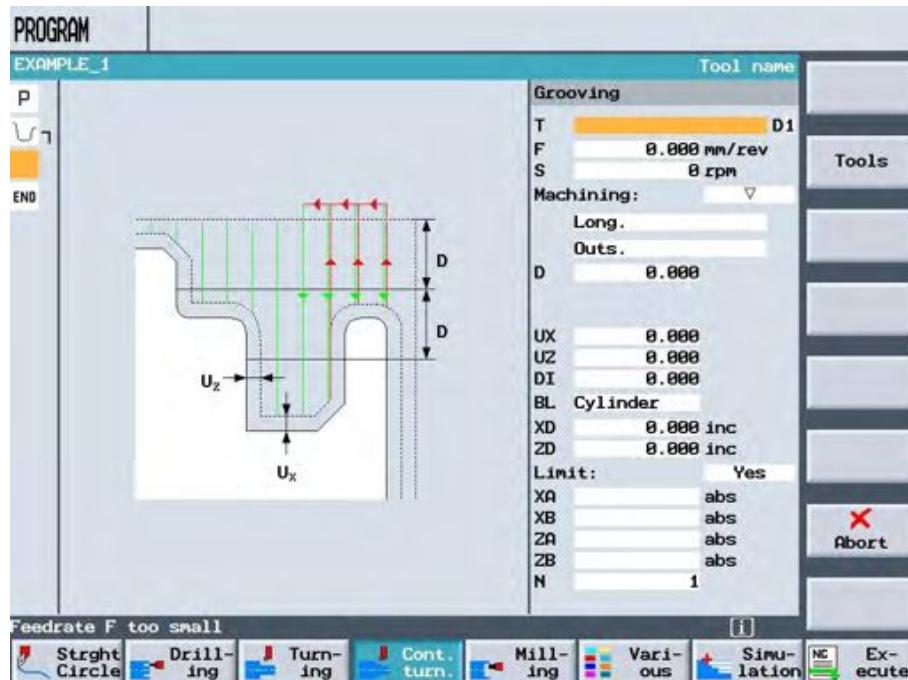
Notes

Grooving

### Grooving / Residue grooving

#### 7.1 Grooving

By pressing the VSK 4 "Grooving" the function "Grooving" will be selected.

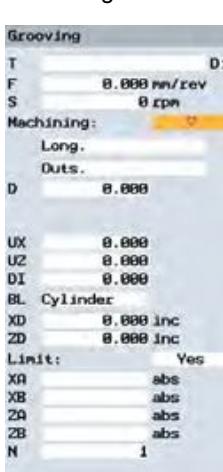
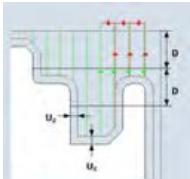
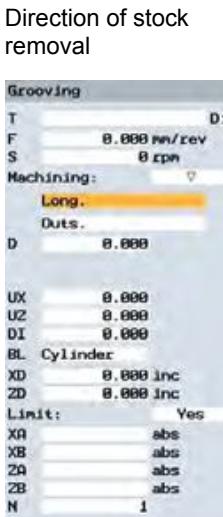
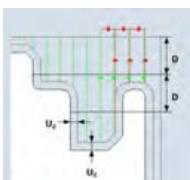
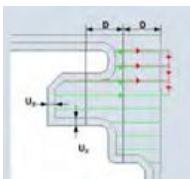
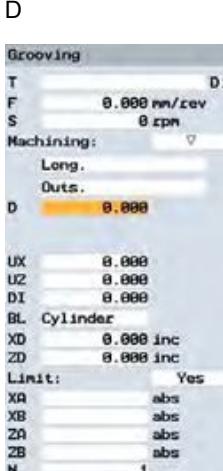
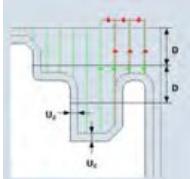


Before the groove can be programmed , the contour of the groove has to be entered.

If a groove is wider than the width of the active tool, the width will be cut in several steps. For this the tool will be re-located (maximum 80% of the tool width) for each successive cut.

## Section 7

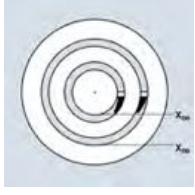
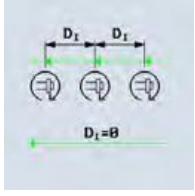
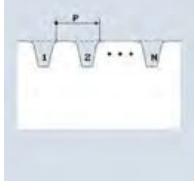
### Grooving / Residue grooving

Parameter	Help picture	Description and hints	Notes
T, D, F, S / V		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Threads, Positions", Section 3 "Centring".	
Machining 	 Selection of machining Alternatively:   Hint: For finishing the following input field is available: Allowance NO / YES - Input parameter "U1" Limitation NO / YES – See Section Limitation in this Module Undercuts NO / YES		
Direction of stock removal 	 Selection of direction of stock removal <b>Long.</b> [External, Internal]   <b>Face</b> [Front end, Rear end]		
D 	 Maximum infeed		

## Section 7

### Grooving / Residue grooving

Notes

Parameter	Help picture	Description and hints
XDA, XDB		<p>1. / 2. Grooving limit Only available in direction of stock removal "Planar".</p>
U, UX, UZ		See section "Stock removal" / „Cut resid. stock“
DI		Distance feed interruption
BL, XD, ZD, Eingrenzen		See section "Stock removal" / "Cut resid. stock"
N		Number of grooves

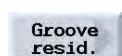
## Section 7

### Grooving / Residue grooving

Parameter	Help picture	Description and hints	Notes
P  Grooving: T D1 F 8.000 mm/rev S 8 rpm Machining: Long. Dots. D 0.000  UX 8.000 UZ 8.000 DI 8.000 BL Cylinder XD 8.000 inc ZD 8.000 inc Limit: Yes XA abs XB abs ZA abs ZB abs N 2 P 0.000 inc		Distance between grooves [inc] Only available if "N" > 1.	

## 7.2 Residue grooving

If the material, that remained during grooving, is to be removed, the function "Residue groove material" can be used. During the grooving ShopTurn recognizes automatically whether any residue material has remained and generates an up-dated raw material contour. Material that remains due to a finishing allowance is no residue material. With the function "Residue stock removal" the excess material is machined with a further tool. The function "Residue material removal" is a Software-Option.



By pressing the VSK 5 "Groove resid." the respective function for the machining is selected.

For the parameter input see Section "Grooving".

## Section 8

### Parting / Residue parting

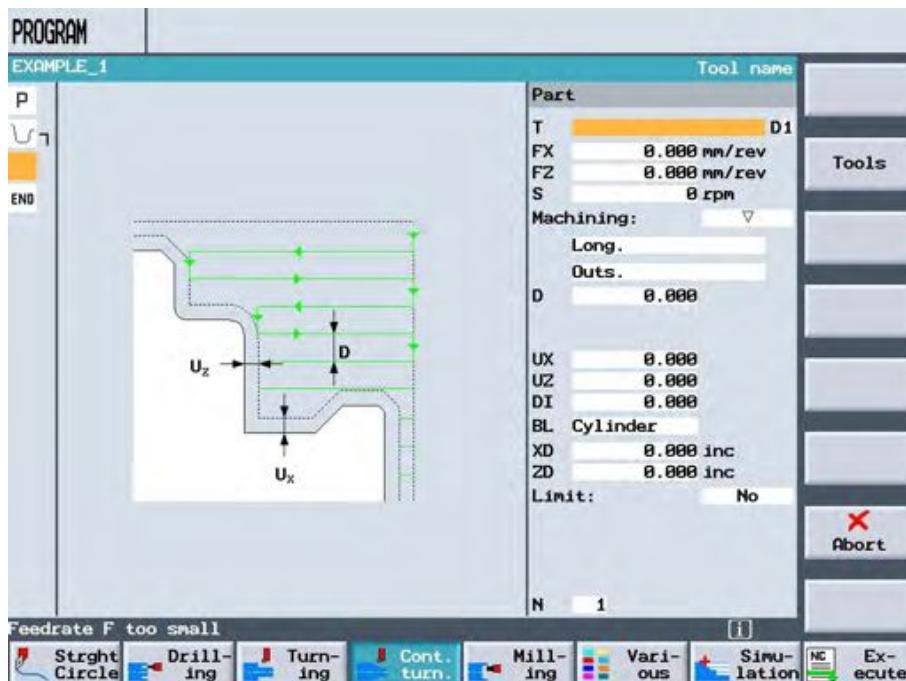
Notes

#### 8.1 Parting

If a groove of any shape is to be machined , the function "Part" must be used. When parting, as compared to grooving, lateral machining takes place after every plunge such that the machining time is considerably reduced. Contrary to Stock removal it is therefore possible to also machine contours with Parting, where a vertical infeed is required.



By pressing the VSK 4 "Part" the function "Grooving" is selected.

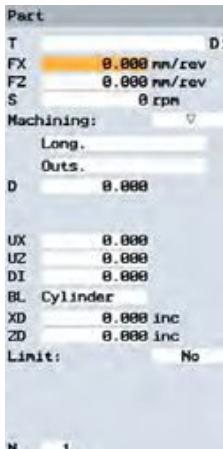


Hint:

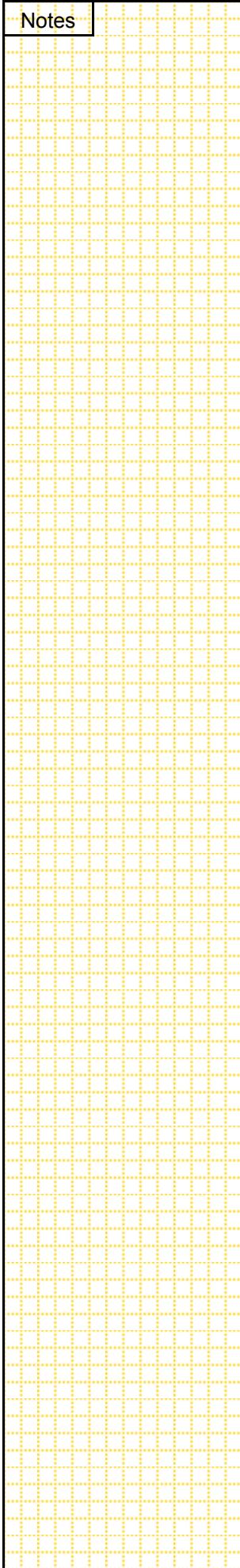
For the Parting a special tool is required that is capable of plunging and which in addition must be capable of taking the lateral cutting forces.

## Section 8

### Parting / Residue parting

Parameter	Help picture	Description and hints	Notes
T, D		See Module A115 – "Operating Area Program - Programming Centring, Drilling, Threads, Positions", Section 3 "Centring".	
FX		Feed rate in X-direction [mm/rev]	
FZ		Feed rate in Z-direction [mm/rev]	
S, Machining, Direction of stock removal, D, XDA, XDB U, UX, UZ, DI, BL, XD, ZD, Eingrenzen,		See section "Grooving" / "Groove resid."	

Notes



## 1 Brief description

**Objective of the module:**

With the aid of this module you will learn to programme the functions of standard milling cycles.

**Description of the module:**

This module describes in general the functions and features of ShopTurn regarding the standard milling cycles. The standard milling cycles include the milling of:

- Pockets (rectangular and circular pockets)
- Spigot (rectangular and circular spigot)
- Slots (longitudinal and circular slots)
- Multi-edges
- Engravings

**Contents of the module:**

- Selection and general functions under "Milling"
  - Selection of the function "Milling"
  - Vertical Softkey strip
- Pocket
  - Rectangular pocket
  - Circular pocket
- Spigot
  - Rectangular spigot
  - Circular spigot
- Slot
  - Longitudinal slot
  - Circular slot
- Position pattern when milling
- Engraving cycle
  - Vertical Softkey strip

# Operating Area Program editor - Programming milling/Standard cycles

Selection and general functions  
under "Milling"

Selection of the function  
"Milling"

Vertical Softkey strip

Section 2

Pocket

Rectangular pocket

Circular pocket

Section 3

Spigot

Rectangular spigot

Circular spigot

Section 4

Slot

Longitudinal slot

Circular slot

Section 5

Position pattern when milling

Section 6

Multi-edges

Section 7

Engraving cycle

Vertical Softkey strip

Section 8

## Section 2

### Selection and general functions under "Milling"

#### 2.1 Selection of the function "Milling"

After the definition of the program heading the program blocks for the machining, e.g. milling of pockets, spigots or slots, can be programmed.



By pressing the HSK 3 "Milling" select the section for the programming of standard milling cycles.



Notes

#### 2.2 Vertical Softkey strip

Pocket

By pressing the VSK 1 "Pocket" the respective functions for the programming of rectangular and circular pockets is called up.

See Section 3 "Pocket".

Spigot

By pressing the VSK 2 "Spigot" the respective function for the programming of rectangular and circular spigots is called up.

See Section 4 "Spigot".

Slot

By pressing the VSK 3 "Slot" the respective function for the programming of longitudinal and circular slots is called up.

See Section 5 "Slot".

Multi-edge

By pressing the VSK 4 "Multi-edge" the respective function for the programming of Multi-edges will be called up.

See Section 6 "Multi-edge".

Engraving

By pressing the VSK 7 "Engraving" the respective function is called up.

See Section 7 "Engraving".

Contour milling

By pressing the VSK 8 "Contour milling" the function for the programming of contours will be called up. The contour milling will be described in Module A123 "Operating Area Program - Programming Contour milling".

## Section 3

Notes

### Pocket

ShopTurn provides the means for the milling of pockets in the section 'Milling'.

A differentiation is made between rectangular and circular pockets.

#### 3.1 Rectangular pockets

The following variants are available for machining rectangular pockets :

Milling a rectangular pocket into the full material

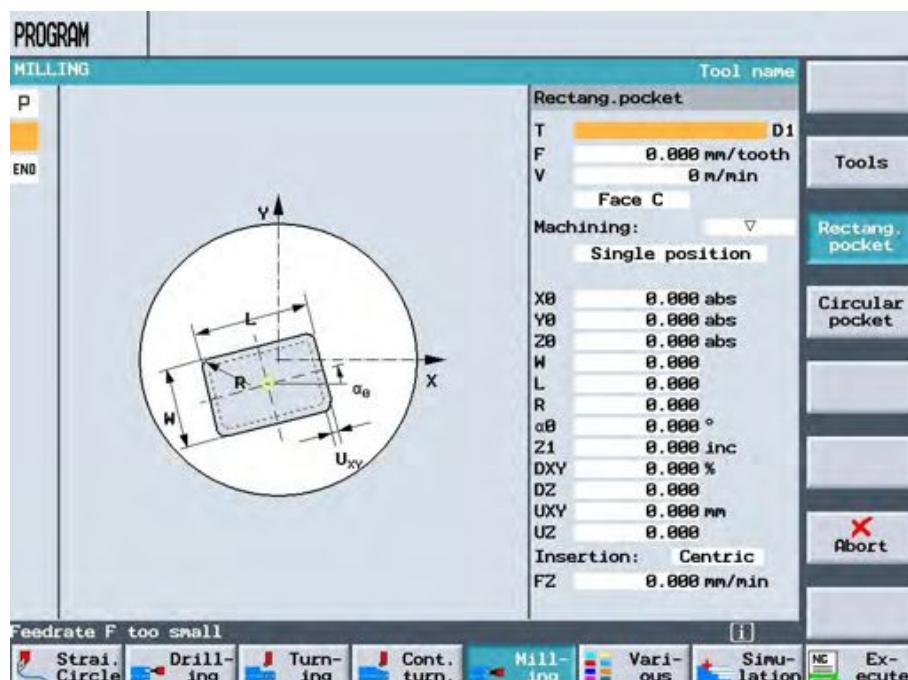
Rectangular pocket with centric pre-drilling, for instance if a slot drill cannot be used (programme the blocks drilling, rectangular pocket and position one after the other).

Pocket

By pressing the VSK 2 "Pocket" select the section for the programming of pockets.

Rectang.  
pocket

By pressing the VSK 3 "Rectang. pocket" select the section for the programming of rectangular pockets.

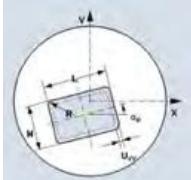
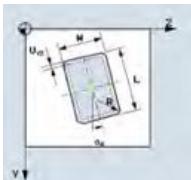
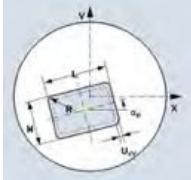
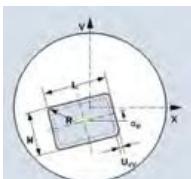


**Approach/leaving  
the contour:**

1. The tool traverses with rapid traverse at the level of the retraction plane to the centre of the pocket and then feeds to the safety distance.
2. The tool enters the material depending on the chosen strategy.
3. The machining of the pocket takes place with the chosen type of machining always from the inside outward.
4. Retraction with rapid traverse to the safety distance.

## Section 3

### Pocket

Parameter	Help picture	Description and hints		Notes
T, D, F, S / V		<i>See module A115 – "Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".</i>		
Location	<p><b>Rectang.pocket</b></p> <pre> T           D1 F           0.000 mm/tooth V           0 m/min Face C Machining: ▽ Single position X0          0.000 abs Y0          0.000 abs Z0          0.000 W           0.000 L           0.000 R           0.000 αθ         0.000 ° Z1          0.000 Jnc DXV         0.000 % D2          0.000 UXY         0.000 mm UZ          0.000 Insertion: Centric FZ          0.000 mm/min </pre>	 	<b>Face C</b> <b>Face Y</b>	Face C Face Y
Machining	<p><b>Rectang.pocket</b></p> <pre> T           D1 F           0.000 mm/tooth V           0 m/min Face C Machining: ▽ Single position X0          0.000 abs Y0          0.000 abs Z0          0.000 W           0.000 L           0.000 R           0.000 αθ         0.000 ° Z1          0.000 Jnc DXV         0.000 % D2          0.000 UXY         0.000 mm UZ          0.000 Insertion: Centric FZ          0.000 mm/min </pre>		▽ ▽▽▽ ▽▽▽ wall	When roughing the individual planes of the pocket are machined from the centre outward until the depth Z1 is attained.  When finishing the edge is always machined first. For this the edge is approach with a quarter circle which leads into the corner radius. With the last infeed the bottom is also machined from the centre outward.  The finishing of the edge is similar to finishing, only the machining of the bottom with the last infeed is omitted.
Position	<p><b>Rectang.pocket</b></p> <pre> T           D1 F           0.000 mm/tooth V           0 m/min Face C Machining: ▽ Single position X0          0.000 abs Y0          0.000 abs Z0          0.000 W           0.000 L           0.000 R           0.000 αθ         0.000 ° Z1          0.000 Jnc DXV         0.000 % D2          0.000 UXY         0.000 mm UZ          0.000 Insertion: Centric FZ          0.000 mm/min </pre>		Single position <b>Single position</b> A rectangular pocket is to be milled at the programmed position (X0, Y0, Z0). Position pattern <b>Position pattern</b> Several rectangular pockets are to be milled on a position pattern (e.g. full circle, pitch circle, grid etc.). <i>See Section 7 – "Position pattern when milling".</i>	

## Section 3

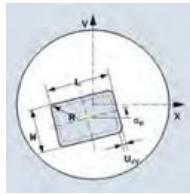
### Pocket

Notes

X0, L0

```
Rectang.pocket
T          D1
F      0.000 mm/tooth
V      8 m/min
Face C
Machining: 
Single position
X0      0.000 abs
Y0      0.000 abs
Z0      0.000 abs
H      0.000
L      0.000
R      0.000
aθ      0.000 °
Z1      0.000 inc
DXY      0.000 %
DZ      0.000
UXY      0.000 mm
UZ      0.000
Insertion: Centric
FZ      0.000 mm/min
```

Face C:



X0: Reference point in X-direction

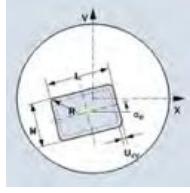
[abs]

Alternatively:

L0: Reference point polar length

Y0, C0

```
Rectang.pocket
T          D1
F      0.000 mm/tooth
V      8 m/min
Face C
Machining: 
Single position
X0      0.000 abs
Y0      0.000 abs
Z0      0.000 abs
H      0.000
L      0.000
R      0.000
aθ      0.000 °
Z1      0.000 inc
DXY      0.000 %
DZ      0.000
UXY      0.000 mm
UZ      0.000
Insertion: Centric
FZ      0.000 mm/min
```



Y0: Reference point in Y-direction

[abs]

Alternatively:

C0: Reference point polar angle

[Degrees]

## Section 3

### Pocket

Parameter	Help picture	Description and hints	Notes
Face C:			
Z0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining: Single position</p> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 αθ 0.000 ° Z1 0.000 inc DXV 0.000 % D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>	<p>Reference point in Z-direction [abs]</p>	
Z1	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining: Single position</p> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 αθ 0.000 ° Z1 0.000 inc DXV 0.000 % D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>	<p>Depth referred to Z0 [mm]</p>	
DXY	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining: Single position</p> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 αθ 0.000 ° Z1 0.000 inc DXY 0.000 % D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>	<p>Maximum infeed in the XY-plane [mm]</p> <p>Alternatively:</p> <p>Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm)</p> <p>[%]</p> <p>Available only for "Roughing" and "Finishing"</p>	

## Section 3

### Pocket

Notes

Parameter

Help picture

Description and hints

Face C:	
DZ	<p>Rectang.pocket</p> <p>T D1 F 0.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining: Single position</p> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 aB 0.000 ° Z1 0.000 inc DXY 0.000 % DZ 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>
UXY	<p>Rectang.pocket</p> <p>T D1 F 0.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining: Single position</p> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 aB 0.000 ° Z1 0.000 inc DXY 0.000 % DZ 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>
UZ	<p>Rectang.pocket</p> <p>T D1 F 0.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining: Single position</p> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 aB 0.000 ° Z1 0.000 inc DXY 0.000 % DZ 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>

## Section 3

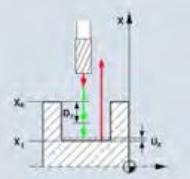
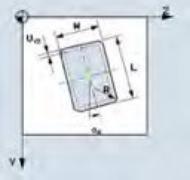
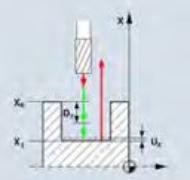
### Pocket

Parameter	Help picture	Description and hints	Notes
Perimeter C:			
Y0, C0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Per.surf.C</p> <p>Machining: Single position</p> <p>Y0 0.000 abs Z0 8.000 abs X0 0.000 abs W 8.000 L 8.000 R 8.000 αθ 8.000 ° X1 8.000 inc DY2 8.000 % DX 8.000 UY2 8.000 mm UX 8.000 Insertion: Centric FX 8.000 mm/min</p>	<p>Y0: Reference point in Y-direction [abs]</p> <p>Alternatively:</p> <p>C0: Reference point polar angle [Degrees]</p>	
Z0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Per.surf.C</p> <p>Machining: Single position</p> <p>Y0 0.000 abs Z0 0.000 abs X0 0.000 abs W 8.000 L 8.000 R 8.000 αθ 8.000 ° X1 8.000 inc DY2 8.000 % DX 8.000 UY2 8.000 mm UX 8.000 Insertion: Centric FX 8.000 mm/min</p>	<p>Reference point in Z-direction [abs]</p> <p>Alternatively:</p> <p>If C0: Reference point polar length [abs]</p>	
X0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Per.surf.C</p> <p>Machining: Single position</p> <p>Y0 0.000 abs Z0 8.000 abs X0 0.000 abs W 8.000 L 8.000 R 8.000 αθ 8.000 ° X1 8.000 inc DY2 8.000 % DX 8.000 UY2 8.000 mm UX 8.000 Insertion: Centric FX 8.000 mm/min</p>	<p>Cylinder diameter <math>\emptyset</math> [abs]</p>	

## Section 3

### Pocket

Notes

Parameter	Help picture	Description and hints
Perimeter C:		
X1	 <pre> Rectang.pocket T           D1 F     8.000 mm/tooth V      8 m/min Per.surf.C Machining: Single position Y0   0.000 abs Z0   0.000 abs X0   0.000 abs W   0.000 H   0.000 L   0.000 R   0.000 o8   0.000 ° X1  0.000 inc DY2  0.000 % DX   0.000 UV2  0.000 mm UX   0.000 Insertion: Centric FX   8.000 mm/min </pre>	Depth referred to X0 $\emptyset$ [abs, inc]
DYZ	 <pre> Rectang.pocket T           D1 F     8.000 mm/tooth V      8 m/min Per.surf.C Machining: Single position Y0   0.000 abs Z0   0.000 abs X0   0.000 abs W   0.000 H   0.000 L   0.000 R   0.000 o8   0.000 ° X1  0.000 inc DY2  0.000 % DX   0.000 UV2  0.000 mm UX   0.000 Insertion: Centric FX   8.000 mm/min </pre>	Maximum infeed in the YZ-plane Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%], [mm] Available only for "Roughing" and "Finishing".
DX	 <pre> Rectang.pocket T           D1 F     8.000 mm/tooth V      8 m/min Per.surf.C Machining: Single position Y0   0.000 abs Z0   0.000 abs X0   0.000 abs W   0.000 H   0.000 L   0.000 R   0.000 o8   0.000 ° X1  0.000 inc DY2  0.000 % DX  0.000 UV2  0.000 mm UX   0.000 Insertion: Centric FX   8.000 mm/min </pre>	Max. Infeed depth - X-direction

## Section 3

### Pocket

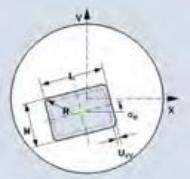
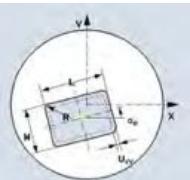
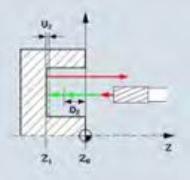
Notes

Parameter	Help picture	Description and hints
Perimeter C:		
UYZ	<p>Rectang.pocket</p> <p>T D1 F 0.000 mm/tooth V 0 m/min</p> <p>Per.surf.C</p> <p>Machining: Single position</p> <p>YB 0.000 abs ZB 0.000 abs XB 0.000 abs W 0.000 L 0.000 R 0.000 oB 0.000 ° X1 0.000 inc DY2 0.000 % DX 0.000 UV2 0.000 mm UX 0.000 Insertion: Centric FX 0.000 mm/min</p>	Finishing allowance in the plane (pocket wall) [mm]
UX	<p>Rectang.pocket</p> <p>T D1 F 0.000 mm/tooth V 0 m/min</p> <p>Per.surf.C</p> <p>Machining: Single position</p> <p>YB 0.000 abs ZB 0.000 abs XB 0.000 abs W 0.000 L 0.000 R 0.000 oB 0.000 ° X1 0.000 inc DY2 0.000 % DX 0.000 UV2 0.000 mm UX 0.000 Insertion: Centric FX 0.000 mm/min</p>	Finishing allowance at the bottom (bottom of pocket) Available only for "Roughing" and "Finishing".
Face Y:		
CP	<p>Rectang.pocket</p> <p>T D1 F 0.000 mm/tooth V 0 m/min</p> <p>Face Y</p> <p>Machining: Single position</p> <p>CP 0.000 ° XB 0.000 abs YB 0.000 abs ZB 0.000 abs W 0.000 L 0.000 R 0.000 oB 0.000 ° Z1 0.000 inc DXY 0.000 % DZ 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/min</p>	Positioning angle for machining range [Degrees]

## Section 3

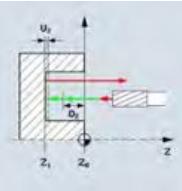
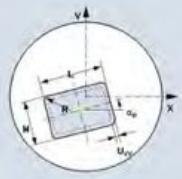
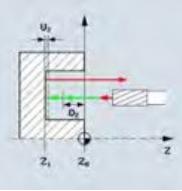
### Pocket

Notes

Parameter	Help picture	Description and hints
<b>Face Y:</b>		
X0, L0	 <b>Rectang.pocket</b> T D1 F 8.000 mm/tooth V 8 m/min Face Y Machining: Single position CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 uB 0.000 ° Z1 0.000 inc DXY 0.000 % D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 8.000 mm/min	Reference point in X-direction [abs] Alternatively: Reference point polar length
Y0, C0	 <b>Rectang.pocket</b> T D1 F 8.000 mm/tooth V 8 m/min Face Y Machining: Single position CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 uB 0.000 ° Z1 0.000 inc DXY 0.000 % D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 8.000 mm/min	Reference point in Y-direction [abs] Alternatively: Reference point polar angle [Degrees]
Z0	 <b>Rectang.pocket</b> T D1 F 8.000 mm/tooth V 8 m/min Face Y Machining: Single position CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 uB 0.000 ° Z1 0.000 inc DXY 0.000 % D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 8.000 mm/min	Reference point in Z-direction [abs]

## Section 3

### Pocket

Parameter	Help picture	Description and hints	Notes																																										
Z1	<p>Face Y:</p> 	<p>Depth referred to Z0 [abs, inc]</p>																																											
DXY	<p>Rectang.pocket</p> <table border="1"> <tr><td>T</td><td>D<sub>c</sub></td></tr> <tr><td>F</td><td>8.000 mm/tooth</td></tr> <tr><td>V</td><td>8 m/min</td></tr> <tr><td colspan="2">Face Y</td></tr> <tr><td>Machining:</td><td>▼</td></tr> <tr><td>Single position</td><td></td></tr> <tr><td>CP</td><td>0.000 °</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>W</td><td>8.000</td></tr> <tr><td>L</td><td>8.000</td></tr> <tr><td>R</td><td>8.000</td></tr> <tr><td>α<sub>B</sub></td><td>8.000 °</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> <tr><td>DXY</td><td>8.000 %</td></tr> <tr><td>D2</td><td>8.000</td></tr> <tr><td>UXY</td><td>8.000 mm</td></tr> <tr><td>UZ</td><td>8.000</td></tr> <tr><td>Insertion:</td><td>Centric</td></tr> <tr><td>FZ</td><td>8.000 mm/min</td></tr> </table> 	T	D <sub>c</sub>	F	8.000 mm/tooth	V	8 m/min	Face Y		Machining:	▼	Single position		CP	0.000 °	X0	0.000 abs	Y0	0.000 abs	Z0	0.000 abs	W	8.000	L	8.000	R	8.000	α <sub>B</sub>	8.000 °	Z1	0.000 inc	DXY	8.000 %	D2	8.000	UXY	8.000 mm	UZ	8.000	Insertion:	Centric	FZ	8.000 mm/min	<p>Maximum infeed in the XY-plane [mm]</p> <p>Alternatively:</p> <p>Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm)</p> <p>[%]</p> <p>Available only for "Roughing" and "Finishing".</p>	
T	D <sub>c</sub>																																												
F	8.000 mm/tooth																																												
V	8 m/min																																												
Face Y																																													
Machining:	▼																																												
Single position																																													
CP	0.000 °																																												
X0	0.000 abs																																												
Y0	0.000 abs																																												
Z0	0.000 abs																																												
W	8.000																																												
L	8.000																																												
R	8.000																																												
α <sub>B</sub>	8.000 °																																												
Z1	0.000 inc																																												
DXY	8.000 %																																												
D2	8.000																																												
UXY	8.000 mm																																												
UZ	8.000																																												
Insertion:	Centric																																												
FZ	8.000 mm/min																																												
DZ	<p>Rectang.pocket</p> <table border="1"> <tr><td>T</td><td>D<sub>c</sub></td></tr> <tr><td>F</td><td>8.000 mm/tooth</td></tr> <tr><td>V</td><td>8 m/min</td></tr> <tr><td colspan="2">Face Y</td></tr> <tr><td>Machining:</td><td>▼</td></tr> <tr><td>Single position</td><td></td></tr> <tr><td>CP</td><td>0.000 °</td></tr> <tr><td>X0</td><td>0.000 abs</td></tr> <tr><td>Y0</td><td>0.000 abs</td></tr> <tr><td>Z0</td><td>0.000 abs</td></tr> <tr><td>W</td><td>8.000</td></tr> <tr><td>L</td><td>8.000</td></tr> <tr><td>R</td><td>8.000</td></tr> <tr><td>α<sub>B</sub></td><td>8.000 °</td></tr> <tr><td>Z1</td><td>8.000 inc</td></tr> <tr><td>DXY</td><td>8.000 %</td></tr> <tr><td>D2</td><td>8.000</td></tr> <tr><td>UXY</td><td>8.000 mm</td></tr> <tr><td>UZ</td><td>8.000</td></tr> <tr><td>Insertion:</td><td>Centric</td></tr> <tr><td>FZ</td><td>8.000 mm/min</td></tr> </table> 	T	D <sub>c</sub>	F	8.000 mm/tooth	V	8 m/min	Face Y		Machining:	▼	Single position		CP	0.000 °	X0	0.000 abs	Y0	0.000 abs	Z0	0.000 abs	W	8.000	L	8.000	R	8.000	α <sub>B</sub>	8.000 °	Z1	8.000 inc	DXY	8.000 %	D2	8.000	UXY	8.000 mm	UZ	8.000	Insertion:	Centric	FZ	8.000 mm/min	<p>Max. Infeed depth - Z-direction [mm]</p>	
T	D <sub>c</sub>																																												
F	8.000 mm/tooth																																												
V	8 m/min																																												
Face Y																																													
Machining:	▼																																												
Single position																																													
CP	0.000 °																																												
X0	0.000 abs																																												
Y0	0.000 abs																																												
Z0	0.000 abs																																												
W	8.000																																												
L	8.000																																												
R	8.000																																												
α <sub>B</sub>	8.000 °																																												
Z1	8.000 inc																																												
DXY	8.000 %																																												
D2	8.000																																												
UXY	8.000 mm																																												
UZ	8.000																																												
Insertion:	Centric																																												
FZ	8.000 mm/min																																												

## Section 3

### Pocket

Notes

Parameter

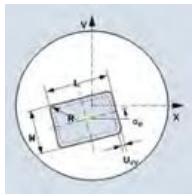
Help picture

Description and hints

Face Y:

UXY

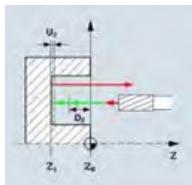
```
Rectang.pocket
T           D1
F     0.000 mm/tooth
V     0 m/min
Face Y
Machining:
Single position
CP   0.000 °
X0   0.000 abs
Y0   0.000 abs
Z0   0.000 abs
H    0.000
L    0.000
R    0.000
u0   0.000 °
Z1   0.000 inc
DXY  0.000 %
DZ   0.000
UXY  0.000 mm
UZ   0.000
Insertion: Centric
FZ   0.000 mm/min
```



Finishing allowance in the plane (pocket wall)  
[mm]

UZ

```
Rectang.pocket
T           D1
F     0.000 mm/tooth
V     0 m/min
Face Y
Machining:
Single position
CP   0.000 °
X0   0.000 abs
Y0   0.000 abs
Z0   0.000 abs
H    0.000
L    0.000
R    0.000
u0   0.000 °
Z1   0.000 inc
DXY  0.000 %
DZ   0.000
UXY  0.000 mm
UZ  0.000
Insertion: Centric
FZ   0.000 mm/min
```

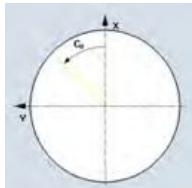


Finishing allowance at the bottom (bottom of pocket)  
Available only for "Roughing" and "Finishing".

Perimeter Y:

C0

```
Rectang.pocket
T           D1
F     0.000 mm/tooth
V     0 m/min
Per.surf.Y
Machining:
Single position
C0  0.000 °
Y0   0.000 abs
Z0   0.000 abs
X0   0.000 abs
H    0.000
L    0.000
R    0.000
u0   0.000 °
X1   0.000 inc
DY2  0.000 %
DX   0.000
UY2  0.000 mm
UX   0.000
Insertion: Centric
FX   0.000 mm/min
```



Reference point  
[Degrees]

## Section 3

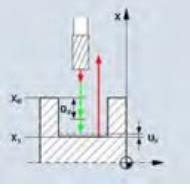
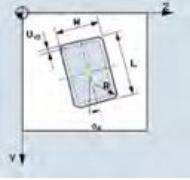
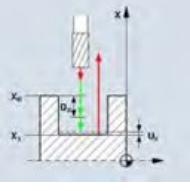
### Pocket

Parameter	Help picture	Description and hints	Notes
Perimeter Y:			
Y0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min Per.surf.V</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> <li>C0 0.000 °</li> <li>Y0 0.000 abs</li> <li>ZB 0.000 abs</li> <li>XB 0.000 abs</li> <li>W 0.000</li> <li>L 0.000</li> <li>R 0.000</li> <li>αB 0.000 °</li> <li>X1 0.000 inc</li> <li>DY2 0.000 %</li> <li>DX 0.000</li> <li>UV2 0.000 mm</li> <li>UX 0.000</li> </ul> <p>Insertion: Centric</p> <p>FX 8.000 mm/min</p>	Reference point in Y-direction [abs]	
Z0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min Per.surf.V</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> <li>C0 0.000 °</li> <li>Y0 0.000 abs</li> <li>ZB 0.000 abs</li> <li>XB 0.000 abs</li> <li>W 0.000</li> <li>L 0.000</li> <li>R 0.000</li> <li>αB 0.000 °</li> <li>X1 0.000 inc</li> <li>DY2 0.000 %</li> <li>DX 0.000</li> <li>UV2 0.000 mm</li> <li>UX 0.000</li> </ul> <p>Insertion: Centric</p> <p>FX 8.000 mm/min</p>	Reference point in Z-direction [abs]	
X0	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min Per.surf.V</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> <li>C0 0.000 °</li> <li>Y0 0.000 abs</li> <li>ZB 0.000 abs</li> <li>XB 0.000 abs</li> <li>W 0.000</li> <li>L 0.000</li> <li>R 0.000</li> <li>αB 0.000 °</li> <li>X1 0.000 inc</li> <li>DY2 0.000 %</li> <li>DX 0.000</li> <li>UV2 0.000 mm</li> <li>UX 0.000</li> </ul> <p>Insertion: Centric</p> <p>FX 8.000 mm/min</p>	Reference point in X-direction [abs]	

## Section 3

### Pocket

Notes

Parameter	Help picture	Description and hints
Perimeter Y:		
X1	 <b>Rectang.pocket</b> T      D1 F      0.000 mm/tooth V      0 m/min Per.surf.Y Machining: Single position C8      0.000 ° V8      0.000 abs Z8      0.000 abs X8      0.000 abs H      0.000 L      0.000 R      0.000 o8      0.000 ° X1      0.000 inc DY2      0.000 % DX      0.000 UY2      0.000 mm UX      0.000 Insertion: Centric FX      0.000 mm/min	Depth referred to X0 $\emptyset$ [abs, inc]
DYZ	 <b>Rectang.pocket</b> T      D1 F      0.000 mm/tooth V      0 m/min Per.surf.Y Machining: Single position C8      0.000 ° V8      0.000 abs Z8      0.000 abs X8      0.000 abs H      0.000 L      0.000 R      0.000 o8      0.000 ° X1      0.000 inc DY2      0.000 % DX      0.000 UY2      0.000 mm UX      0.000 Insertion: Centric FX      0.000 mm/min	Maximum infeed in the YZ-plane  Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm)  [%], [mm]  Available only for "Roughing" and "Finishing".
DX	 <b>Rectang.pocket</b> T      D1 F      0.000 mm/tooth V      0 m/min Per.surf.Y Machining: Single position C8      0.000 ° V8      0.000 abs Z8      0.000 abs X8      0.000 abs H      0.000 L      0.000 R      0.000 o8      0.000 ° X1      0.000 inc DY2      0.000 % DX      0.000 UY2      0.000 mm UX      0.000 Insertion: Centric FX      0.000 mm/min	Max. Infeed depth - X-direction [mm]

## Section 3

### Pocket

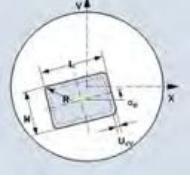
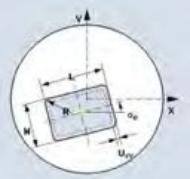
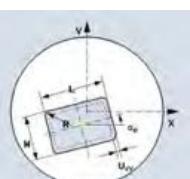
Notes

Parameter	Help picture	Description and hints																														
Perimeter Y:																																
UYZ	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Per.surf.Y</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <table border="1"> <tr><td>C8</td><td>0.000 °</td></tr> <tr><td>Y8</td><td>0.000 abs</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>X8</td><td>0.000 abs</td></tr> <tr><td>W</td><td>0.000</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>R</td><td>0.000</td></tr> <tr><td>aB</td><td>0.000 °</td></tr> <tr><td>X1</td><td>0.000 inc</td></tr> <tr><td>DY2</td><td>0.000 %</td></tr> <tr><td>DX</td><td>0.000</td></tr> <tr><td>UY2</td><td>0.000 mm</td></tr> <tr><td>UX</td><td>0.000</td></tr> <tr><td>Insertion:</td><td>Centric</td></tr> <tr><td>FX</td><td>8.000 mm/min</td></tr> </table>	C8	0.000 °	Y8	0.000 abs	Z8	0.000 abs	X8	0.000 abs	W	0.000	L	0.000	R	0.000	aB	0.000 °	X1	0.000 inc	DY2	0.000 %	DX	0.000	UY2	0.000 mm	UX	0.000	Insertion:	Centric	FX	8.000 mm/min	Finishing allowance in the plane (pocket wall) [mm]
C8	0.000 °																															
Y8	0.000 abs																															
Z8	0.000 abs																															
X8	0.000 abs																															
W	0.000																															
L	0.000																															
R	0.000																															
aB	0.000 °																															
X1	0.000 inc																															
DY2	0.000 %																															
DX	0.000																															
UY2	0.000 mm																															
UX	0.000																															
Insertion:	Centric																															
FX	8.000 mm/min																															
UX	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Per.surf.Y</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <table border="1"> <tr><td>C8</td><td>0.000 °</td></tr> <tr><td>Y8</td><td>0.000 abs</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>X8</td><td>0.000 abs</td></tr> <tr><td>W</td><td>0.000</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>R</td><td>0.000</td></tr> <tr><td>aB</td><td>0.000 °</td></tr> <tr><td>X1</td><td>0.000 inc</td></tr> <tr><td>DY2</td><td>0.000 %</td></tr> <tr><td>DX</td><td>0.000</td></tr> <tr><td>UY2</td><td>0.000 mm</td></tr> <tr><td>UX</td><td>0.000</td></tr> <tr><td>Insertion:</td><td>Centric</td></tr> <tr><td>FX</td><td>8.000 mm/min</td></tr> </table>	C8	0.000 °	Y8	0.000 abs	Z8	0.000 abs	X8	0.000 abs	W	0.000	L	0.000	R	0.000	aB	0.000 °	X1	0.000 inc	DY2	0.000 %	DX	0.000	UY2	0.000 mm	UX	0.000	Insertion:	Centric	FX	8.000 mm/min	Finishing allowance at the bottom (bottom of pocket) Available only for "Roughing" and "Finishing".
C8	0.000 °																															
Y8	0.000 abs																															
Z8	0.000 abs																															
X8	0.000 abs																															
W	0.000																															
L	0.000																															
R	0.000																															
aB	0.000 °																															
X1	0.000 inc																															
DY2	0.000 %																															
DX	0.000																															
UY2	0.000 mm																															
UX	0.000																															
Insertion:	Centric																															
FX	8.000 mm/min																															
All positions:																																
W	<p>Rectang.pocket</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <table border="1"> <tr><td>X8</td><td>0.000 abs</td></tr> <tr><td>Y8</td><td>0.000 abs</td></tr> <tr><td>Z8</td><td>0.000 abs</td></tr> <tr><td>W</td><td>0.000</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>R</td><td>0.000</td></tr> <tr><td>aB</td><td>0.000 °</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> <tr><td>DXY</td><td>0.000 %</td></tr> <tr><td>DZ</td><td>0.000</td></tr> <tr><td>UXY</td><td>0.000 mm</td></tr> <tr><td>UZ</td><td>0.000</td></tr> <tr><td>Insertion:</td><td>Centric</td></tr> <tr><td>FZ</td><td>8.000 mm/min</td></tr> </table>	X8	0.000 abs	Y8	0.000 abs	Z8	0.000 abs	W	0.000	L	0.000	R	0.000	aB	0.000 °	Z1	0.000 inc	DXY	0.000 %	DZ	0.000	UXY	0.000 mm	UZ	0.000	Insertion:	Centric	FZ	8.000 mm/min	Pocket width [mm]		
X8	0.000 abs																															
Y8	0.000 abs																															
Z8	0.000 abs																															
W	0.000																															
L	0.000																															
R	0.000																															
aB	0.000 °																															
Z1	0.000 inc																															
DXY	0.000 %																															
DZ	0.000																															
UXY	0.000 mm																															
UZ	0.000																															
Insertion:	Centric																															
FZ	8.000 mm/min																															

## Section 3

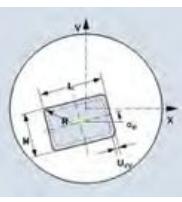
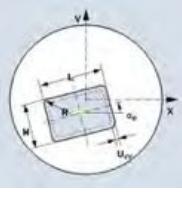
### Pocket

Notes

Parameter	Help picture	Description and hints
L	 <b>Rectang.pocket</b> <ul style="list-style-type: none"> <li>T D1</li> <li>F 8.000 mm/tooth</li> <li>V 8 m/min</li> <li>Face C</li> <li>Machining: Single position</li> <li>X0 0.000 abs</li> <li>Y0 0.000 abs</li> <li>Z0 0.000 abs</li> <li>H 0.000</li> <li>L <b>0.000</b></li> <li>R 0.000</li> <li>alpha_B 0.000 °</li> <li>Z1 0.000 inc</li> <li>DXY 0.000 %</li> <li>DZ 0.000</li> <li>UXY 0.000 mm</li> <li>UZ 0.000</li> <li>Insertion: Centric</li> <li>FZ 8.000 mm/min</li> </ul>	Pocket length [mm]
R	 <b>Rectang.pocket</b> <ul style="list-style-type: none"> <li>T D1</li> <li>F 8.000 mm/tooth</li> <li>V 8 m/min</li> <li>Face C</li> <li>Machining: Single position</li> <li>X0 0.000 abs</li> <li>Y0 0.000 abs</li> <li>Z0 0.000 abs</li> <li>H 0.000</li> <li>L 0.000</li> <li>R <b>0.000</b></li> <li>alpha_B 0.000 °</li> <li>Z1 0.000 inc</li> <li>DXY 0.000 %</li> <li>DZ 0.000</li> <li>UXY 0.000 mm</li> <li>UZ 0.000</li> <li>Insertion: Centric</li> <li>FZ 8.000 mm/min</li> </ul>	Corner radius [mm]
a0	 <b>Rectang.pocket</b> <ul style="list-style-type: none"> <li>T D1</li> <li>F 8.000 mm/tooth</li> <li>V 8 m/min</li> <li>Face C</li> <li>Machining: Single position</li> <li>X0 0.000 abs</li> <li>Y0 0.000 abs</li> <li>Z0 0.000 abs</li> <li>H 0.000</li> <li>L 0.000</li> <li>R 0.000</li> <li>alpha_B <b>0.000 °</b></li> <li>Z1 0.000 inc</li> <li>DXY 0.000 %</li> <li>DZ 0.000</li> <li>UXY 0.000 mm</li> <li>UZ 0.000</li> <li>Insertion: Centric</li> <li>FZ 8.000 mm/min</li> </ul>	Rotation angle [Degrees] End face: a0 refers to the X-axis, in case of polar reference point to the position C0 Perimeter: a0 refers to the Y-axis

## Section 3

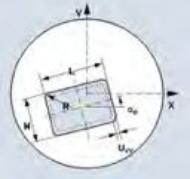
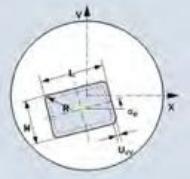
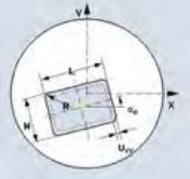
### Pocket

Parameter	Help picture	Description and hints	Notes
Plunging		<p>Centric (vertical plunging at pocket centre)</p> <p><b>Centric</b></p> <p>The evaluated infeed to depth is carried out vertically at the centre of the pocket.</p> <p><b>Hint:</b> For this selection either a slot drill must be used or else an entry hole must be pre-drilled</p> <p>Helical (plunging along a helical path)</p> <p><b>helical</b></p> <p>The centre of the milling cutter traverses along a helical path determined by the radius and the depth per revolution. Once the cutting depth for one infeed is attained a full circle will be carried out in order to even out the oblique surface due to the helical motion.</p> <p>Oscillating (plunging with an oscillatory motion along the centre axis of the pocket)</p> <p><b>oscillat</b></p> <p>The milling cutter centre oscillates along a straight until the depth of infeed is attained. Once the depth is attained the path is carried out once more without feed motion to even the oblique path due to the plunging.</p>	
FZ, FX		<p>FZ: Infeed feed rate at the bottom "Plunging centric" available only for "End face C" and „End face Y".</p> <p>FX: Infeed feed rate at the bottom "Plunging centric" available only for "Perimeter C" and "Perimeter Y" [mm/min, mm/tooth]</p>	
SINUMERIK 840D Training, Operation and Service		Page 19	A121

## Section 3

### Pocket

Notes

Parameter	Help picture	Description and hints
<b>EP</b> <pre>Rectang.pocket T           D1 F      0.000 mm/tooth V       8 m/min Face C Machining:  Single position X0      0.000 abs Y0      0.000 abs Z0      0.000 abs W      0.000 L      0.000 R      0.000 αθ     0.000 ° Z1      0.000 inc DXY    0.000 % DZ      0.000 UXY    0.000 mm UZ      0.000 Insertion: helical EP      0.000 mm/rev ER      0.000 mm</pre>		Maximum plunging pitch [mm/rev] Available only with "Plunging helical".
<b>ER</b> <pre>Rectang.pocket T           D1 F      0.000 mm/tooth V       8 m/min Face C Machining:  Single position X0      0.000 abs Y0      0.000 abs Z0      0.000 abs W      0.000 L      0.000 R      0.000 αθ     0.000 ° Z1      0.000 inc DXY    0.000 % DZ      0.000 UXY    0.000 mm UZ      0.000 Insertion: helical EP      0.000 mm/rev ER      0.000 mm</pre>		Plunging radius [mm] Available only with "Plunging helical".
<b>EW</b> <pre>Rectang.pocket T           D1 F      0.000 mm/tooth V       8 m/min Face C Machining:  Single position X0      0.000 abs Y0      0.000 abs Z0      0.000 abs W      0.000 L      0.000 R      0.000 αθ     0.000 ° Z1      0.000 inc DXY    0.000 % DZ      0.000 UXY    0.000 mm UZ      0.000 Insertion: oscillat EN      0.000 °</pre>		Plunging angle [Degrees] Only for plunging "Oscillatory".

## Section 3

### Pocket

#### 3.2 Circular pockets

For the machining of circular pockets the following variants are available:

Milling of circular pockets from the full material

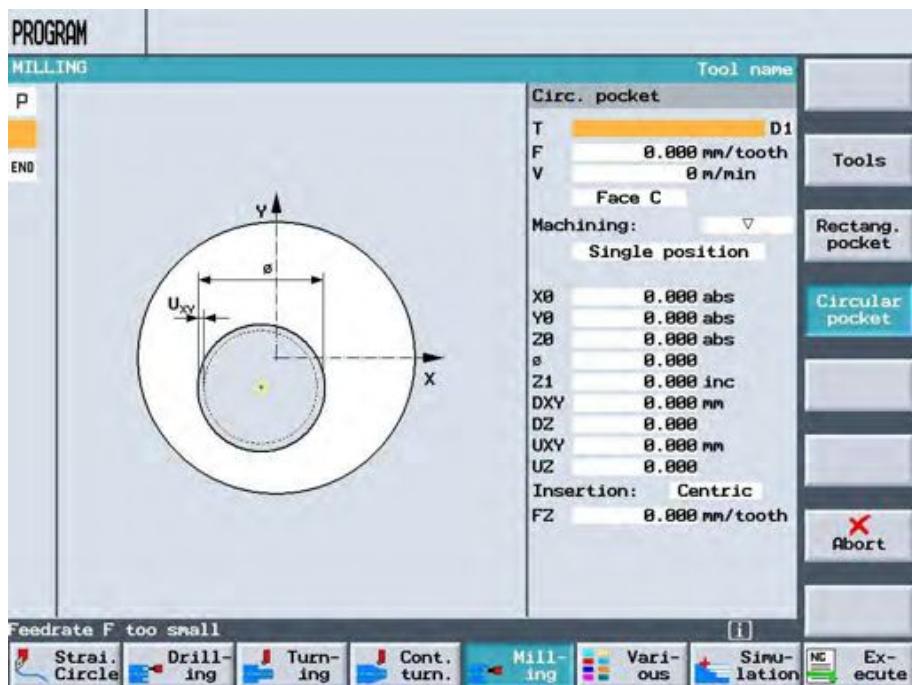
Circular pocket with centric pre-drilling, for instance if a slot drill cannot be used (programme the blocks drilling, rectangular pocket and position one after the other).

**Pocket**

By pressing the VSK 2 "Pocket" select the section for the programming of pockets.

**Circular pocket**

By pressing the VSK 4 "Circular pocket" select the section for the programming of circular pockets.



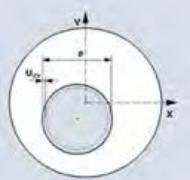
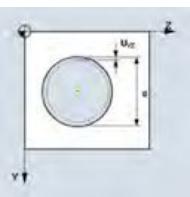
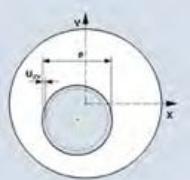
Approach/leaving the contour: See rectangular pocket

Notes

## Section 3

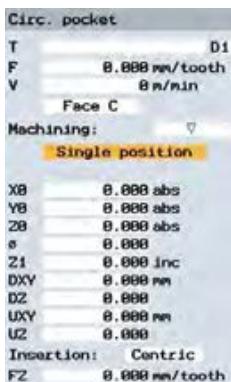
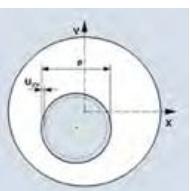
### Pocket

Notes

Parameter	Help picture	Description and hints	
T, D, F, S / V		See module A115 – "Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".	
Location	 	<b>Face C</b> <b>Face Y</b>	End face C End face Y
		<b>Per.surf.C</b> <b>Per.surf.Y</b>	Perimeter C Perimeter Y
Machining		<span style="color: orange;">▽</span> <span style="color: orange;">▽▽▽</span> <span style="color: orange;">▽▽▽ wall</span>	When roughing the individual planes of the pocket are machined from the centre outward until the depth Z1 is attained.  When finishing the edge is always machined first. For this the edge is approach with a quarter circle which leads into the corner radius. With the last infeed the bottom is also machined from the centre outward.  The finishing of the edge is similar to finishing, only the machining of the bottom with the last infeed is omitted.

## Section 3

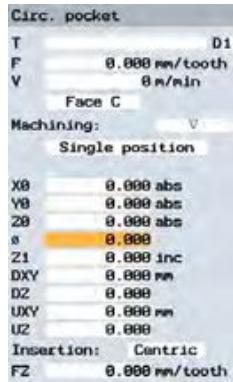
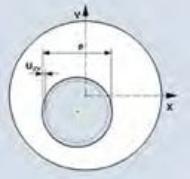
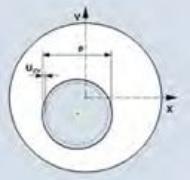
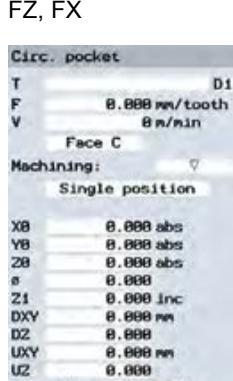
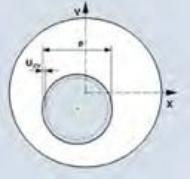
### Pocket

Parameter	Help picture	Description and hints	Notes
<p>Position</p> 		<p>Single position</p> <p><b>Single position</b></p> <p>A circular pocket is to be milled at the programmed position (<math>X_0, Y_0, Z_0</math>).</p> <p>Position pattern</p> <p><b>Position pattern</b></p> <p>Several circular pockets are to be milled on a position pattern (e.g. full circle, pitch circle, grid etc.).</p> <p>See Section 7 – "Position pattern when milling".</p>	
Face C:			
X0,L0; Y0,C0; Z0; X1;DXY; DZ; UXY; UZ		See Section 3.1 "Rectangular pockets"	
Perimeter C:			
Y0,C0; Z0; X0; X1; DYZ; DX; UYZ; UX		See Section 3.1 "Rectangular pockets"	
Face Y:			
CP; X0,L0; Y0,C0; Z0; Z1; DXY; DZ; UXY; UZ		See Section 3.1 "Rectangular pockets"	
Perimeter Y:			
C0; Y0; Z0; X0; X1; DYZ; DX; UYZ; UX		See Section 3.1 "Rectangular pockets"	

## Section 3

### Pocket

Notes

Parameter	Help picture	Description and hints
<b>Ø</b> 	<p>Diameter of the pocket [mm]</p> 	
<b>Plunging</b> 		<p>Centric (vertical plunging at pocket centre)</p> <p><b>Centric</b></p> <p>The evaluated infeed to depth is carried out vertically at the centre of the pocket.</p> <p><b>Hint:</b> For this selection either a slot drill must be used or else an entry hole must be pre-drilled</p> <p>Helical (plunging along a helical path)</p> <p><b>helical</b></p> <p>The centre of the milling cutter traverses along a helical path determined by the radius and the depth per revolution. Once the cutting depth for one infeed is attained a full circle will be carried out in order to even out the oblique surface due to the helical motion.</p>
<b>FZ, FX</b> 		<p>FZ: Infeed feed rate at the bottom Available only with "End face C" and "End face Y" - "Plunging centric".</p> <p>FX: Infeed feed rate at the bottom Available only with "Perimeter C" and "Perimeter Y" - "Plunging centric".</p> <p>[mm/min, mm/tooth]</p>

## Section 3

### Pocket

Parameter	Help picture	Description and hints	Notes
EP	<p><b>Circ. pocket</b></p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs ε 0.000 Z1 0.000 inc DX0 0.000 mm D2 0.000 UX0 0.000 mm U2 0.000</p> <p>Insertion: helical</p> <p>EP 8.000 mm/rev</p> <p>ER 0.000 mm</p>	<p>Maximum plunging pitch [mm/rev]</p> <p>Available only with "Plunging helical".</p>	
ER	<p><b>Circ. pocket</b></p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs ε 0.000 Z1 0.000 inc DX0 0.000 mm D2 0.000 UX0 0.000 mm U2 0.000</p> <p>Insertion: helical</p> <p>EP 8.000 mm/rev</p> <p>ER 0.000 mm</p>	<p>Plunging radius [mm]</p> <p>Available only with "Plunging helical".</p>	

## Section 4

### Spigot

Notes

ShopTurn provides the means for the milling of spigots in the section 'Milling'.

A differentiation is made between rectangular and circular spigots

#### 4.1 Rectangular spigot

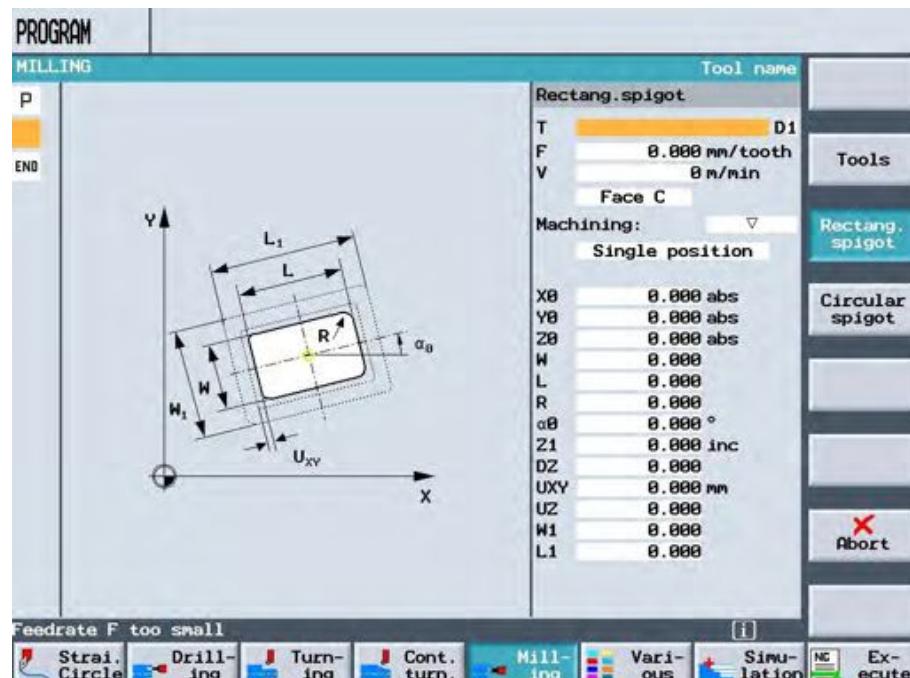
The following forms of rectangular spigots with and without corner radii can be milled:

Spigot

By pressing the VSK 3 "Spigot" select the section for the programming of spigot.

Rectang.  
spigot

By pressing the VSK 3 "Rectang. spigot" select the section for the programming of rectangular spigot.



## Section 4

### Spigot

#### Hint:

In addition to the rectangular spigot, a raw material spigot must be defined.

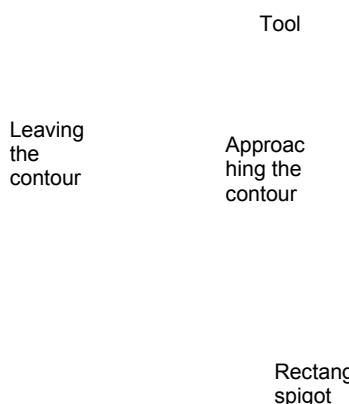
The raw material spigot defines the range outside which there is no more material, i.e. where motions will take place with rapid traverse. The raw material spigot must not overlap any adjoining raw material spigot and is arranged by Shop-Mill automatically concentrically about the finished part spigot.

The spigot is machined with only one infeed. If the machining is to be carried out with several infeeds, the function "Rectangular spigot" must be programmed with ever decreasing finishing allowances.

#### Notes

#### Approach/leaving the contour:

1. The tool traverses at the level of the retraction plane to the Start point and then feeds to the safety distance. The Start point lies on the positive X-axis which has been rotated by  $0^\circ$ .
2. The spigot contour is approached with the tool from the side in a semicircle with machining feed rate. Firstly there is the infeed to the machining depth followed by the motion in the plane. The spigot is machined, depending on the programmed machining direction (conventional/climb milling) and the cutter rotation (clockwise/anticlockwise), either in the clockwise or the anti-clockwise direction.
3. Once the spigot has been machined all round, the tool leaves the contour with a semicircle and then advanced to next depth,
4. The spigot is again approached with a semicircle and machined all round. This process is repeated until the programmed spigot depth has been attained.
5. The tool is retracted to the safety depth.



## Section 4

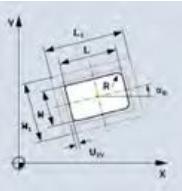
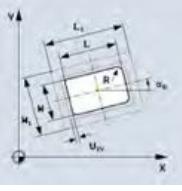
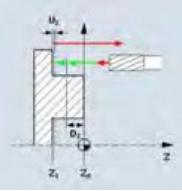
### Spigot

Notes

Parameter	Help picture	Description and hints	
T, D, F, S / V		See module A115 – "Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".	
Location		<b>Face C</b> <b>Face Y</b>	Face C Face Y
Machining		<b>▽</b> <b>▽▽▽</b>	For stock removal the individual planes will be machined one after the other from the outside diameter inwards until the depth Z1 is attained.  The wall of the spigot will always be finished first of all. For this the edge will be approached with a semi-circle which ends at the end of a corner radius. Together with the last infeed the bottom will be finished from the outside inward.
Position		Single position <b>Single position</b> A rectangular spigot is milled at the programmed position (X0, Y0, Z0).  Position pattern <b>Position pattern</b> Several rectangular spigot are milled on a position pattern (e.g. full circle, pitch circle, grid etc.).  See Section 7 – "Position pattern when milling".	

## Section 4

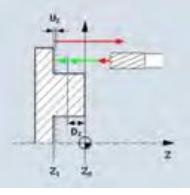
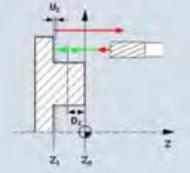
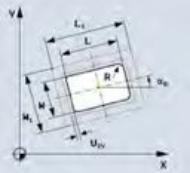
### Spigot

Parameter	Help picture	Description and hints	Notes
X0, L0	<p>Face C:</p> 	<p>Reference point in X-direction [abs]</p> <p>Alternatively: Reference point polar length</p>	
Y0, C0		<p>Reference point in Y-direction [abs]</p> <p>Alternatively: Reference point polar angle [Degrees]</p>	
Z0		<p>Reference point in Z-direction [abs]</p>	

## Section 4

### Spigot

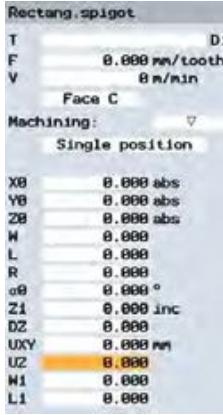
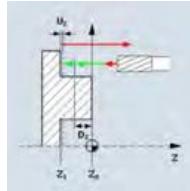
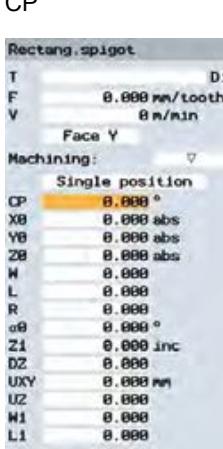
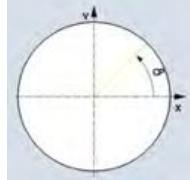
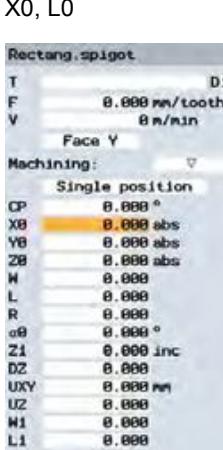
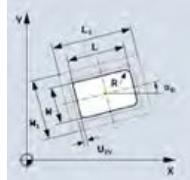
Notes

Parameter	Help picture	Description and hints
<b>Face C:</b>		
Z1	 <b>Rectang.spigot</b> T      D1 F    0.000 mm/tooth V    0 m/min Face C Machining:      ✓ Single position X0    0.000 abs Y0    0.000 abs Z0    0.000 abs H    0.000 L    0.000 R    0.000 aθ    0.000 ° Z1    0.000 inc DZ    0.000 UXY    0.000 mm UZ    0.000 W1    0.000 L1    0.000	Depth referred to Z0 [abs, inc]
DZ	 <b>Rectang.spigot</b> T      D1 F    0.000 mm/tooth V    0 m/min Face C Machining:      ✓ Single position X0    0.000 abs Y0    0.000 abs Z0    0.000 abs H    0.000 L    0.000 R    0.000 aθ    0.000 ° Z1    0.000 inc DZ    0.000 UXY    0.000 mm UZ    0.000 W1    0.000 L1    0.000	Max. Infeed depth - Z-direction [mm]
UXY	 <b>Rectang.spigot</b> T      D1 F    0.000 mm/tooth V    0 m/min Face C Machining:      ✓ Single position X0    0.000 abs Y0    0.000 abs Z0    0.000 abs H    0.000 L    0.000 R    0.000 aθ    0.000 ° Z1    0.000 inc DZ    0.000 UXY    0.000 mm UZ    0.000 W1    0.000 L1    0.000	Finishing allowance in the plane (spigot wall)

## Section 4

### Spigot

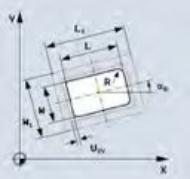
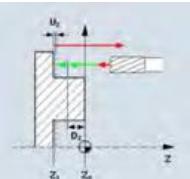
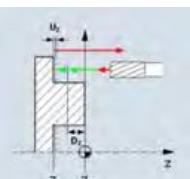
Notes

Parameter	Help picture	Description and hints
<b>Face C:</b> UZ 		Finishing allowance at the bottom (spigot bottom)
<b>Face Y:</b> CP 		Positioning angle for machining range
X0, L0 		X0: Reference point in X-direction [abs] Alternatively: L0: Reference point polar length

## Section 4

### Spigot

Notes

Parameter	Help picture	Description and hints
<b>Face Y:</b>		
<b>Y0, C0</b>		<b>Y0:</b> Reference point in Y-direction [abs] <b>C0:</b> Reference point polar angle [Degrees]
<b>Z0</b>		Reference point in Z-direction [abs]
<b>Z1</b>		Depth referred to Z0 [abs, inc]

## Section 4

### Spigot

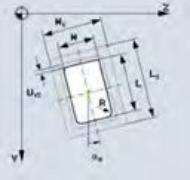
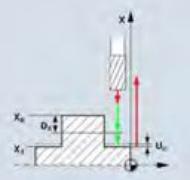
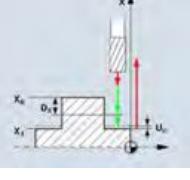
Notes

Parameter	Help picture	Description and hints
<b>Face Y:</b>		
DZ	<p>Rectang.spigot</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face Y</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <p>CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 αθ 0.000 ° Z1 0.000 inc D2 0.000 UXY 0.000 mm UZ 0.000 W1 0.000 L1 0.000</p>	<p>Max. Infeed depth - Z-direction [mm]</p>
UXY, UZ	<p>Rectang.spigot</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face Y</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <p>CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 αθ 0.000 ° Z1 0.000 inc D2 0.000 UXY 0.000 mm UZ 0.000 W1 0.000 L1 0.000</p>	<p>UXY: Finishing allowance in the plane (spigot wall)</p> <p>UZ: Finishing allowance at the bottom (spigot bottom)</p>
<b>Perimeter Y:</b>		
C0	<p>Rectang.spigot</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Per.surf.Y</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <p>CB 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs W 0.000 L 0.000 R 0.000 αθ 0.000 ° X1 0.000 inc DX 0.000 UYZ 0.000 mm UX 0.000 W1 0.000 L1 0.000</p>	<p>Reference point [Degrees]</p>

## Section 4

### Spigot

Notes

Parameter	Help picture	Description and hints
<b>Perimeter Y:</b>		
Y0, Z0, X0  Rectang.spigot T D1 F 8.000 mm/tooth V 8 m/min Per.surf.Y Machining: Single position OB 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs H 0.000 L 0.000 R 0.000 aθ 0.000 ° X1 0.000 inc DX 0.000 UY2 0.000 mm UX 0.000 W1 0.000 L1 0.000		Y0: Reference point in Y-direction Z0: Reference point in Z-direction X0: Reference point in X-direction [abs]
X1  Rectang.spigot T D1 F 8.000 mm/tooth V 8 m/min Per.surf.Y Machining: Single position OB 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs H 0.000 L 0.000 R 0.000 aθ 0.000 ° X1 0.000 inc DX 0.000 UY2 0.000 mm UX 0.000 W1 0.000 L1 0.000		Depth referred to X0 [abs, inc]
DX  Rectang.spigot T D1 F 8.000 mm/tooth V 8 m/min Per.surf.Y Machining: Single position OB 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs H 0.000 L 0.000 R 0.000 aθ 0.000 ° X1 0.000 inc DX 0.000 UY2 0.000 mm UX 0.000 W1 0.000 L1 0.000		Max. Infeed depth - X-direction [mm]

## Section 4

### Spigot

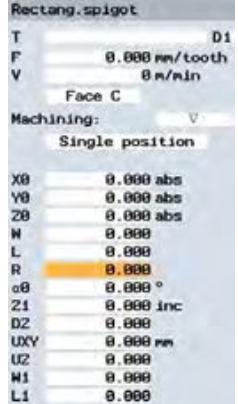
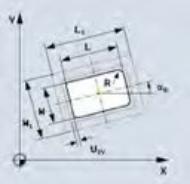
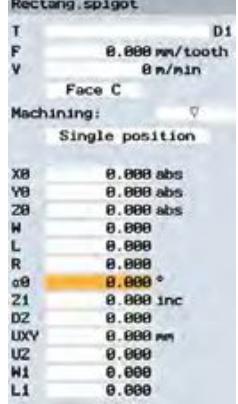
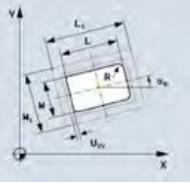
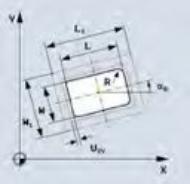
Notes

Parameter	Help picture	Description and hints																												
Perimeter Y:																														
UYZ, UX	<p>Rectang.spigot</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Par.surf.Y</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <table border="1"> <tr><td>CB</td><td>0.000 °</td></tr> <tr><td>YB</td><td>0.000 abs</td></tr> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>XB</td><td>0.000 abs</td></tr> <tr><td>W</td><td>0.000</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>R</td><td>0.000</td></tr> <tr><td>oB</td><td>0.000 °</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> <tr><td>D1</td><td>0.000</td></tr> <tr><td>UYZ</td><td>0.000 mm</td></tr> <tr><td>UX</td><td>0.000</td></tr> <tr><td>W1</td><td>0.000</td></tr> <tr><td>L1</td><td>0.000</td></tr> </table>	CB	0.000 °	YB	0.000 abs	ZB	0.000 abs	XB	0.000 abs	W	0.000	L	0.000	R	0.000	oB	0.000 °	Z1	0.000 inc	D1	0.000	UYZ	0.000 mm	UX	0.000	W1	0.000	L1	0.000	<p>UYZ: Finishing allowance in the plane (spigot wall)</p> <p>UX: Finishing allowance at the bottom (spigot bottom)</p>
CB	0.000 °																													
YB	0.000 abs																													
ZB	0.000 abs																													
XB	0.000 abs																													
W	0.000																													
L	0.000																													
R	0.000																													
oB	0.000 °																													
Z1	0.000 inc																													
D1	0.000																													
UYZ	0.000 mm																													
UX	0.000																													
W1	0.000																													
L1	0.000																													
All positions:																														
W	<p>Rectang.spigot</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <table border="1"> <tr><td>XB</td><td>0.000 abs</td></tr> <tr><td>YB</td><td>0.000 abs</td></tr> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>W</td><td>0.000</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>R</td><td>0.000</td></tr> <tr><td>oB</td><td>0.000 °</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> <tr><td>D2</td><td>0.000</td></tr> <tr><td>UXV</td><td>0.000 mm</td></tr> <tr><td>UZ</td><td>0.000</td></tr> <tr><td>W1</td><td>0.000</td></tr> <tr><td>L1</td><td>0.000</td></tr> </table>	XB	0.000 abs	YB	0.000 abs	ZB	0.000 abs	W	0.000	L	0.000	R	0.000	oB	0.000 °	Z1	0.000 inc	D2	0.000	UXV	0.000 mm	UZ	0.000	W1	0.000	L1	0.000	<p>Width of the spigot [mm]</p>		
XB	0.000 abs																													
YB	0.000 abs																													
ZB	0.000 abs																													
W	0.000																													
L	0.000																													
R	0.000																													
oB	0.000 °																													
Z1	0.000 inc																													
D2	0.000																													
UXV	0.000 mm																													
UZ	0.000																													
W1	0.000																													
L1	0.000																													
L	<p>Rectang.spigot</p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C</p> <p>Machining:</p> <ul style="list-style-type: none"> <li>Single position</li> </ul> <table border="1"> <tr><td>XB</td><td>0.000 abs</td></tr> <tr><td>YB</td><td>0.000 abs</td></tr> <tr><td>ZB</td><td>0.000 abs</td></tr> <tr><td>W</td><td>0.000</td></tr> <tr><td>L</td><td>0.000</td></tr> <tr><td>R</td><td>0.000</td></tr> <tr><td>oB</td><td>0.000 °</td></tr> <tr><td>Z1</td><td>0.000 inc</td></tr> <tr><td>D2</td><td>0.000</td></tr> <tr><td>UXV</td><td>0.000 mm</td></tr> <tr><td>UZ</td><td>0.000</td></tr> <tr><td>W1</td><td>0.000</td></tr> <tr><td>L1</td><td>0.000</td></tr> </table>	XB	0.000 abs	YB	0.000 abs	ZB	0.000 abs	W	0.000	L	0.000	R	0.000	oB	0.000 °	Z1	0.000 inc	D2	0.000	UXV	0.000 mm	UZ	0.000	W1	0.000	L1	0.000	<p>Length of the spigot [mm]</p>		
XB	0.000 abs																													
YB	0.000 abs																													
ZB	0.000 abs																													
W	0.000																													
L	0.000																													
R	0.000																													
oB	0.000 °																													
Z1	0.000 inc																													
D2	0.000																													
UXV	0.000 mm																													
UZ	0.000																													
W1	0.000																													
L1	0.000																													

## Section 4

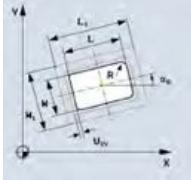
### Spigot

Notes

Parameter	Help picture	Description and hints
All positions:		
R	 	Corner radius [mm]
a0	 	Rotation angle [Degrees] End face: a0 refers to the X-axis, in case of polar reference to the position of C0 Perimeter: a0 refers to the Y-axis
W1	 	Width of the raw material spigot [mm] This parameter is important for the determination of the approach position.

## Section 4

### Spigot

Parameter	Help picture	Description and hints	Notes
All positions:			
L1  Rectang_spigot T D1 F 8.000 mm/tooth V 8 m/min Face C Machining: Single position  X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 R 0.000 aθ 0.000 ° Z1 0.000 JNC DZ 0.000 UXY 0.000 mm UZ 0.000 W1 0.000 L1 0.000 L1 0.000		Length of the raw material spigot [mm]  This parameter is important for the determination of the approach position.	

## Section 4

Notes

Spigot

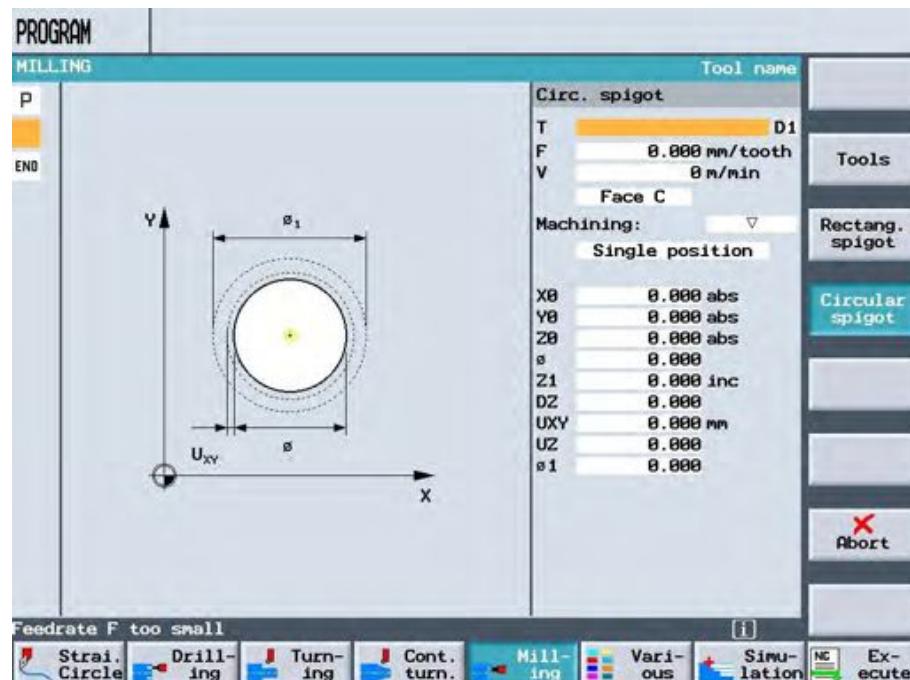
### 4.2 Circular spigot

Spigot

By pressing the VSK 3 "Spigot" select the section for the programming of spigot.

Circular  
spigot

By pressing the VSK 4 "circular spigot" select the section for the programming of circular spigot.



**Hint:**

The hints in Section 4.1 "Rectangular spigot" must be observed.

**Approach/leaving the contour:**

The approach and leaving strategy for circular spigot-milling is similar to the strategy of rectangular spigot-milling. See Section 4.1 "Rectangular spigot".

Tool

Leaving  
the  
contour

Approach  
of contour

Circular spigot

## Section 4

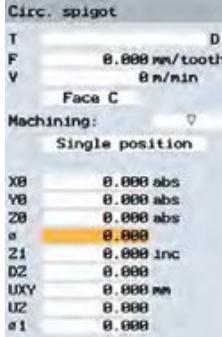
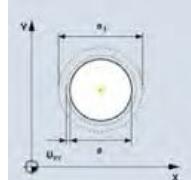
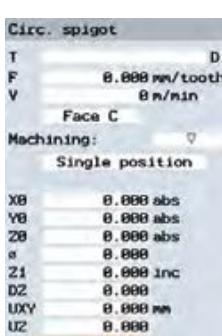
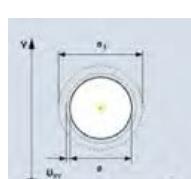
### Spigot

Parameter	Help picture	Description and hints		Notes
T, D, F, S / V		See module A115 – "Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".		
Location  <b>Circ. spigot</b> T D1 F 8.000 mm/tooth V 8 m/min Face C Machining: Single position  X0 8.000 abs Y0 8.000 abs Z0 8.000 abs σ 8.000 Z1 8.000 inc D2 8.000 UXY 8.000 mm UZ 8.000 σ1 8.000		<b>Face C</b>	Face C	
		<b>Face Y</b>	Face Y	
Machining  <b>Circ. spigot</b> T D1 F 8.000 mm/tooth V 8 m/min Face C Machining: Single position  X0 8.000 abs Y0 8.000 abs Z0 8.000 abs σ 8.000 Z1 8.000 inc D2 8.000 UXY 8.000 mm UZ 8.000 σ1 8.000		<b>Per. surf. Y</b>	Perimeter Y	
Position  <b>Circ. spigot</b> T D1 F 8.000 mm/tooth V 8 m/min Face C Machining: Single position  X0 8.000 abs Y0 8.000 abs Z0 8.000 abs σ 8.000 Z1 8.000 inc D2 8.000 UXY 8.000 mm UZ 8.000 σ1 8.000		<b>▽</b>	For stock removal the individual planes will be machined one after the other from the outside diameter inwards until the depth Z1 is attained.	
		<b>▽▽▽</b>	The wall of the spigot will always be finished first of all. For this the edge will be approached with a semi-circle which ends at the end of a corner radius. Together with the last infeed the bottom will be finished from the outside inward.	
Position  <b>Circ. spigot</b> T D1 F 8.000 mm/tooth V 8 m/min Face C Machining: Single position  X0 8.000 abs Y0 8.000 abs Z0 8.000 abs σ 8.000 Z1 8.000 inc D2 8.000 UXY 8.000 mm UZ 8.000 σ1 8.000		<p>Single position  <b>Single position</b> A circular spigot is milled at the programmed position (X0, Y0, Z0).</p> <p>Position pattern  <b>Position pattern</b> Several circular spigots are milled on a position pattern (e.g. full circle, pitch circle, grid etc.).</p> <p>See Section 7 – "Position pattern when milling".</p>		

## Section 4

### Spigot

Notes

Parameter	Help picture	Description and hints
Face C: X0,L0; Y0,C0; Z0; Z1; DZ; UXY; UZ		See Section 4.1 "Rectangular spigot"
Face Y: CP; X0,L0; Y0,C0; Z0; Z1; DZ; UXY; UZ		See Section 4.1 "Rectangular spigot"
Perimeter Y: C0; Y0; Z0; X0; X1; DX; UYZ; UX		See Section 4.1 "Rectangular spigot"
All positions:  <b>Ø</b> 		Diameter of the spigots [mm]
<b>Ø1</b> 		Diameter of the raw material spigot [mm]  This parameter is important for the determination of the approach position.

ShopTurn provides in the sector milling the means of milling slots. A differentiation is made between longitudinal and circular slots.

Notes

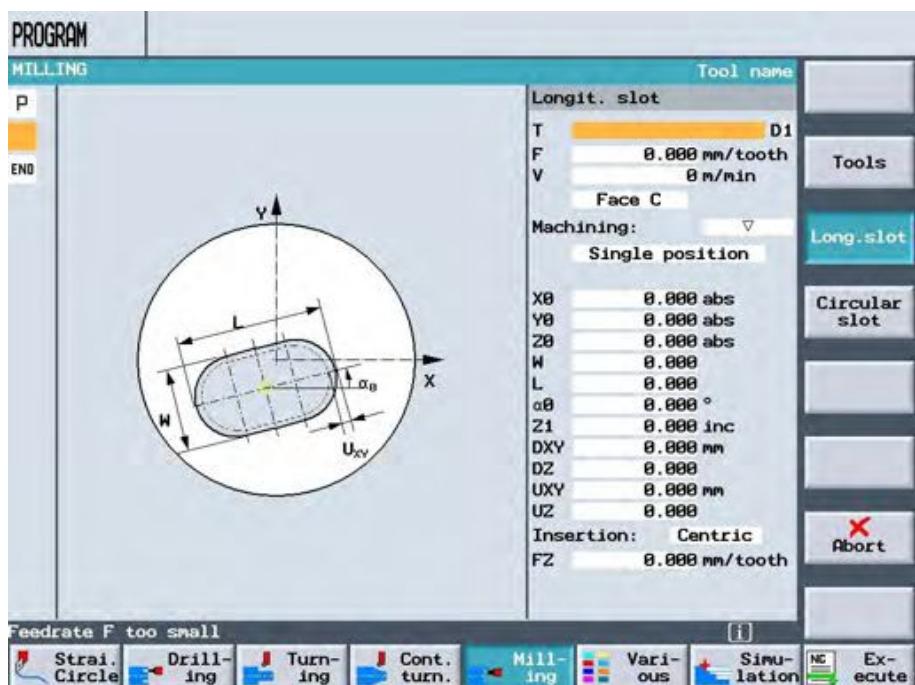
#### 5.1 Longitudinal slot

Slot

By pressing the VSK 4 "Slot" select the section for the programming of slots.

Long.slot

By pressing the VSK 3 "Longitudinal slot" select the section for the programming of longitudinal slots.



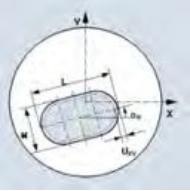
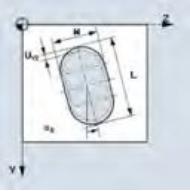
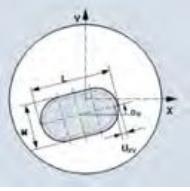
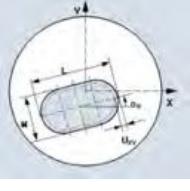
##### Approach/leaving the contour:

1. The tool traverses with rapid traverse at the level of the retraction plane and then feeds to the safety distance.
2. The tool plunges into the material in a manner depending on the chosen strategy.
3. The machining of the longitudinal slot ensues with the chosen type of machining always from the inside outward.
4. The tool is retracted with rapid traverse to the safety distance.

## Section 5

Notes

Slot

Parameter	Help picture	Description and hints	
T, D, F, S / V		<i>See module A115 – "Operating Area Program – Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".</i>	
Location	 	<b>Face C</b> <b>Face Y</b>	Face C Face Y
Machining		<b>Per.surf.C</b> <b>Per.surf.Y</b>	Perimeter C Perimeter Y
Position		When roughing the individual planes of the slot are machined from the centre outward until the depth Z1 is attained.  When finishing the edge is always machined first. For this the edge is approach with a quarter circle which leads into the corner radius. With the last infeed the bottom is also machined from the centre outward.  The finishing of the edge is similar to finishing, only the machining of the bottom with the last infeed is omitted.	

## Section 5

### Slot

Notes

Face C:

X0,L0; Y0,C0; Z0;  
Z1;DXY; DZ; UXY;  
UZ

*See Section 3.1 "Rectangular pockets"*

Perimeter C:

Y0,C0; Z0; X0; X1;  
DYZ; DX; UYZ; UX

*See Section 3.1 "Rectangular pockets"*

Face Y:

CP; X0,L0; Y0,C0;  
Z0; Z1; DXY; DZ;  
UXY; UZ

*See Section 3.1 "Rectangular pockets"*

Perimeter Y:

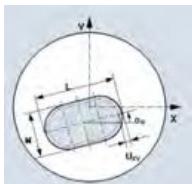
C0; Y0; Z0; X0; X1;  
DYZ; DX; UYZ; UX

*See Section 3.1 "Rectangular pockets"*

All positions:

W

```
Longit. slot
T          D1
F      0.000 mm/tooth
V      0 m/min
Face C
Machining: V
Single position
X0      0.000 abs
Y0      0.000 abs
Z0      0.000 abs
W      0.000
L      0.000
aB      0.000 °
Z1      0.000 inc
DXY      0.000 mm
D2      0.000
UXV      0.000 mm
UZ      0.000
Insertion: Centric
FZ      0.000 mm/tooth
```



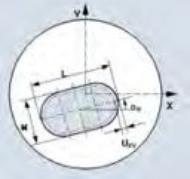
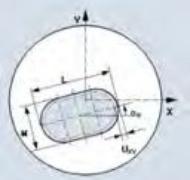
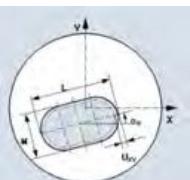
Width of the slot

[mm]

## Section 5

Notes

Slot

Parameter	Help picture	Description and hints
L  Longit. slot T D1 F 0.000 mm/tooth V 8 m/min Face C Machining: ▼ Single position  X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 α0 0.000 ° Z1 0.000 inc DXY 0.000 mm D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/tooth		Length of the slot [mm]
0  Longit. slot T D1 F 0.000 mm/tooth V 8 m/min Face C Machining: ▼ Single position  X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 α0 0.000 ° Z1 0.000 inc DXY 0.000 mm D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/tooth		Rotation angle [Degrees]  End face: α0 refers to the X-axis, in case of polar reference to the position of C0  Perimeter: α0 refers to the Y-axis
Plunging  Longit. slot T D1 F 0.000 mm/tooth V 8 m/min Face C Machining: ▼ Single position  X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 L 0.000 α0 0.000 ° Z1 0.000 inc DXY 0.000 mm D2 0.000 UXY 0.000 mm UZ 0.000 Insertion: Centric FZ 0.000 mm/tooth		Centric (vertical plunging at centre of the longitudinal slot):  <b>Centric</b> The infeed to depth is carried out vertically at the centre of the slot.  Hint: For this selection either a slot drill must be used.  Oscillating - Oscillating along the centre line of the longitudinal slot:  <b>oscillat</b> The milling cutter centre oscillates along a straight until the depth of infeed is attained. Once the depth is attained the path is carried out once more without feed motion to even the oblique path due to the plunging.

## Section 5

### Slot

Parameter	Help picture	Description and hints	Notes
FZ	<pre>Longit. slot T           D1 F       0.000 mm/tooth V       0 m/min Face C Machining: V Single position X0      0.000 abs Y0      0.000 abs Z0      0.000 abs W      0.000 L      0.000 aB      0.000 ° Z1      0.000 Jnc DXV     0.000 mm DZ      0.000 UXY     0.000 mm UZ      0.000 Insertion: Centric FZ      0.000 mm/tooth</pre>	<p>FZ: Infeed feed rate depth Available only with "End face C" and "End face Y" - "Plunging centric".</p> <p>FX: Infeed feed rate depth Available only with "Perimeter C" and "Perimeter Y" - "Plunging centric".</p> <p>[mm/min, mm/tooth]</p>	
EW	<pre>Longit. slot T           D1 F       0.000 mm/tooth V       0 m/min Face C Machining: V Single position X0      0.000 abs Y0      0.000 abs Z0      0.000 abs W      0.000 L      0.000 aB      0.000 ° Z1      0.000 Jnc DXV     0.000 mm DZ      0.000 UXY     0.000 mm UZ      0.000 Insertion: oscillat EW      0.000 °</pre>	<p>Plunging angle [Degrees]</p> <p>Available only with "Oscillating".</p>	

## Section 5

Notes

Slot

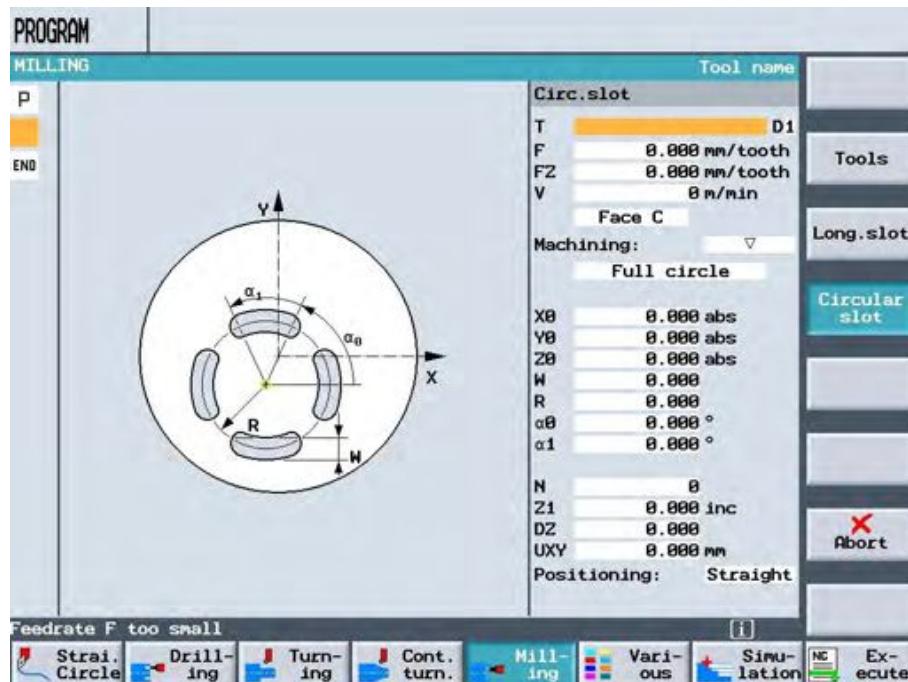
### 5.2 Circular slot

Slot

By pressing the VSK 4 "Slot" select the section for the programming of slots.

Circular slot

By pressing the VSK 4 "Circular slot" select the section for the programming of circular slots.



If one or more identical circular slots are to be milled on a full or a pitch circle, the function "Circular slot" can be used. By input of the parameter  $\alpha_1$  the curvature of each slot can be specified.

**Approach/leaving the contour:**

1. The tool moves with rapid traverse to the height of the retraction plane to the centre point of the semi-circle at the end of the slot and then feeds to the safety distance.
2. After that the tool enters the work piece with the machining feed rate. For this the max. infeed in the Z-direction (for end face machining) and in the X-direction (for perimeter machining) as well as the finishing allowance can be taken into account. The circular slot can be machined either clockwise or anti-clockwise depending on the sense of rotation (conventional or climb milling).
3. When the first circular slot is machined the tool is retracted with rapid traverse to the retraction plane.

## Section 5

### Slot

Notes

4. The next circular slot is approached with a straight or a circular path and then machined.
5. The tool is retracted with rapid traverse to the safety distance level.

#### Hints:

##### Tool size:

When circular slots are being machined the milling cutter must not be smaller than a minimum size:

##### Roughing:

$\frac{1}{2}$  slot width W - finishing allowance UXY  $\leq$  cutter diameter

##### Finishing:

$\frac{1}{2}$  slot width W  $\leq$  cutter diameter

##### Finishing edge:

Finishing allowance UXY  $\leq$  cutter diameter

##### Circular slot:

For the machining of a circular slot the following values must be entered for the parameter.

Number N = 1

Opening angle = 360 °

Parameter	Help picture	Description and hints
T, D, F, S / V		See module A115 – "Operation sector Program – Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".
Location	 	<p><b>Face C</b> Face C</p> <p><b>Face Y</b> Face Y</p> <p><b>Per.surf.C</b> Perimeter C</p> <p><b>Per.surf.Y</b> Perimeter Y</p>

## Section 5

Notes

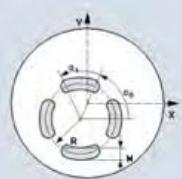
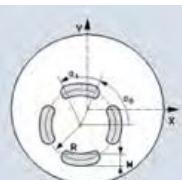
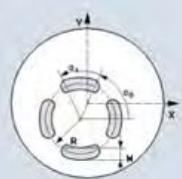
Slot

Parameter	Help picture	Description and hints
Machining  		Roughing Finishing The individual planes are roughed out one after the other starting from the centre of the semi-circle at the end of the slot until the depth Z1 is reached. For finishing, the wall of the slot is always machined first until the depth Z1 is reached. The wall in this case is approached with a quarter circle which meets the wall in the end radius of the slot. With the last infeed the bottom is also finished starting from the centre point of the semi-circle at the
		Finishing wall wall The "Finishing wall" is the same as the normal finishing, only the last infeed (finishing bottom) is omitted.
Voll-/Pitch circle  		Full circle Full circle The circular slots are positioned on a full circle . The distance between one circular slot to the next circular slot is always the same and is evaluated by the control unit.
		Pitch circle Pitchcircle The circular slots are positioned on a pitch circle. The distance between one circular slot to the next circular slot can be determined by means of the angle 2.
Face C:		
X0,L0; Y0,C0; Z0; Z1; DZ; UXY		See Section 3.1 "Rectangular pockets"
Perimeter C:		
Y0,C0; Z0; X0; X1; DX; UYZ		See Section 3.1 "Rectangular pockets"
Face Y:		
CP; X0,L0; Y0,C0; Z0; Z1; DZ; UXY		See Section 3.1 "Rectangular pockets"

## Section 5

### Slot

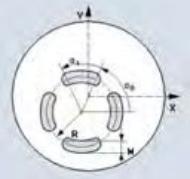
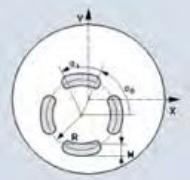
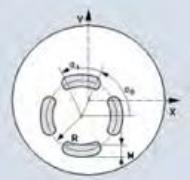
Notes

Parameter	Help picture	Description and hints
Perimeter Y:		
C0; Y0; Z0; X0; X1; DX; UYZ		See Section 3.1 "Rectangular pockets"
All positions:		
W  <b>Circ.slot</b> T D1 F 0.000 mm/tooth FZ 0.000 mm/tooth V 0 m/min Face C Machining: V Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 R 0.000 α0 0.000 ° α1 0.000 ° N 0 Z1 0.000 inc D2 0.000 UXY 0.000 mm Positioning: Straight	 Width of the slot [mm]	
R  <b>Circ.slot</b> T D1 F 0.000 mm/tooth FZ 0.000 mm/tooth V 0 m/min Face C Machining: V Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 R 0.000 α0 0.000 ° α1 0.000 ° N 0 Z1 0.000 inc D2 0.000 UXY 0.000 mm Positioning: Straight	 Radius of the circular slot [mm]	
0  <b>Circ.slot</b> T D1 F 0.000 mm/tooth FZ 0.000 mm/tooth V 0 m/min Face C Machining: V Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 R 0.000 α0 0.000 ° α1 0.000 ° N 0 Z1 0.000 inc D2 0.000 UXY 0.000 mm Positioning: Straight	 Rotation angle [Degrees]  Hint: α0 is referred to the X-axis in case of the end face and to the Y-axis for the perimeter face	

## Section 5

Notes

Slot

Parameter	Help picture	Description and hints
1  <b>Circ.slot</b> T D1 F 0.000 mm/tooth FZ 0.000 mm/tooth V 0 m/min Face C Machining: V Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 R 0.000 a0 0.000 ° a1 0.000 ° N 0 Z1 0.000 inc D2 0.000 UXY 0.000 mm Positioning: Straight		Opening angle of the slot [Degrees]
2  <b>Circ.slot</b> T D1 F 0.000 mm/tooth FZ 0.000 mm/tooth V 0 m/min Face C Machining: ▽ Pitchcircle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 R 0.000 a0 0.000 ° a1 0.000 ° a2 0.000 ° N 0 Z1 0.000 inc D2 0.000 UXY 0.000 mm Positioning: Straight		Advance angle [Degrees] Only available for "pitch circle".
N  <b>Circ.slot</b> T D1 F 0.000 mm/tooth FZ 0.000 mm/tooth V 0 m/min Face C Machining: ▽ Full circle X0 0.000 abs Y0 0.000 abs Z0 0.000 abs W 0.000 R 0.000 a0 0.000 ° a1 0.000 ° N 0 Z1 0.000 inc D2 0.000 UXY 0.000 mm Positioning: Straight		Number of slots

## Section 5

### Slot

Parameter	Help picture	Description and hints	Notes	
<p>Positioning</p> <p><b>Circ.slot</b></p> <p>T D1 F 8.000 mm/tooth FZ 8.000 mm/tooth V 0 m/min</p> <p>Face C</p> <p>Machining: Full circle</p> <p>X0 8.000 abs Y0 8.000 abs Z0 8.000 abs W 8.000 R 8.000 α0 8.000 ° α1 8.000 °</p> <p>N 8 Z1 8.000 inc DZ 8.000 UXV 8.000 mm</p> <p>Positioning: <b>Straight</b></p>		<p>Straight</p> <p><b>Straight</b></p> <p>Circle</p> <p><b>Circle</b></p>	<p>The next position is approached along a straight with rapid traverse.</p> <p>The next position is approached on a circular path at the feed rate, which is stored in a machine datum.</p>	

## Section 6

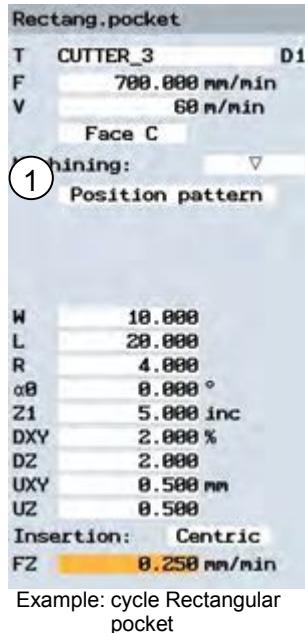
Notes

### Position pattern when milling

If pockets, spigots and longitudinal slots are to be machined at several locations, a separate position block must be programmed following the technology block.

The switching from "Single position" to "Position pattern" in the parameter field must be effected together with the call-up of the milling cycle. In this case the parameters for the reference point of the single position X0, Y0 and Z0 are suppressed.

Example (Milling cycle, "rectangular pocket" on a grid):



Example: cycle Rectangular pocket



Press HSK 3 "Milling".



Press VSK 1 "Pocket".



Press VSK 3 "Rectangular pocket".



Place cursor on the parameter field "Single position" (1).



Select the parameter alternatively "Position pattern".



Enter the other technology values for the milling cycle.

Press VSK 8 "Accept".

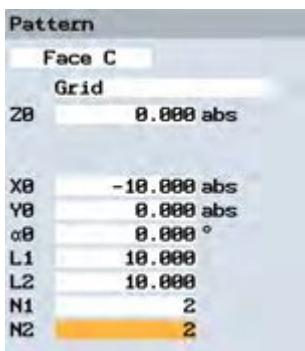
The cycle will be transferred to the work plan.



(2) The cycle "Rectangular pocket" is transferred to the work plan (Start of the linked machining, the bracket is open).

## Section 6

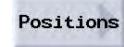
### Position pattern when milling



Example: Position pattern „grid“



Press HSK 2 "Drilling".



Press VSK 7 "Positions".



Press VSK 4 „Line“ and select, for example "grid".



Enter the position values for the pockets.

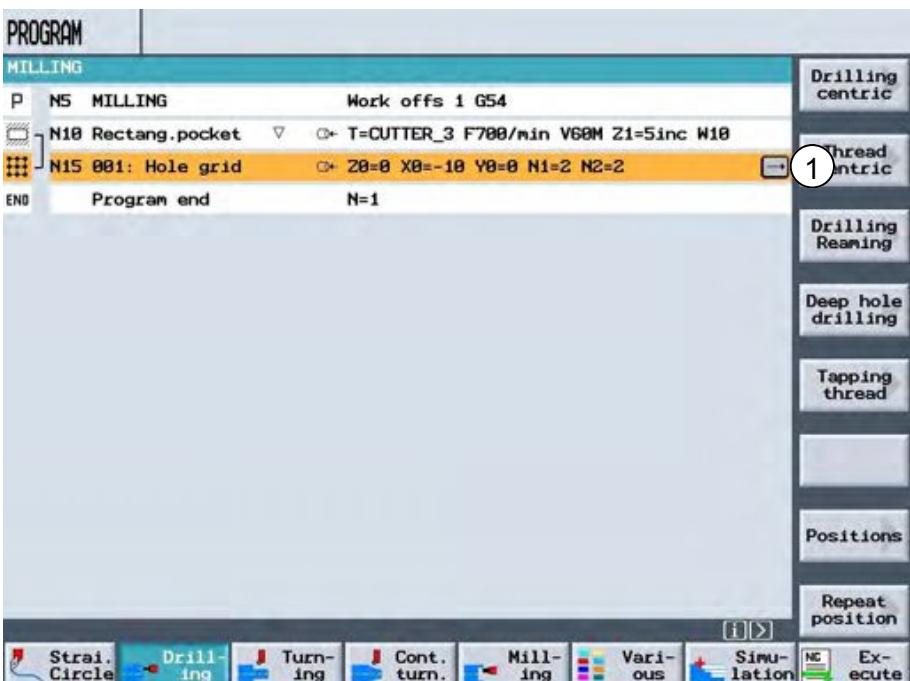


Press VSK 8 "Accept.".



The position values are transferred to the work plan.

Notes



①

The position pattern is transferred to the work plan (end of the linked machining, the bracket is closed).

The position pattern "Grid" is used only as an example. All other position patterns can also be programmed. (See Module A115 - Section 8 "Position patterns")

## Section 7

### Multi-edge

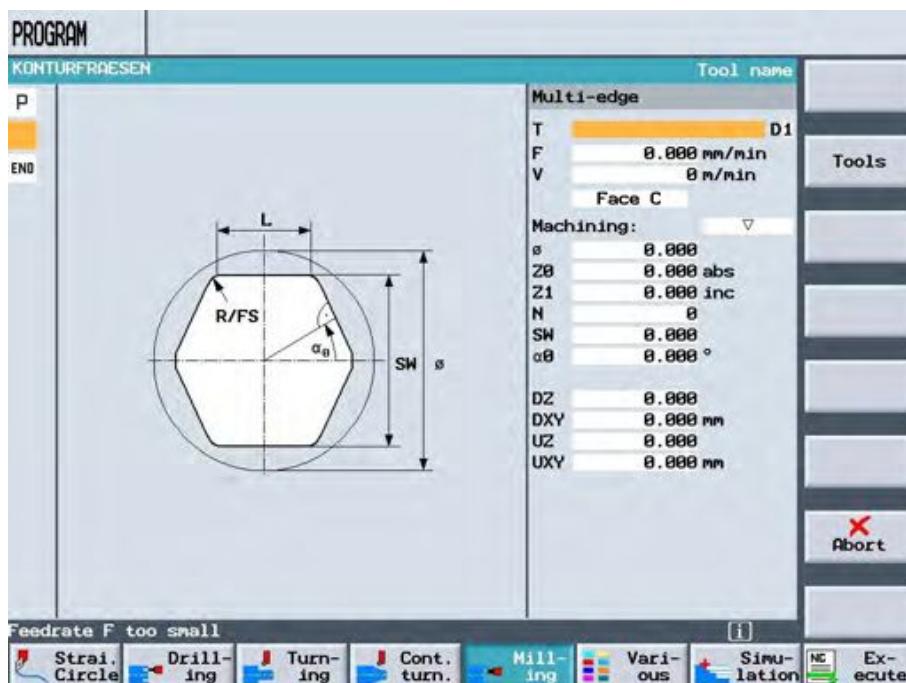
Notes

In the milling range, ShopTurn provides the possibility of the milling of concentric multi-edged contours with any number of corners on the end face.

The following shapes with or without corner radius or chamfer are available:

Mehrkant

By pressing the VSK 4 "Multi-edge" select the range for the programming of multi-edges.



**Approach/leaving the contour:**

1. The tool traverses with rapid traverse at the level of the retraction plane to the starting point and the moves to the safety level.
2. The tool approaches the multi-edge with a quarter circle at machining feed rate. Now follows the infeed to the machining depth and the further motion in the plane. The multi-edge will be machined clockwise or anti-clockwise depending on the selected sense of rotation (conventional/climb milling).
3. When the first plane is finished, the tool leaves the contour with a quarter circle and the infeed to the next plane ensues.

## Section 7

### Multi-edge

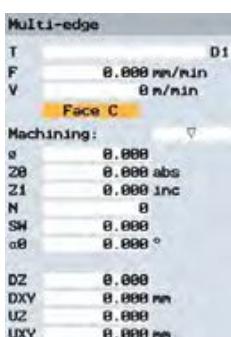
4. Again the multi-edge is approached with a quarter circle. This process is repeated until the programmed depth of the multi-edge is reached.
5. The tool is retracted with rapid traverse to the safety level.

Notes

Tool

Leaving  
the  
contour      Approach  
h of  
contour

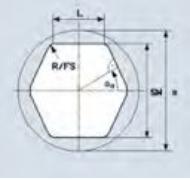
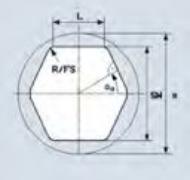
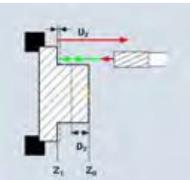
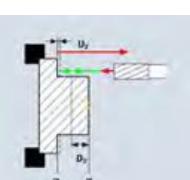
Multi-edge

Parameter	Hilfebild	Beschreibung und Hinweise	
T, D, F, S / V		<i>See module A115 – "Operating Area Program – Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".</i>	
Location		<b>Face C</b> <b>Face Y</b>	Face C Face Y

## Section 7

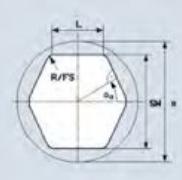
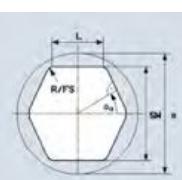
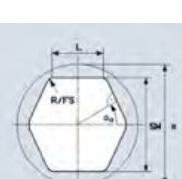
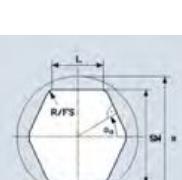
### Multi-edge

Notes

Parameter	Help picture	Description and hints
Machining		<p>When roughing the individual planes of the pocket are machined from the centre outward until the depth Z1 is attained.</p> <p>The wall is always machined first when finishing. For this the multi-edge is approached with a quarter circle at a point where one of the faces ends. With the last infeed the bottom is also finished from the outside inward.</p> <p>The finishing of the edge is similar to finishing, only the machining of the bottom with the last infeed is omitted.</p>
$\emptyset$		Diameter of the raw material
Z0		Reference point [abs]
Z1		End point Z1 [abs, inc]

## Section 7

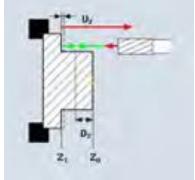
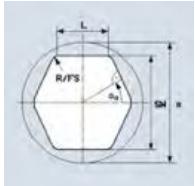
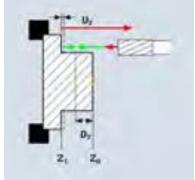
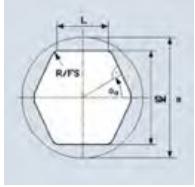
### Multi-edge

Parameter	Help picture	Description and hints	Notes
N		Number of edges (1, 2,.....)	
SW, L		A/F: Width across flats Available only with "N = 1" or „N = even numbered“. Alternatively: L: Length of edge	
alpha_0		Rotation angle of the first corner referred to the X-axis $\alpha_0 > 0$ : Multi-edge rotates clockwise $\alpha_0 < 0$ : Multi-edge rotates anti-clockwise.	
R, FS		Rounding – from 3 edges Alternatively: FS: Chamfer – from 3 edges	

## Section 7

### Multi-edge

Notes

Parameter	Help picture	Description and hints
DZ		Max. Infeed depth - Z-direction
DXY		Maximum infeed in the XY-plane [mm] Alternatively: Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%]
UZ		Finishing allowance at the bottom Available only for "Roughing" and "Finishing".
UXY		Finishing allowance in the plane

## Section 8

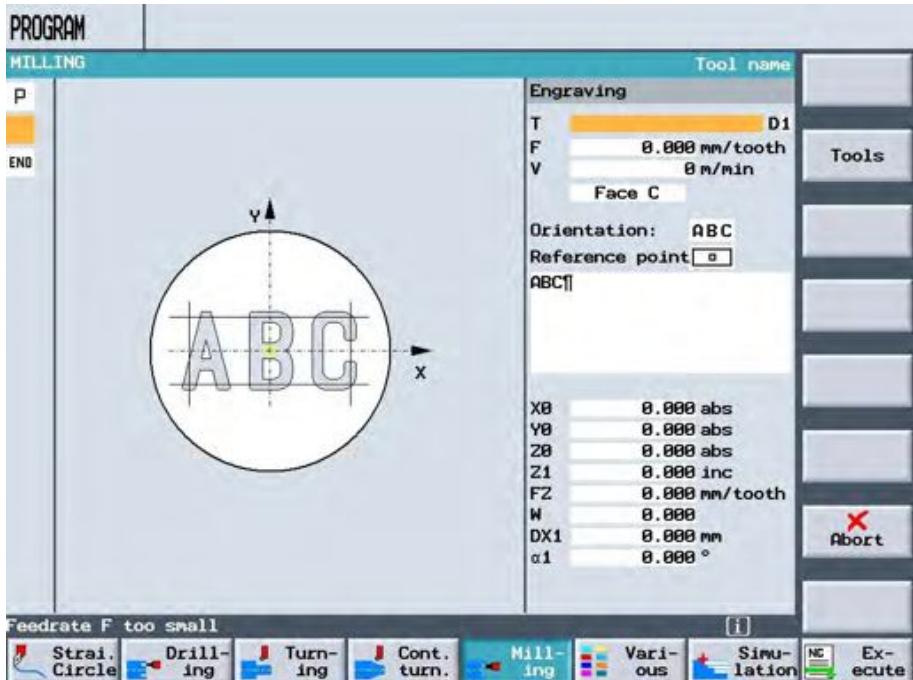
### Engraving cycle

By means of the function "Engraving" it is possible to engrave text along a line or a circular arc.

Engraving

By pressing the VSK 7 "Engraving" select the section for the programming of engravings.

Notes



**Hint:**

ShopTurn uses a proportional type, i.e. the individual characters are of variable width.

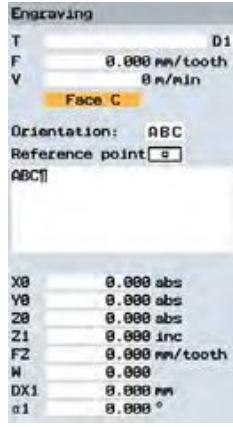
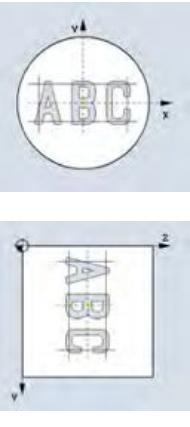
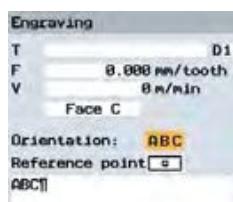
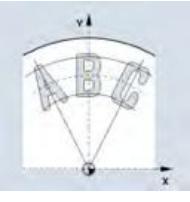
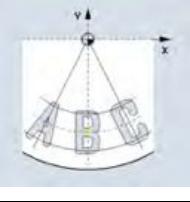
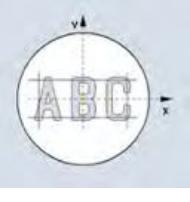
**Approach/leaving the contour:**

1. The tool moves with rapid traverse at the level of the retraction plane to the Start point and then feeds to the safety distance.
2. The tool moves with the infeed feed rate by FZ or FX to the machining depth Z1 or X1 and mills the first character.
3. The tool retracts with rapid traverse to the safety distance and moves along a straight to the next character.
4. Step 2 and 3 are repeated until the whole text has been milled.

## Section 8

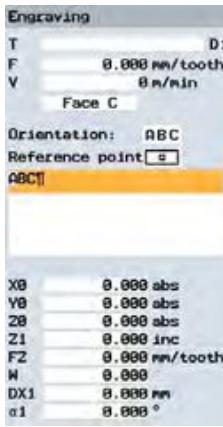
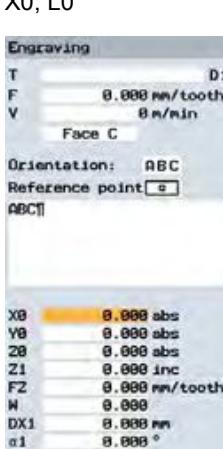
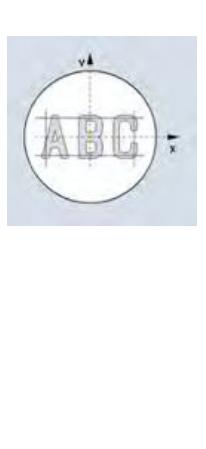
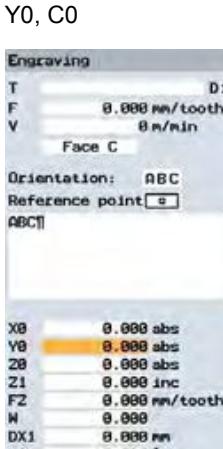
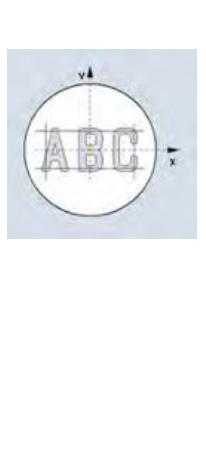
### Engraving cycle

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		See module A115 – "Operating Area Program – Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".
Location	 	<b>Face C</b> Face C <b>Face Y</b> Face Y <b>Per.surf.C</b> Perimeter C <b>Per.surf.Y</b> Perimeter Y
Alignment	  	Alignment of text along a line. Alignment of text along the upper part of a circular arc. Alignment of text along the lower part of a circular arc.
Reference point		Location of the reference point within the text e.g. Text centre Further reference points lie at the corners of the text and at the beginning and end of a line through the middle of the text

## Section 8

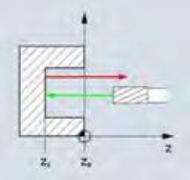
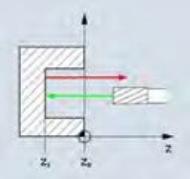
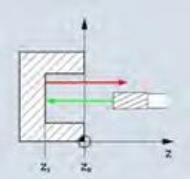
### Engraving cycle

Parameter	Help picture	Description and hints	Notes
Engraving text   <pre> Engraving T          D1 F          0.000 mm/tooth V          0 m/min Face C Orientation: ABC Reference point: e ABC    X0      0.000 abs Y0      0.000 abs Z0      0.000 abs Z1      0.000 inc FZ      0.000 mm/tooth W       0.000 DX1     0.000 mm d1      0.000 ° </pre>		Input field for the engraving text max. 91 characters (See Section 8.1)	
X0, L0   <pre> Engraving T          D1 F          0.000 mm/tooth V          0 m/min Face C Orientation: ABC Reference point: e ABC    X0      0.000 abs Y0      0.000 abs Z0      0.000 abs Z1      0.000 inc FZ      0.000 mm/tooth W       0.000 DX1     0.000 mm d1      0.000 ° </pre>		X0: Reference point in X-direction [abs] Alternatively: L0: Reference point polar length	
Y0, C0   <pre> Engraving T          D1 F          0.000 mm/tooth V          0 m/min Face C Orientation: ABC Reference point: e ABC    X0      0.000 abs Y0      0.000 abs Z0      0.000 abs Z1      0.000 inc FZ      0.000 mm/tooth W       0.000 DX1     0.000 mm d1      0.000 ° </pre>		Y0: Reference point in Y-direction [abs] Alternatively: C0: Reference point polar angle	

## Section 8

### Engraving cycle

Notes

Parameter	Help picture	Description and hints
<b>Face C:</b>		
Z0		Reference point in Z-direction [abs]
Z1		Machining depth [abs, inc]
FZ		Plunging feed rate [mm/min, mm/tooth]

## Section 8

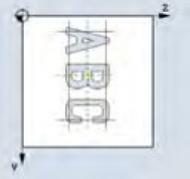
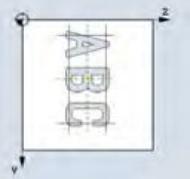
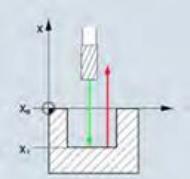
### Engraving cycle

Parameter	Help picture	Description and hints	Notes																					
Face C:																								
W	<p>Engraving</p> <table border="0"> <tr><td>T</td><td>0.000 mm/tooth</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td><td>V</td></tr> <tr><td>V</td><td>0 m/min</td><td></td></tr> <tr><td></td><td></td><td>Face C</td></tr> <tr><td>Orientation:</td><td>ABC</td><td></td></tr> <tr><td>Reference point</td><td>■</td><td></td></tr> <tr><td>ABC</td><td>■</td><td></td></tr> </table> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs Z1 0.000 inc FZ 0.000 mm/tooth W 0.000 DX1 0.000 mm α1 0.000 °</p>	T	0.000 mm/tooth	D1	F	0.000 mm/min	V	V	0 m/min				Face C	Orientation:	ABC		Reference point	■		ABC	■		<p>Character height [mm]</p>	
T	0.000 mm/tooth	D1																						
F	0.000 mm/min	V																						
V	0 m/min																							
		Face C																						
Orientation:	ABC																							
Reference point	■																							
ABC	■																							
DX1, DX2, α2	<p>Engraving</p> <table border="0"> <tr><td>T</td><td>0.000 mm/tooth</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td><td>V</td></tr> <tr><td>V</td><td>0 m/min</td><td></td></tr> <tr><td></td><td></td><td>Face C</td></tr> <tr><td>Orientation:</td><td>ABC</td><td></td></tr> <tr><td>Reference point</td><td>■</td><td></td></tr> <tr><td>ABC</td><td>■</td><td></td></tr> </table> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs Z1 0.000 inc FZ 0.000 mm/tooth W 0.000 DX1 0.000 mm α1 0.000 °</p>	T	0.000 mm/tooth	D1	F	0.000 mm/min	V	V	0 m/min				Face C	Orientation:	ABC		Reference point	■		ABC	■		<p>Distance between characters [mm]</p> <p>Alternatively:</p> <p>DX2: Total width [mm]</p> <p>Available only with "Linear alignment".</p> <p>Alternatively:</p> <p>α2: Opening angle [Degree]</p> <p>Available only with "Curved alignment".</p> <p>The centre point of the circular arc is the work piece zero point..</p>	
T	0.000 mm/tooth	D1																						
F	0.000 mm/min	V																						
V	0 m/min																							
		Face C																						
Orientation:	ABC																							
Reference point	■																							
ABC	■																							
α1	<p>Engraving</p> <table border="0"> <tr><td>T</td><td>0.000 mm/tooth</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td><td>V</td></tr> <tr><td>V</td><td>0 m/min</td><td></td></tr> <tr><td></td><td></td><td>Face C</td></tr> <tr><td>Orientation:</td><td>ABC</td><td></td></tr> <tr><td>Reference point</td><td>■</td><td></td></tr> <tr><td>ABC</td><td>■</td><td></td></tr> </table> <p>X0 0.000 abs Y0 0.000 abs Z0 0.000 abs Z1 0.000 inc FZ 0.000 mm/tooth W 0.000 DX1 0.000 mm α1 0.000 °</p>	T	0.000 mm/tooth	D1	F	0.000 mm/min	V	V	0 m/min				Face C	Orientation:	ABC		Reference point	■		ABC	■		<p>α1: Text direction [Degrees]</p> <p>Available only with "Linear alignment".</p>	
T	0.000 mm/tooth	D1																						
F	0.000 mm/min	V																						
V	0 m/min																							
		Face C																						
Orientation:	ABC																							
Reference point	■																							
ABC	■																							

## Section 8

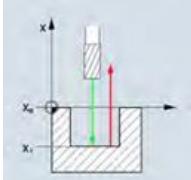
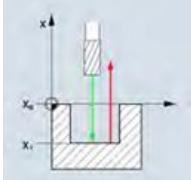
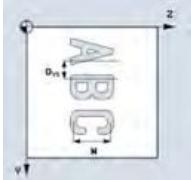
### Engraving cycle

Notes

Parameter	Help picture	Description and hints
<b>Perimeter C:</b>		
Y0, C0, R	 <b>Engraving</b> T D1 F 0.000 mm/tooth V 8 m/min Per.surf.C Orientation: ABC Reference point <input type="checkbox"/> ABC    Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm α1 0.000 °	Y0: Reference point in Y-direction [abs] Alternatively: C0: Reference point [Degrees] Available only with "Linear alignment". Alternatively: R: Reference point polar length [abs] Available only with "Curved alignment".
Z0, α0	 <b>Engraving</b> T D1 F 0.000 mm/tooth V 8 m/min Per.surf.C Orientation: ABC Reference point <input type="checkbox"/> ABC    Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm α1 0.000 °	Z0: Reference point in Z-direction [abs] Alternatively: α0: Reference point polar angle [Degrees] Available only with "Curved alignment".
X0	 <b>Engraving</b> T D1 F 0.000 mm/tooth V 8 m/min Per.surf.C Orientation: ABC Reference point <input type="checkbox"/> ABC    Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm α1 0.000 °	Reference point in X-direction [abs]

## Section 8

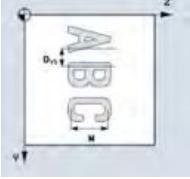
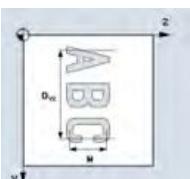
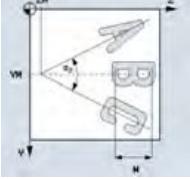
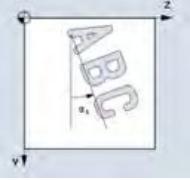
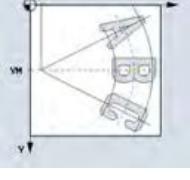
### Engraving cycle

Parameter	Help picture	Description and hints	Notes
X1	<p>Perimeter C:</p>  <pre> Engraving T D1 F 0.000 mm/tooth V 0 m/min Per.surf.C Orientation: ABC Reference point e ABC1  Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DV1 0.000 mm a1 0.000 ° </pre>	Machining depth [abs, inc]	
FX	 <pre> Engraving T D1 F 0.000 mm/tooth V 0 m/min Per.surf.C Orientation: ABC Reference point e ABC1  Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DV1 0.000 mm a1 0.000 ° </pre>	Plunging feed rate [mm/min, mm/tooth]	
W	 <pre> Engraving T D1 F 0.000 mm/tooth V 0 m/min Per.surf.C Orientation: ABC Reference point e ABC1  Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DV1 0.000 mm a1 0.000 ° </pre>	Character height [mm]	

## Section 8

### Engraving cycle

Notes

Parameter	Help picture	Description and hints
Perimeter C:		
DY1, DY2, α2	  	<p>DY1: Distance between characters [mm]</p> <p>Alternatively:</p> <p>DY2: Total width [mm]</p> <p>Available only with "Linear alignment".</p> <p>Alternatively:</p> <p>α2: Opening angle [Degree]</p> <p>Available only with "Curved alignment".</p>
α1		<p>Text direction [Degrees]</p> <p>Available only with "Linear alignment".</p>
YC, CC		<p>YC: Centre point of arc [abs]</p> <p>Alternatively:</p> <p>CC: Centre point of arc polar angle [Degrees]</p> <p>Available only with "Curved alignment".</p>

## Section 8

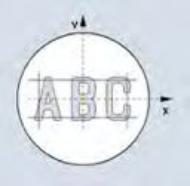
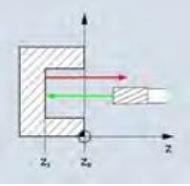
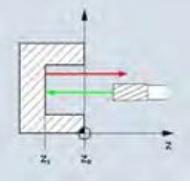
### Engraving cycle

Parameter	Help picture	Description and hints	Notes
Perimeter C:			
ZC	<p><b>Engraving</b></p> <p>T D1 F 0.000 mm/tooth V 0 m/min Per.surf.C Orientation: ABC Reference point <input type="checkbox"/> ABC  </p> <p>YB 0.000 abs ZB 0.000 abs XB 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm YC 0.000 abs ZC 0.000 abs</p>	<p>Centre point of arc</p> <p>[abs]</p> <p>Available only with "Curved alignment".</p>	
Face Y:	<p><b>CP</b></p> <p><b>Engraving</b></p> <p>T D1 F 0.000 mm/tooth V 0 m/min Face Y Orientation: ABC Reference point <input type="checkbox"/> ABC  </p> <p>CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs Z1 0.000 inc FZ 0.000 mm/tooth W 0.000 DX1 0.000 mm α1 0.000 °</p>	<p>Positioning angle for machining range</p> <p>[Degrees]</p>	
X0, L0	<p><b>Engraving</b></p> <p>T D1 F 0.000 mm/tooth V 0 m/min Face Y Orientation: ABC Reference point <input type="checkbox"/> ABC  </p> <p>CP 0.000 ° X0 0.000 abs Y0 0.000 abs Z0 0.000 abs Z1 0.000 inc FZ 0.000 mm/tooth W 0.000 DX1 0.000 mm α1 0.000 °</p>	<p>X0: Reference point in X-direction</p> <p>[abs]</p> <p>Alternatively:</p> <p>L0: Reference point polar length</p>	

## Section 8

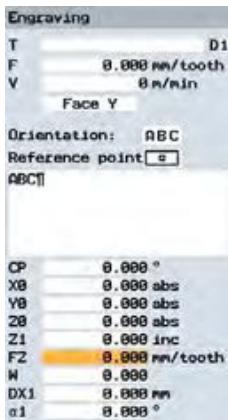
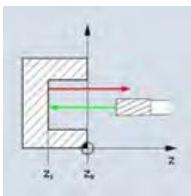
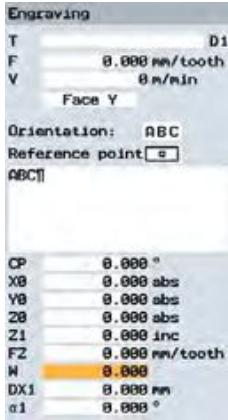
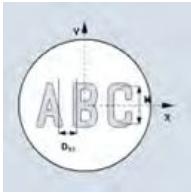
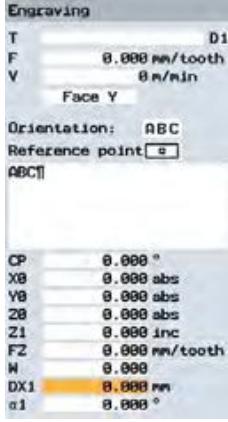
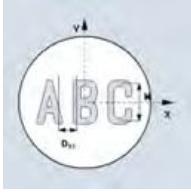
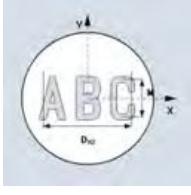
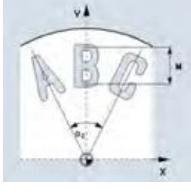
### Engraving cycle

Notes

Parameter	Help picture	Description and hints
Face Y:		
Y0, C0		Reference point in Y-direction [abs] Alternatively: Reference point polar angle [Degrees]
Z0		Reference point in Z-direction [abs]
Z1		Machining depth [abs, inc]

## Section 8

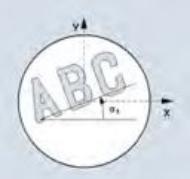
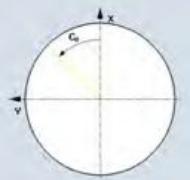
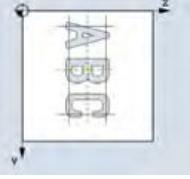
### Engraving cycle

Parameter	Help picture	Description and hints	Notes
<b>Face Y:</b>  <b>FZ</b>  		Plunging feed rate [mm/min, mm/tooth]	
<b>W</b>  <b>Engraving</b>  		Character height [mm]	
<b>DX1, DX2, α2</b>  <b>Engraving</b>  		DX1: Distance between characters [mm] Alternatively:  	
		DX2: Total width [mm] Available only with "Linear alignment". Alternatively:  	
		α2: Opening angle [Degrees] Available only with "Curved alignment". The centre point of the arc is the work piece zero point.	

## Section 8

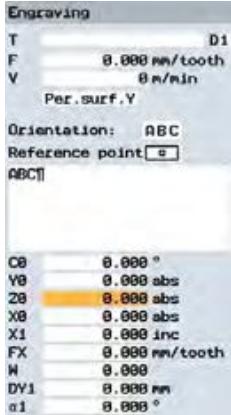
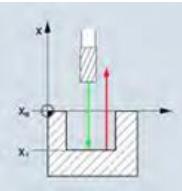
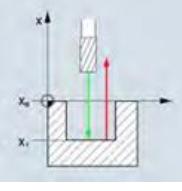
### Engraving cycle

Notes

Parameter	Help picture	Description and hints
Face Y:		
a1		Text direction [Degrees] Available only with "Linear alignment".
Perimeter Y:		
C0		Reference point
Y0		Y0: Reference point [abs] Alternatively: R: Reference point polar length [abs]

## Section 8

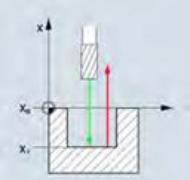
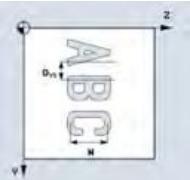
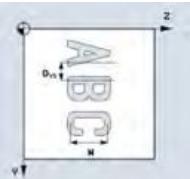
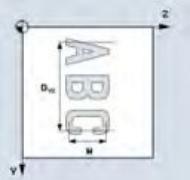
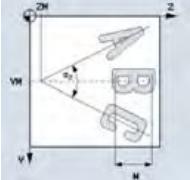
### Engraving cycle

Parameter	Help picture	Description and hints	Notes
Z0	<p>Perimeter Y:</p> 	<p>Z0: Reference point in Z-direction [abs]</p> <p>Alternatively:</p> <p><math>\alpha_0</math>: Reference point polar angle [Degrees]</p> <p>Available only with "Curved alignment".</p>	
X0		Reference point in X-direction [abs]	
X1		Machining depth [abs, inc]	

## Section 8

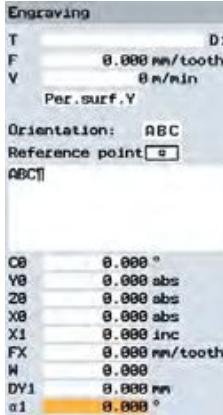
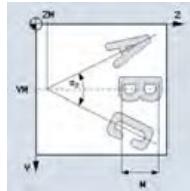
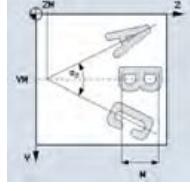
### Engraving cycle

Notes

Parameter	Help picture	Description and hints
Perimeter Y:		
FX	 <b>Engraving</b> T D1 F 0.000 mm/tooth V 8 m/min Per.surf.Y Orientation: ABC Reference point: ABC    C0 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm α1 0.000 °	Plunging feed rate [mm/min, mm/tooth]
W	 <b>Engraving</b> T D1 F 0.000 mm/tooth V 8 m/min Per.surf.Y Orientation: ABC Reference point: ABC    C0 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm α1 0.000 °	Character height [mm]
DY1, DY2, α2	   <b>Engraving</b> T D1 F 0.000 mm/tooth V 8 m/min Per.surf.Y Orientation: ABC Reference point: ABC    C0 0.000 ° Y0 0.000 abs Z0 0.000 abs X0 0.000 abs X1 0.000 inc FX 0.000 mm/tooth W 0.000 DY1 0.000 mm α1 0.000 °	<p>DY1: Distance between characters [mm]</p> <p>Alternatively:</p> <p>DX2: Total width [mm]</p> <p>Available only with "Linear alignment".</p> <p>Alternatively:</p> <p>α2: Opening angle [Degrees]</p> <p>Available only with "Curved alignment".</p> <p>The centre point of the arc is the work piece zero point.</p>

## Section 8

### Engraving cycle

Parameter	Help picture	Description and hints	Notes
Perimeter Y:			
<b>α1</b>  <b>C8</b> 0.000 ° <b>Y8</b> 0.000 abs <b>Z8</b> 0.000 abs <b>X8</b> 0.000 abs <b>X1</b> 0.000 inc <b>FX</b> 0.000 mm/tooth <b>W</b> 0.000 <b>DY1</b> 0.000 mm <b>α1</b> 0.000 °	Text direction [Degrees] Available only with "Linear alignment".		
<b>YC</b>  <b>C8</b> 0.000 ° <b>Y8</b> 0.000 abs <b>Z8</b> 0.000 abs <b>X8</b> 0.000 abs <b>X1</b> 0.000 inc <b>FX</b> 0.000 mm/tooth <b>W</b> 0.000 <b>DY1</b> 0.000 mm <b>YC</b> 0.000 abs <b>ZC</b> 0.000 abs	Centre of circular arc [abs] Available only with "Curved alignment".		
<b>ZC</b>  <b>C8</b> 0.000 ° <b>Y8</b> 0.000 abs <b>Z8</b> 0.000 abs <b>X8</b> 0.000 abs <b>X1</b> 0.000 inc <b>FX</b> 0.000 mm/tooth <b>W</b> 0.000 <b>DY1</b> 0.000 mm <b>YC</b> 0.000 abs <b>ZC</b> 0.000 abs	Centre of circular arc [abs] Available only with "Curved alignment".		

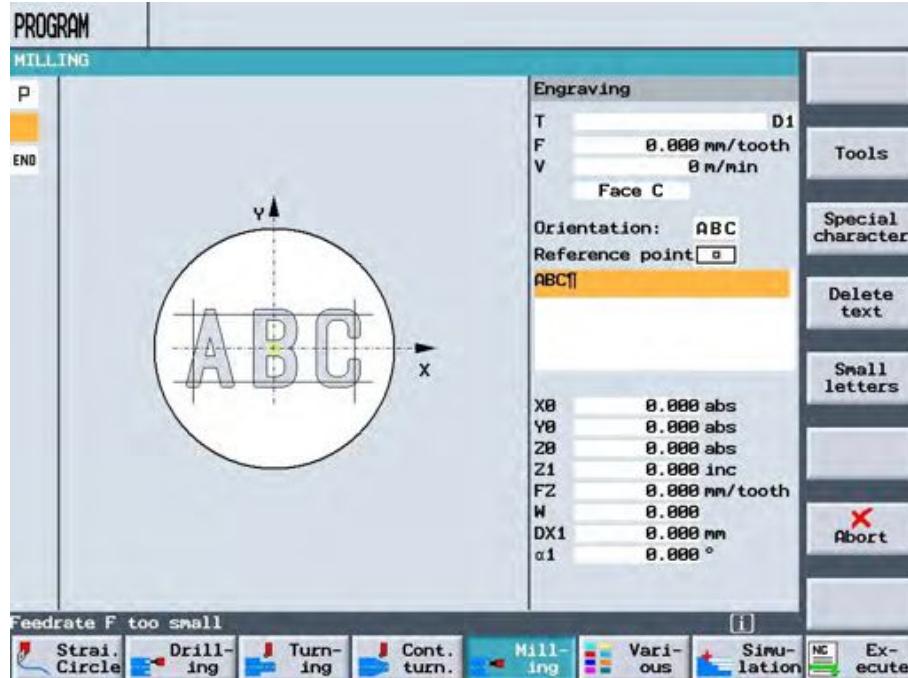
## Section 8

Notes

### Engraving cycle

#### 8.1 Vertical Softkey strip

For the selection of the input field for the engraving text a number of various functions for the input of text are available.



Special character

By pressing the VSK 3 "Special character" a window with special characters, which are not available on the keyboard, is displayed.



#### Plunging of special characters:

1. Set the cursor on the required character.
2. Press VSK 8

The special character will be inserted at the current cursor position.

## Section 8

### Engraving cycle

Delete  
text

By pressing the VSK 4 "Delete text" the entire text will be cleared.

Notes

✓  
OK

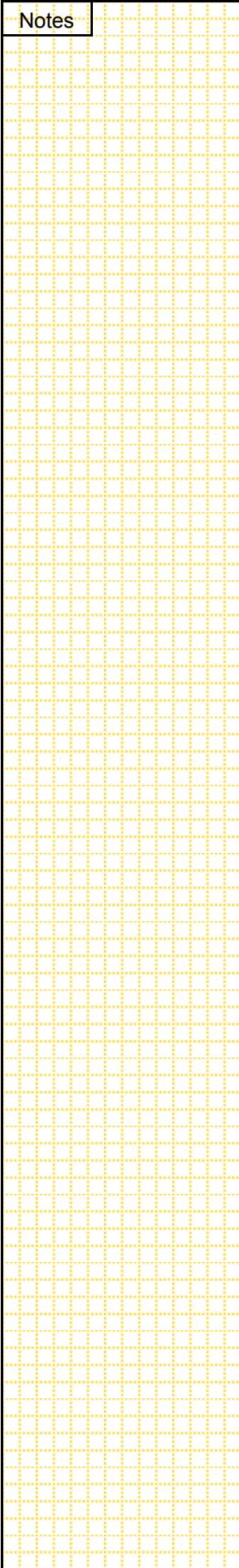
The deletion must be acknowledged with the VSK 8 "OK".

Small  
letters

By pressing the VSK 5 "Small letters" lower case letters can be entered.

By pressing the Softkey once more will revert to capitals.

Notes



## 1 Brief description

### Aim of the module:

This module explains the programming of contours and their machining under ShopTurn.

### Description of the module:

Apart from previously described standard milling functions ShopTurn also permits the programming and machining of simple and complex contours.

To this end the contour must be programmed using the following contour elements:

- Straight
- Oblique
- Circle

Following this the contour can be machined using the following technologies:

- Continuous control milling
- Pre-drilling (centring and drilling)
- Pocket milling and removal of any residue material from the pocket
- Milling of spigots and removal of any residue material from the spigots

### Content of the module:

- Pre-selection and general functions under "Contour milling"
- Pre-selection of the function "Contour milling"
- Vertical Softkey-strip
- Basic hints
- Description of the contour
- Starting a contour
- Entering of contour elements
- Alteration of contour elements
  - Adding a contour element
  - Alteration of a contour element
  - Changing the dialogue selection
  - Insertion of a contour element
  - Deletion of a contour element
- Path milling
- Pre-drilling

# Operating Area Program Editor - Programming contour milling

Milling of contour pockets and removal of any residue material

- Milling of contour pockets
- Milling of spigots and removal of any residue material from contour pocket

Milling of spigots and removal of any residue material

- Milling of spigots
- Milling of spigots and removal of any residue material from spigot contour

Pre-selection and general functions under "Contour milling"

Pre-selection of the function  
"Contour milling"

Vertical Softkey-strip

Basic hints

Description of the contour

Section 2

Starting a contour

Section 3

Entering of contour elements

Section 4

Alteration of contour elements

Adding a contour element

Alteration of a contour element

Changing the dialogue selection

Insertion of a contour element

Deletion of a contour element

Section 5

## Operating Area Program Editor - Programming contour milling

Continuous path milling

Section 6

Pre-drilling

Section 7

Milling of contour pockets and milling of spigots and removal of any residue material from contour pocket

Milling of contour pockets

Milling of spigots and removal of any residue material from contour pocket

Section 8

Milling of spigots and removal of any residue material from contour spigot

Milling of spigots

Milling of spigots and removal of any residue material from contour spigots

Section 9

## Section 2

Notes

Pre-selection and general functions under "Contour milling"

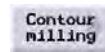
### 2.1 Pre-selection of the function "Contour milling"

After writing the program heading the program blocks for the machining, e.g. for the contour milling, can be programmed.

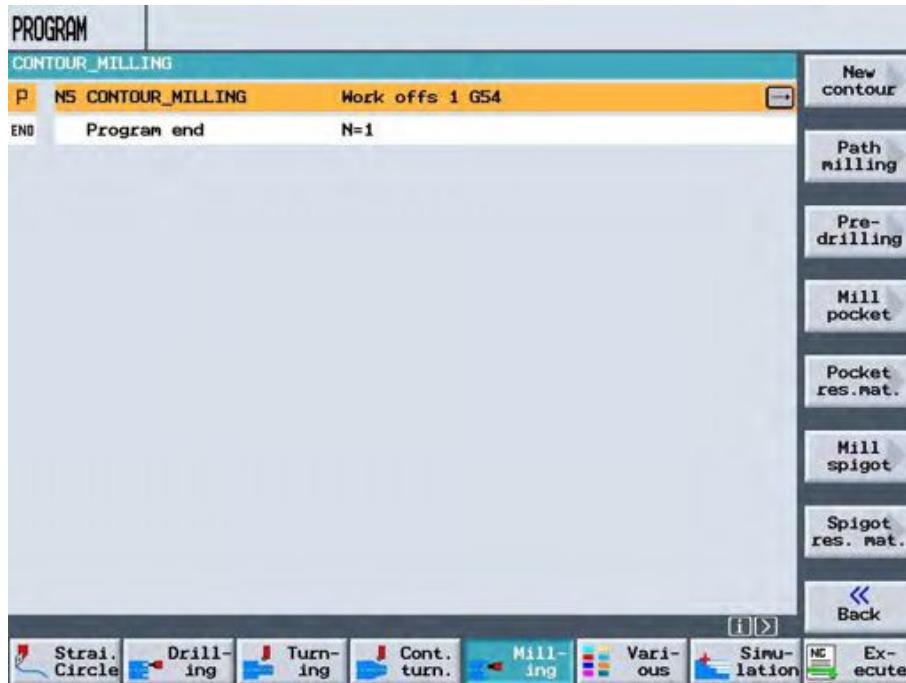
In this module it is assumed that the machining plane is the XY-plane and the tool axis is the Z-axis. The tool axis is selected in the program heading, see *Module A112 - "Operating Area Program Editor - Program header"*.



By pressing the HSK 5 "Milling" the range for the programming of standard milling cycles (See *Module A121 "Operating Area Program - Standard milling cycles"*).



By pressing the VSK 8 "Contour milling" the function for the programming of contours will be called up.



## Section 2

### Pre-selection and general functions under "Contour milling"

#### 2.2 Vertical Softkey-strip

New contour

By pressing the VSK 1 "New contour" a new contour is started. Following this the contour elements can be programmed.

See

- Section 3 "Starting a contour",*
- Section 4 "Entering of contour elements"*
- Section 5 "Hints regarding the contour"*

Path milling

By pressing the VSK 2 "Path milling" the respective function is called up.

See *Section 6 "Continuous path milling"*.

Pre-drilling

By pressing the VSK 3 "Pre-drilling" the following functions will be called up:

- Centring
- Pre-drilling

See *Section 7 "Pre-drilling"*.

Mill pocket

By pressing the VSK 4 "Mill pocket" the respective function will be called up.

See *Section 8.1 "Mill pocket"*.

Pocket res.mat.

By pressing the VSK 5 "Pocket res.mat." the function "Milling of spigots and removal of any residue material from contour pocket" will be called up.

See *Section 8.2 "Milling of spigots and removal of any residue material from contour pocket"*.

Mill spigot

By pressing the VSK 6 "Mill spigot" the respective function will be called up.

See *Section 9.1 "Mill spigot"*.

Spigot res. mat.

By pressing the VSK 7 "Spigot res.mat." the function "Milling of spigots and removal of any residue material from spigot contour" will be called up.

See *Section 9.2 "Milling of spigots and removal of any residue material from spigot contour"*.

Notes

## Section 2

Pre-selection and general functions under "Contour milling"

Notes

### 2.3 Basic hints

**Types of contours:** The following contours can be programmed:  
open contours and  
closed contours (pockets, islands and spigots)

**Number of contour elements:** A contour consists of individual contour elements. A contour must consist of:  
at least 2 elements and  
a maximum of 250 elements

Additionally it is possible to programme  
chamfers,  
radii or  
tangential transitions  
between the contour elements.

**Contour calculator:** The integrated contour calculator evaluates point of intersection of individual contour elements taking into account geometrical aspects and permits the input of not sufficiently dimensioned elements.

**Programming** When contour milling the contour must always be programmed first followed by the technology.  
The contours can be  
machined with path milling  
cleaned-up as pockets with/without islands  
and cleared as spigots.

### Examples for the programming of contours

#### Any contour:

1. Enter the contour  
Create the contour by adding different contour elements following the previous one.
2. Path milling (roughing )  
The contour will be machined taking into account the approach and retraction strategies.
3. Continuous path milling (finishing)  
If during roughing a finishing allowance was programmed, the contour will be finished.

## Section 2

### Pre-selection and general functions under "Contour milling"

#### Contours for pockets or islands:

Contours for pockets or islands must be closed, i.e. the start and end point of the contour are identical. It is also possible to mill pockets with one or more islands or which overlap. The first stated contour is taken by ShopTurn to be a pocket contour, all others are treated as islands.

Notes

The machining of contour pockets with islands is programmed as follows:

1. Enter the contour of the pocket
2. Enter the contour of the island  
The contour of the island is entered after the contour pocket.
3. Centring, pre-drilling of the contour pocket  
If the contour pocket is to be pre-drilled, prior centring is possible to prevent the drill from running out.
4. Pre-drilling the contour pocket  
If vertical plunging is intended for clearing the contour pocket and there is no slot drill available, the pocket can be pre-drilled.
5. Cleaning-up a contour pocket with island (roughing )  
The contour pocket is cleaned-up taking into account the plunging strategy
6. Clearing of residue material (roughing )  
For clearing the pocket ShopMill automatically recognizes any rest material that remained. Using a suitable tool this can be cleared away without re-machining the entire pocket.
7. Finishing a contour pocket with island (Finishing of edge)  
If during roughing a finishing allowance was left on the edge, the pocket edge will be finished with an additional pass.

All machining steps used for contour milling are summarized in the work plan with a square bracket.



Example of a work plan for machining a contour pocket

## Section 2

Notes

Pre-selection and general functions under "Contour milling"

**Contours for spigots:** Contours for spigots must be closed, i.e. the Start- and End point of the contour are identical. Several spigots can be defined and these may overlap. ShopMill interprets the first stated contour as being the raw-material contour, the rest as spigots.

The machining of contour spigots is programmed as follows:

1. Enter the contour of the raw material.  
The raw material contour specifies the range outside of which there is no material; i.e. in that range all motions may take place with rapid traverse. The material between the raw material contour and the spigot contour will be removed.
2. Enter the contour of the spigot.  
The contour of the spigot must be entered after the raw material contour.
3. Roughing the spigot contour.  
The stock outside the spigot is removed.
4. Clearing of residue material (roughing )  
ShopMill automatically recognizes during milling, whether any material remains standing. With a suitable tool this rest material can be cleared without machining the spigot again.
5. Finishing the spigot contour (Finishing sides/bottom)  
If during roughing a finishing allowance for the sides/bottom was programmed, the sides and the bottom of the spigot will be machined once more.

## Section 2

Pre-selection and general functions under "Contour milling"

### 2.4 Description of the contour



ShopTurn shows a contour in the work plan as one program block (with an open square bracket).



If this block is opened with the key "Cursor right" the individual contour elements will be shown symbolically next to the graphics window.

Notes

Contour element	Symbol	Meaning
Start point		Start point of the contour
Straight upwards		Straight on a 90°-grid
Straight downwards		Straight on a 90°-grid
Straight to the right		Straight on a 90°-grid
Straight to the left		Straight on a 90°-grid
Any straight (Oblique)		Straight at any angle
Circular arc to the right (Clockwise)		Circle (Radius)
Circular arc to the left (Anti-clockwise)		Circle (Radius)
End of contour		End of the contour description

#### Status of the contour-elements

The various colours of the symbols indicate their status:

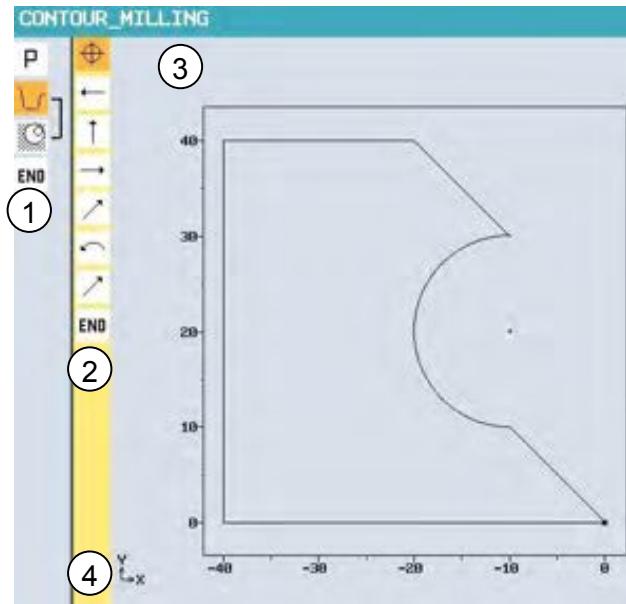
Symbol	foreground	Background	Meaning
	black	orange	Cursor located on a new or current element
	black	white	Normal element
	red	white	Element not observed at present

## Section 2

Notes

Pre-selection and general functions under "Contour milling"

**Graphic representation** Con-currently to the sequential input of the contour elements the progress of the contour programming is shown in the graphics window by way of a line graphics.



- ① Symbols of the work plan
- ② Symbols of the contour elements
- ③ Graphics window with scaling
- ④ Co-ordinate system

**Description of the contour elements** The produced contour elements can take on various line types and colours during programming .

Black	Completely programmed contour elements
Orange	Current contour element
Green dotted	Alternative contour element
Blue dotted	Partially described contour element

### Scaling

The scaling of the co-ordinate system adapts itself to the changes of the complete contour.

### Co-ordinate system

The position of the co-ordinate system is shown in the graphics window.

## Section 3

### Starting a contour

Notes

Every contour, that is to be milled, must have its own contour program.

The contours will be stored at the end of the program.

When a new contour is started, the starting point must be defined first of all. ShopMill defines the end of the contour automatically.

If the tool axis is changed, ShopMill automatically matches the starting point axes.

Any sort of additional commands for the starting point (max. 40 characters) can be entered in the form of G-Codes.

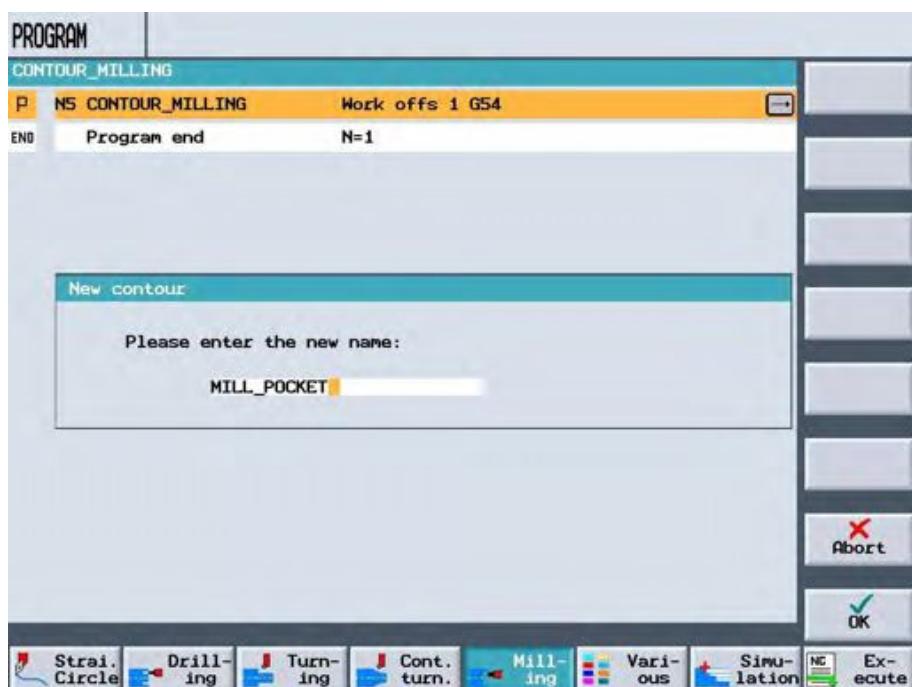
Hint:

If a contour is to be created, which is to be similar to an existing contour, the existing contour can be copied, re-named and the respective contour elements altered.

If, however, the same contour is to be used again at another place in the program, the copy must not be re-named. Alterations in the one contour will automatically be transferred to the other one.

New contour

By pressing the VSK 1 "New contour" the window for the contour name will appear.



Enter the name for the new contour.  
The contour name must be specific.

## Section 3

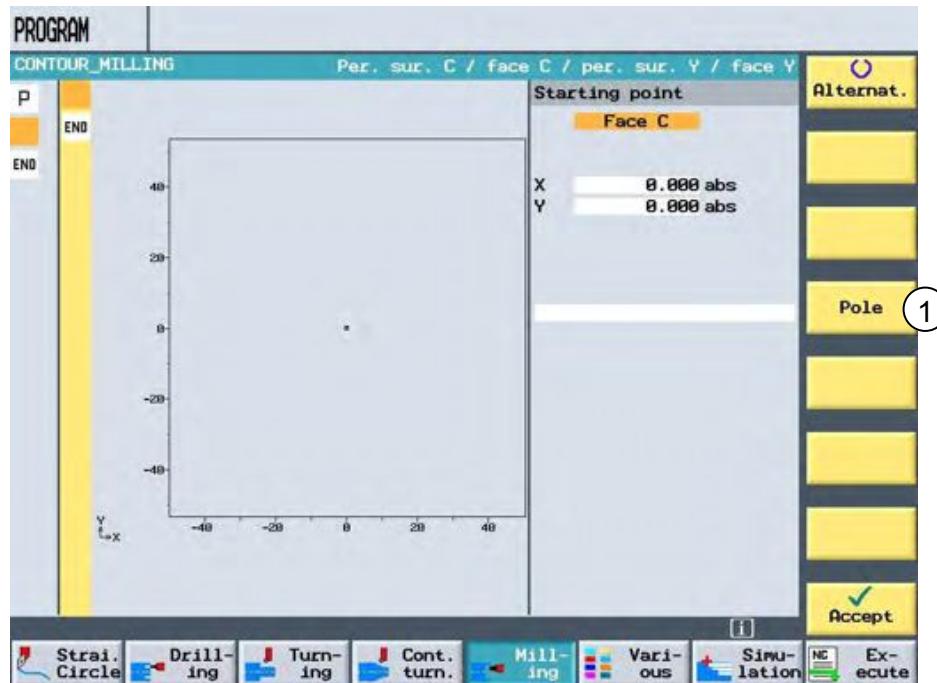
### Starting a contour

Notes



By pressing the VSK 8 "OK" a new contour will be started.

The window for the contour start point appears.



#### Start point Cartesian:

Enter the following parameters in the window:  
Tool axis

Absolute start point in X- and Y-axis  
(depending on the chosen tool axis)

Any auxiliary commands (max. 40 characters)



By pressing the VSK 8 "Accept." this Start point is laid down for the contour.

See also Module A100 - "Basic aspects for milling machine tools".

or

#### Start point polar:



After pressing the VSK 4 "Pole" (①) the Start point of the contour can be specified with polar co-ordinates.

In this case the pole is specified absolutely and the Start point of the contour is determined by means of the parameters L1 and  $\varphi_1$ .



By pressing the VSK 8 "Accept." this Start point will be specified for the contour.

See also Module A100 - "Basic Principles Milling machine".

## Section 3

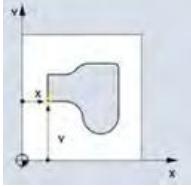
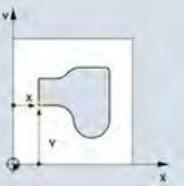
### Starting a contour

Parameter “Start point contour”

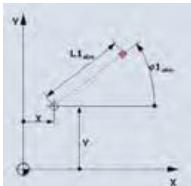
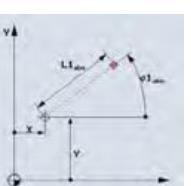
Notes

Parameter	Help picture	Description and hints	
Location		<b>Face C</b>	Face C
<b>Starting point</b> Face C		<b>Face Y</b>	Face Y
X 0.000 abs Y 0.000 abs		<b>Per.surf.C</b>	Perimeter face C
		<b>Per.surf.Y</b>	Perimeter face Y

Face C / Face Y:

Cartesian		
X, Y  <b>Starting point</b> Face C  X 0.000 abs Y 0.000 abs		X: Start point in X-direction [abs]  Y: Start point in Y-direction [abs]
Auxiliary commands  <b>Starting point</b> Face C  X 0.000 abs Y 0.000 abs		Input of auxiliary commands (max.40 characters)

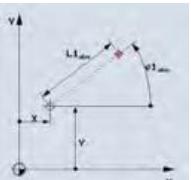
Face C / Face Y:

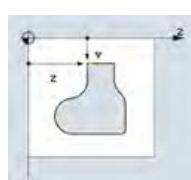
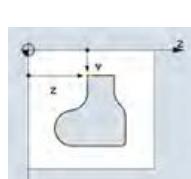
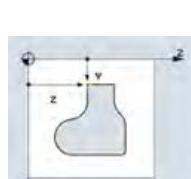
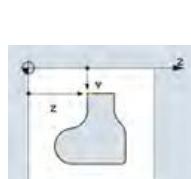
Polar		
X, Y  <b>Starting point</b> Face C  Pole: X 0.000 abs Y 0.000 abs  Starting point: L1 0.000 abs φ1 0.000 °abs		X: Position Pole in X-direction [abs]  Y: Position Pole in Y-direction [abs]
L1  <b>Starting point</b> Face C  Pole: X 0.000 abs Y 0.000 abs  Starting point: L1 0.000 abs φ1 0.000 °abs		Distance between the pole and the starting point of the contour [abs]

## Section 3

### Starting a contour

Notes

Parameter	Help picture	Description and hints
Face C / Face Y:		
Polar		
$\varphi_1$		Polar angle between pole and start point of the contour [abs]
Auxiliary commands		See Face C / Face Y, Cartesian

Perimeter face C:		
Cartesian		
$\emptyset$		Cylinder perimeter diameter
$Y$		Start point in Y-direction – replaces the value in the Ya-input field [abs]
$Ya$		Start angle – can be entered only if the Y-input field is empty [abs]
$Z$		Start point in Z-direction [abs]

## Section 3

### Starting a contour

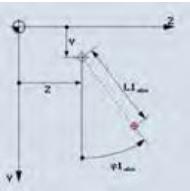
Notes

Parameter	Help picture	Description and hints								
Perimeter face C:										
Cartesian										
Auxiliary commands	<p>Starting point</p> <table border="1"> <tr><td>Par.surf.C</td><td>0.000</td></tr> <tr><td>Y</td><td>0.000 abs</td></tr> <tr><td>Y<sub>a</sub></td><td>abs</td></tr> <tr><td>Z</td><td>0.000 abs</td></tr> </table>	Par.surf.C	0.000	Y	0.000 abs	Y <sub>a</sub>	abs	Z	0.000 abs	Input of auxiliary commands (max.40 characters)
Par.surf.C	0.000									
Y	0.000 abs									
Y <sub>a</sub>	abs									
Z	0.000 abs									
Perimeter face C:										
Polar										
Ø		See Perimeter face C, Cartesian								
Y, Z	<p>Starting point</p> <table border="1"> <tr><td>Par.surf.C</td><td>0.000</td></tr> <tr><td>Pole:</td><td></td></tr> <tr><td>Y</td><td>0.000 abs</td></tr> <tr><td>Z</td><td>0.000 abs</td></tr> </table> <p>Starting point: L<sub>1</sub> 0.000 abs φ<sub>1</sub> 0.000 °abs</p>	Par.surf.C	0.000	Pole:		Y	0.000 abs	Z	0.000 abs	<p>Y: Position Pole in Y-direction [abs]</p> <p>Z: Position Pole in Z-direction [abs]</p>
Par.surf.C	0.000									
Pole:										
Y	0.000 abs									
Z	0.000 abs									
L <sub>1</sub> , φ <sub>1</sub> , Auxiliary commands		See Face C / Face Y, polar								
Perimeter face Y:										
Cartesian										
Y, Z	<p>Starting point</p> <table border="1"> <tr><td>Par.surf.Y</td><td></td></tr> <tr><td>Y</td><td>0.000 abs</td></tr> <tr><td>Z</td><td>0.000 abs</td></tr> </table>	Par.surf.Y		Y	0.000 abs	Z	0.000 abs	<p>Y: Start point in Y-direction [abs]</p> <p>Z: Start point in Z-direction [abs]</p>		
Par.surf.Y										
Y	0.000 abs									
Z	0.000 abs									
Auxiliary commands		See Face C / Face Y, Cartesian								

## Section 3

### Starting a contour

Notes

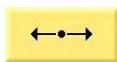
Parameter	Help picture	Description and hints
Perimeter face Y:		
Polar		
Y, Z  Starting point Per. surf. Y  Pole: Y 0.000 abs Z 0.000 abs  Starting point: L1 0.000 abs φ1 0.000 °abs		Y: Position Pole in Y-direction Z: Position Pole in Z-direction
L1, φ1, Auxiliary commands		See Face C / Face Y, polar

After a new contour has been started and the Start point has been specified, the individual contour elements, of which the contour is made up, can be defined.

The following contour elements are available for the definition of a contour:



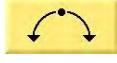
Straight vertical



Straight horizontal



Straight diagonal



Circle/circular arc

**Hints:**

The co-ordinates for a horizontal or vertical straights must be entered as Cartesian co-ordinates. In case of the contour elements straight diagonal and circle/circular arc, however, it is possible to choose between Cartesian and polar co-ordinates.

If polar co-ordinates are used, a pole must be defined beforehand. If the pole is already defined for the Start point the polar co-ordinates can also be referred to this pole; i.e. no further pole must be defined.

## Section 4

### Entering of contour elements

#### Parameter input

If in some of the fields no values have been entered, ShopTurn assumes that these values are unknown and will attempt to evaluate these from other parameter.

In case of contours, where more parameters are entered than absolutely necessary, this can lead to controversies. In this case try to enter fewer parameters and to let ShopTurn as many parameters as possible.

Notes

#### Machining direction

In case of continuous path milling the contour will always be machined in the programmed direction. Programming the contour clockwise or anti-clockwise determines whether the contour is machined by the climb milling or conventional milling method.

External contour		
Required machining rotation	Spindle rotation direction right	Spindle rotation direction left
Climb milling	Programming clockwise, Cutter radius correction left	Programming anti-clockwise, Cutter radius correction right
Conventional milling	Programming anti-clockwise, Cutter radius correction right	Programming clockwise, Cutter radius correction left
Internal contour		
Required machining rotation	Spindle rotation direction right	Spindle rotation direction left
Climb milling	Programming anticlockwise, Cutter radius correction left	Programming clockwise, Cutter radius correction right
Conventional milling	Programming clockwise, Cutter radius correction right	Programming anti-clockwise, Cutter radius correction left

## Section 4

Notes

### Entering of contour elements

#### Contour transition elements

Between any two contour elements a transition element can be inserted. This can be either a:  
radius or  
a chamfer

The transition element is always added at the end of a contour element. The selection of a contour transition element is made in the parameter mask of the respective contour element.

A contour transition element can always be used provided there is an intersection point of the two adjoining elements that can be calculated from the input values. Failing this the contour elements straight/Circle must be used (e.g. beginning or end of an open contour).

In case of a closed contour it's also possible to programme a transition element from the last to the first element of the contour. After programming the contour transition element the Start point lies in this case outside the contour.

#### Auxiliary commands

Any sort of auxiliary command in the form of G-code can be entered for the contour element. For example: for das contour element Circle "G9" speed reduction and a defined spindle stop can be programmed.

The auxiliary commands (max. 40 characters) must be entered in the extended parameter mask (Softkey "All parameters").

#### Further functions

When programming a contour the following further functions are available:

Tangential to the previous element

The transition from the previous element can be programmed as a tangent.

Dialogue selection

If two possible solutions result from the parameters entered so far, one of them must be selected.

Closing a contour

The contour can be closed with a straight connecting the current position to the Start point.

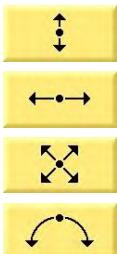
## Section 4

### Entering of contour elements

#### Entering of contour elements

After the Start point has been accepted, the contour elements can be entered. The respective vertical Softkeys are available for this.

Notes



A contour element can be selected by pressing the respective vertical Softkey .

Enter all the data that can be taken from the work piece drawing (e.g. length of a straight, end positions, transition to the next element, pitch angle etc.).



By pressing the VSK 8 "Accept." transfer the contour element to the contour.

Repeat this procedure until the contour is completed.



By pressing the VSK 8 "Accept." transfer the contour to the work plan.



By pressing the VSK 5 "All parameters" further parameters can be shown for individual contour elements, e.g. for entering auxiliary commands.

#### Definition of a pole

If the contour elements "Straight diagonal" and "Circle/circular arc" are to be entered in polar coordinates a pole must be defined beforehand.



Select the input mask for the pole definition by pressing the VSK 6 "More" and VSK 3 "Pole".

Enter the co-ordinates for the pole.



By pressing the VSK 8 "Accept." transfer the pole to the contour.

Now it is possible to toggle between "Cartesian" and "Polar" in the input mask for the contour elements "Straight diagonal" and "Circle/circular arc".

Hint:

After the definition of the pole only the contour element "Straight diagonal" is available. Subsequently the other contour elements can be used once more.

## Section 4

### Entering of contour elements

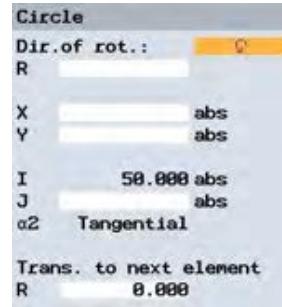
Notes

#### Tangent to previous element

Tangent prev.elem

By pressing the VSK 2 "Tangent prev.elem" the transition to the previous element can be programmed to be tangential.

The angle to the previous element 2 is set to "0°". The selection "tangential" appears in the input field of the parameter.

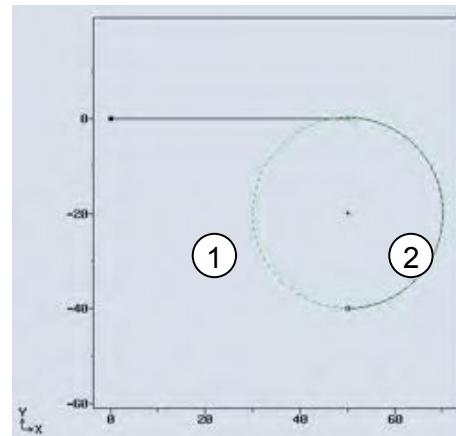


#### Dialogue selection

Dialog select

During data input for a contour element there might result two contour possibilities of which one must be selected.

By pressing the VSK 1 "Dialog select" it is possible to select one of the different possibilities.



Example: Circle: Alternative contour possibilities

- ① Alternative 1 (alternative contour element)
- ② Alternative 2 (programmed contour element)

Dialog accept

By pressing the VSK 8 "Dialog accept" the selected contour possibility (black contour element) will be transferred.

Following this further contour elements can be entered or the contour element can be transferred to the contour.

## Section 4

### Entering of contour elements

#### Closing a contour

For contour pockets, spigots and islands a contour must always be closed. In case of continuous path milling the contour can remain open.

More

Close contour

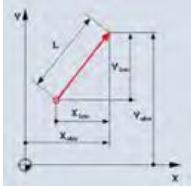
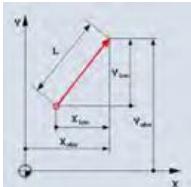
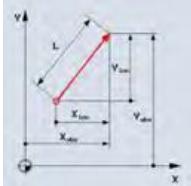
By pressing the VSK 6 "More" and thereafter the VSK 6 "Close contour" the contour will be closed automatically by a straight from the current position to the Start point.

A transition element (chamfer/radius) can also be added to this straight.

Notes

Shown in the following table is a summary of the parameters for the contour elements "Straight in X-direction", "Straight in Y-direction" and "Straight diagonal". The display, however, only shows the relevant parameters for the respective straight.

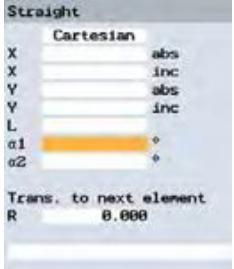
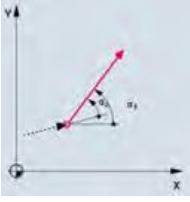
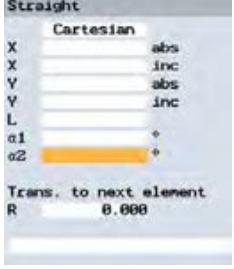
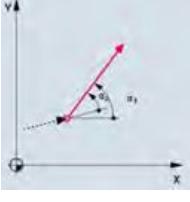
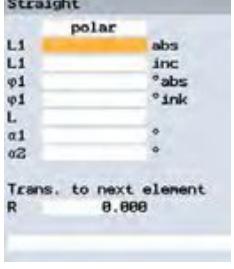
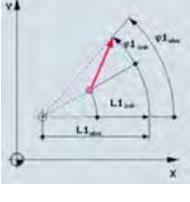
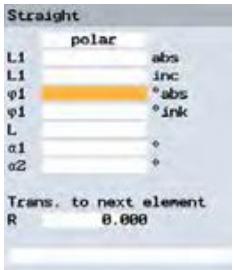
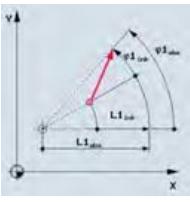
Parameter for contour element "Straight diagonal"; VSK "All parameters" pressed:

Parameter	Help picture	Description and hints															
Face C / Face Y:																	
Cartesian																	
X	<p><b>Straight</b></p> <table border="1" style="margin-left: 10px; margin-bottom: 10px;"> <tr><td>Cartesian</td></tr> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>L</td><td></td></tr> <tr><td><math>\alpha_1</math></td><td>°</td></tr> <tr><td><math>\alpha_2</math></td><td>°</td></tr> </table> <p>Trans. to next element R 0.000</p>	Cartesian	X	abs	X	inc	Y	abs	Y	inc	L		$\alpha_1$	°	$\alpha_2$	°	<p>End point in X-direction</p> <p>[abs, inc]</p> <p>Incremental dimension: The sign is taken into account.</p> 
Cartesian																	
X	abs																
X	inc																
Y	abs																
Y	inc																
L																	
$\alpha_1$	°																
$\alpha_2$	°																
Y	<p><b>Straight</b></p> <table border="1" style="margin-left: 10px; margin-bottom: 10px;"> <tr><td>Cartesian</td></tr> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>L</td><td></td></tr> <tr><td><math>\alpha_1</math></td><td>°</td></tr> <tr><td><math>\alpha_2</math></td><td>°</td></tr> </table> <p>Trans. to next element R 0.000</p>	Cartesian	X	abs	X	inc	Y	abs	Y	inc	L		$\alpha_1$	°	$\alpha_2$	°	<p>End point in Y-direction</p> <p>[abs, inc]</p> <p>Incremental dimension: The sign is taken into account.</p> 
Cartesian																	
X	abs																
X	inc																
Y	abs																
Y	inc																
L																	
$\alpha_1$	°																
$\alpha_2$	°																
L	<p><b>Straight</b></p> <table border="1" style="margin-left: 10px; margin-bottom: 10px;"> <tr><td>Cartesian</td></tr> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>L</td><td>mm</td></tr> <tr><td><math>\alpha_1</math></td><td>°</td></tr> <tr><td><math>\alpha_2</math></td><td>°</td></tr> </table> <p>Trans. to next element R 0.000</p>	Cartesian	X	abs	X	inc	Y	abs	Y	inc	L	mm	$\alpha_1$	°	$\alpha_2$	°	<p>Length of the straight</p> <p>[mm]</p> 
Cartesian																	
X	abs																
X	inc																
Y	abs																
Y	inc																
L	mm																
$\alpha_1$	°																
$\alpha_2$	°																

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Face C / Face Y:		
Cartesian		
1 		Start angle to the X-Axis [Degrees]
2 		Angle to the previous element [Degrees] Tangential transition: 2 = 0
Face C / Face Y:		
Polar		
L1 		abs: Distance to the pole, end point inc: Change of distance to the pole, end point Incremental dimension: The sign is taken into account.
phi1 		abs: Polar angle at the pole and end point inc: Change of polar angle at the pole, end point [Degrees] Incremental dimension: The sign is taken into account.

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints														
Face C / Face Y:																
polar																
L	<p><b>Straight</b> polar</p> <table border="1"> <tr><td>L1</td><td>abs</td></tr> <tr><td>L1</td><td>inc</td></tr> <tr><td>φ1</td><td>*abs</td></tr> <tr><td>φ1</td><td>*ink</td></tr> <tr><td>L</td><td>o</td></tr> <tr><td>α1</td><td>o</td></tr> <tr><td>α2</td><td>o</td></tr> </table> <p>Trans. to next element R 0.000</p>	L1	abs	L1	inc	φ1	*abs	φ1	*ink	L	o	α1	o	α2	o	Length of the straight [mm]
L1	abs															
L1	inc															
φ1	*abs															
φ1	*ink															
L	o															
α1	o															
α2	o															
α1	<p><b>Straight</b> polar</p> <table border="1"> <tr><td>L1</td><td>abs</td></tr> <tr><td>L1</td><td>inc</td></tr> <tr><td>φ1</td><td>*abs</td></tr> <tr><td>φ1</td><td>*ink</td></tr> <tr><td>L</td><td>o</td></tr> <tr><td>α1</td><td>o</td></tr> <tr><td>α2</td><td>o</td></tr> </table> <p>Trans. to next element R 0.000</p>	L1	abs	L1	inc	φ1	*abs	φ1	*ink	L	o	α1	o	α2	o	Start angle to the X-Axis [Degrees]
L1	abs															
L1	inc															
φ1	*abs															
φ1	*ink															
L	o															
α1	o															
α2	o															
α2	<p><b>Straight</b> polar</p> <table border="1"> <tr><td>L1</td><td>abs</td></tr> <tr><td>L1</td><td>inc</td></tr> <tr><td>φ1</td><td>*abs</td></tr> <tr><td>φ1</td><td>*ink</td></tr> <tr><td>L</td><td>o</td></tr> <tr><td>α1</td><td>o</td></tr> <tr><td>α2</td><td>o</td></tr> </table> <p>Trans. to next element R 0.000</p>	L1	abs	L1	inc	φ1	*abs	φ1	*ink	L	o	α1	o	α2	o	Angle to the previous element [Degrees]  Tangential transition: 2 = 0
L1	abs															
L1	inc															
φ1	*abs															
φ1	*ink															
L	o															
α1	o															
α2	o															

Perimeter face C:																				
Cartesian																				
Y	<p><b>Straight</b> Cartesian</p> <table border="1"> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>Ya</td><td>abs</td></tr> <tr><td>Ya</td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td>o</td></tr> <tr><td>α1</td><td>o</td></tr> <tr><td>α2</td><td>o</td></tr> </table> <p>Trans. to next element R 0.000</p>	Y	abs	Y	inc	Ya	abs	Ya	inc	Z	abs	Z	inc	L	o	α1	o	α2	o	End point in Y-direction - replaces the input value in Ya-input field [abs, inc]  Incremental dimension: The sign will be taken into account.
Y	abs																			
Y	inc																			
Ya	abs																			
Ya	inc																			
Z	abs																			
Z	inc																			
L	o																			
α1	o																			
α2	o																			

## Section 4

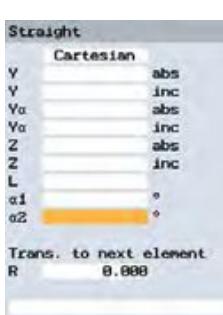
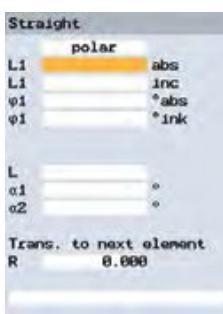
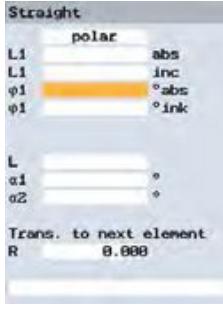
### Entering of contour elements

Notes

Parameter	Help picture	Description and hints																						
Perimeter face C:																								
Cartesian																								
Y <sub>a</sub>	<p><b>Straight</b> Cartesian</p> <table border="1"> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>Y<sub>a</sub></td><td>abs</td></tr> <tr><td>Y<sub>a</sub></td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td></td></tr> <tr><td>α<sub>1</sub></td><td>°</td></tr> <tr><td>α<sub>2</sub></td><td>°</td></tr> <tr><td colspan="2">Trans. to next element</td></tr> <tr><td>R</td><td>0.000</td></tr> </table>	Y	abs	Y	inc	Y <sub>a</sub>	abs	Y <sub>a</sub>	inc	Z	abs	Z	inc	L		α <sub>1</sub>	°	α <sub>2</sub>	°	Trans. to next element		R	0.000	End angle - can be entered only if Y-input field is empty. [abs, inc] Incremental dimension: The sign will be taken into account.
Y	abs																							
Y	inc																							
Y <sub>a</sub>	abs																							
Y <sub>a</sub>	inc																							
Z	abs																							
Z	inc																							
L																								
α <sub>1</sub>	°																							
α <sub>2</sub>	°																							
Trans. to next element																								
R	0.000																							
Z	<p><b>Straight</b> Cartesian</p> <table border="1"> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>Y<sub>a</sub></td><td>abs</td></tr> <tr><td>Y<sub>a</sub></td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td></td></tr> <tr><td>α<sub>1</sub></td><td>°</td></tr> <tr><td>α<sub>2</sub></td><td>°</td></tr> <tr><td colspan="2">Trans. to next element</td></tr> <tr><td>R</td><td>0.000</td></tr> </table>	Y	abs	Y	inc	Y <sub>a</sub>	abs	Y <sub>a</sub>	inc	Z	abs	Z	inc	L		α <sub>1</sub>	°	α <sub>2</sub>	°	Trans. to next element		R	0.000	End point in Z-direction [abs, inc] Incremental dimension: The sign is taken into account.
Y	abs																							
Y	inc																							
Y <sub>a</sub>	abs																							
Y <sub>a</sub>	inc																							
Z	abs																							
Z	inc																							
L																								
α <sub>1</sub>	°																							
α <sub>2</sub>	°																							
Trans. to next element																								
R	0.000																							
L	<p><b>Straight</b> Cartesian</p> <table border="1"> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>Y<sub>a</sub></td><td>abs</td></tr> <tr><td>Y<sub>a</sub></td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td>abs</td></tr> <tr><td>α<sub>1</sub></td><td>°</td></tr> <tr><td>α<sub>2</sub></td><td>°</td></tr> <tr><td colspan="2">Trans. to next element</td></tr> <tr><td>R</td><td>0.000</td></tr> </table>	Y	abs	Y	inc	Y <sub>a</sub>	abs	Y <sub>a</sub>	inc	Z	abs	Z	inc	L	abs	α <sub>1</sub>	°	α <sub>2</sub>	°	Trans. to next element		R	0.000	Length of the straight [mm]
Y	abs																							
Y	inc																							
Y <sub>a</sub>	abs																							
Y <sub>a</sub>	inc																							
Z	abs																							
Z	inc																							
L	abs																							
α <sub>1</sub>	°																							
α <sub>2</sub>	°																							
Trans. to next element																								
R	0.000																							
α <sub>1</sub>	<p><b>Straight</b> Cartesian</p> <table border="1"> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>Y<sub>a</sub></td><td>abs</td></tr> <tr><td>Y<sub>a</sub></td><td>inc</td></tr> <tr><td>Z</td><td>abs</td></tr> <tr><td>Z</td><td>inc</td></tr> <tr><td>L</td><td></td></tr> <tr><td>α<sub>1</sub></td><td>°</td></tr> <tr><td>α<sub>2</sub></td><td>°</td></tr> <tr><td colspan="2">Trans. to next element</td></tr> <tr><td>R</td><td>0.000</td></tr> </table>	Y	abs	Y	inc	Y <sub>a</sub>	abs	Y <sub>a</sub>	inc	Z	abs	Z	inc	L		α <sub>1</sub>	°	α <sub>2</sub>	°	Trans. to next element		R	0.000	Start angle to the Y-Axis [Degrees]
Y	abs																							
Y	inc																							
Y <sub>a</sub>	abs																							
Y <sub>a</sub>	inc																							
Z	abs																							
Z	inc																							
L																								
α <sub>1</sub>	°																							
α <sub>2</sub>	°																							
Trans. to next element																								
R	0.000																							

## Section 4

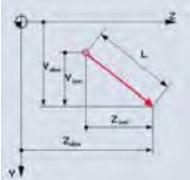
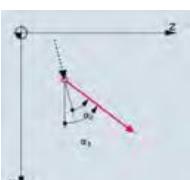
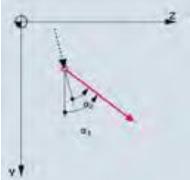
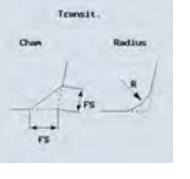
### Entering of contour elements

Parameter	Help picture	Description and hints	Notes
Perimeter face C: Cartesian  α2		Angle to the previous element [Degrees]  Tangential transition: $2 = 0$	
Perimeter face Y: Cartesian  Y, Z, L, α1, α2		See Perimeter face C, Cartesian	
Perimeter face C / Perimeter face Y: Polar  L1		abs: Distance to the pole, end point  inc: Change of distance to the pole, end point  Incremental dimension: The sign is taken into account.	
φ1		abs: Polar angle at the pole and end point  inc: Change of polar angle at the pole, end point [Degrees]  Incremental dimension: The sign is taken into account.	

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Perimeter face C / Perimeter face Y:		
Polar		
L		Length of the straight [mm]
a1		Start angle to the Y-Axis [Degrees]
a2		Angle to the previous element [Degrees] Tangential transition: 2 = 0
General parameters		
FS or R		Transition to the previous element FS: Chamfer R: Radius [mm]

## Section 4

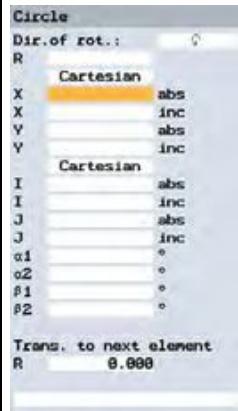
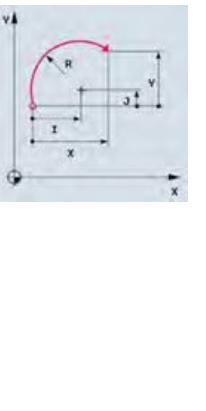
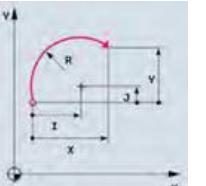
### Entering of contour elements

Parameter	Help picture	Description and hints	Notes																														
General parameters																																	
Auxiliary commands	<p><b>Straight</b></p> <table border="1"> <tr><td>Cartesian</td><td></td></tr> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>L</td><td></td></tr> <tr><td>a1</td><td>°</td></tr> <tr><td>a2</td><td>°</td></tr> </table> <p>Trans. to next element R 0.000</p>	Cartesian		X	abs	X	inc	Y	abs	Y	inc	L		a1	°	a2	°	Input of auxiliary commands in the form of G-Code (max.40 characters)															
Cartesian																																	
X	abs																																
X	inc																																
Y	abs																																
Y	inc																																
L																																	
a1	°																																
a2	°																																
Parameter for the contour element "Circle"; VSK "All parameters" pressed:																																	
Direction of rotation	<p><b>Circle</b></p> <table border="1"> <tr><td>Dir.of rot.:</td><td>?</td></tr> <tr><td>R</td><td></td></tr> <tr><td>Cartesian</td><td></td></tr> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>I</td><td>abs</td></tr> <tr><td>I</td><td>inc</td></tr> <tr><td>J</td><td>abs</td></tr> <tr><td>J</td><td>inc</td></tr> <tr><td>a1</td><td>°</td></tr> <tr><td>a2</td><td>°</td></tr> <tr><td>β1</td><td>°</td></tr> <tr><td>β2</td><td>°</td></tr> </table> <p>Trans. to next element R 0.000</p>	Dir.of rot.:	?	R		Cartesian		X	abs	X	inc	Y	abs	Y	inc	I	abs	I	inc	J	abs	J	inc	a1	°	a2	°	β1	°	β2	°	Rotation clockwise Rotation anti-clockwise	
Dir.of rot.:	?																																
R																																	
Cartesian																																	
X	abs																																
X	inc																																
Y	abs																																
Y	inc																																
I	abs																																
I	inc																																
J	abs																																
J	inc																																
a1	°																																
a2	°																																
β1	°																																
β2	°																																
R	<p><b>Circle</b></p> <table border="1"> <tr><td>Dir.of rot.:</td><td>?</td></tr> <tr><td>R</td><td></td></tr> <tr><td>Cartesian</td><td></td></tr> <tr><td>X</td><td>abs</td></tr> <tr><td>X</td><td>inc</td></tr> <tr><td>Y</td><td>abs</td></tr> <tr><td>Y</td><td>inc</td></tr> <tr><td>I</td><td>abs</td></tr> <tr><td>I</td><td>inc</td></tr> <tr><td>J</td><td>abs</td></tr> <tr><td>J</td><td>inc</td></tr> <tr><td>a1</td><td>°</td></tr> <tr><td>a2</td><td>°</td></tr> <tr><td>β1</td><td>°</td></tr> <tr><td>β2</td><td>°</td></tr> </table> <p>Trans. to next element R 0.000</p>	Dir.of rot.:	?	R		Cartesian		X	abs	X	inc	Y	abs	Y	inc	I	abs	I	inc	J	abs	J	inc	a1	°	a2	°	β1	°	β2	°	Radius of the circle [mm]	
Dir.of rot.:	?																																
R																																	
Cartesian																																	
X	abs																																
X	inc																																
Y	abs																																
Y	inc																																
I	abs																																
I	inc																																
J	abs																																
J	inc																																
a1	°																																
a2	°																																
β1	°																																
β2	°																																

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Face C / Face Y:		
Cartesian		
X		End point in X-direction [abs, inc] Incremental dimension: The sign is taken into account.
Y		End point in Y-direction [abs, inc] Incremental dimension: The sign is taken into account.
I		Circle centre point in X-direction [abs, inc] Incremental dimension: The sign is taken into account.

## Section 4

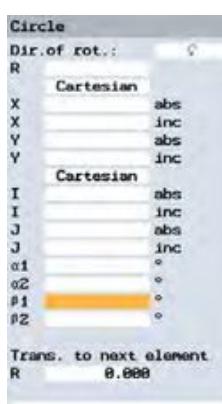
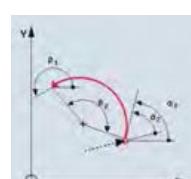
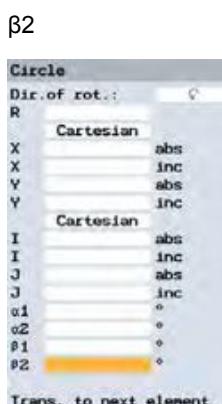
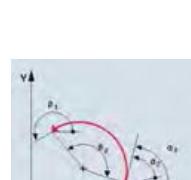
### Entering of contour elements

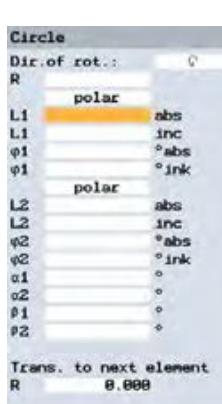
Parameter	Help picture	Description and hints	Notes
Face C / Face Y:			
Cartesian			
J	<p><b>Circle</b> Dir.of rot.: <input checked="" type="radio"/> R R      Cartesian X      abs X      inc Y      abs Y      inc I      Cartesian I      abs I      inc J      abs J      inc α1      ° α2      ° β1      ° β2      °  Trans. to next element R      0.000</p>	<p>Circle centre point in Y-direction [abs, inc]</p> <p>Incremental dimension: The sign is taken into account.</p>	
α1	<p><b>Circle</b> Dir.of rot.: <input checked="" type="radio"/> R R      Cartesian X      abs X      inc Y      abs Y      inc I      Cartesian I      abs I      inc J      abs J      inc α1      ° α2      ° β1      ° β2      °  Trans. to next element R      0.000</p>	<p>Start angle to the X-Axis [Degrees]</p>	
α2	<p><b>Circle</b> Dir.of rot.: <input checked="" type="radio"/> R R      Cartesian X      abs X      inc Y      abs Y      inc I      Cartesian I      abs I      inc J      abs J      inc α1      ° α2      ° β1      ° β2      °  Trans. to next element R      0.000</p>	<p>Angle to the previous element [Degrees]</p> <p>Tangential transition: <math>2 = 0</math></p>	

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Face C / Face Y:		
Cartesian		
<b>β1</b> 		End angle to the X-Axis [Degrees]
<b>β2</b> 		Opening angle of the circle [Degrees]

Face C / Face Y:		
Polar		
<b>L1</b> 		
abs: Distance to the pole, end point inc: Change of distance to the pole, end point Incremental dimension: The sign is taken into account.		

## Section 4

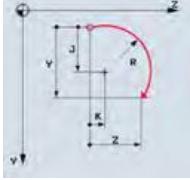
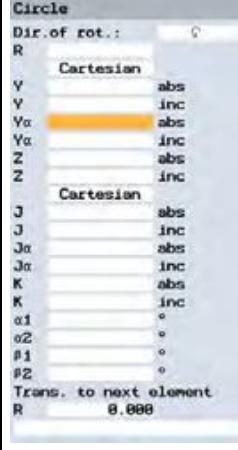
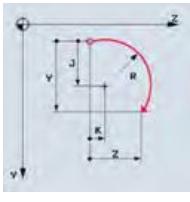
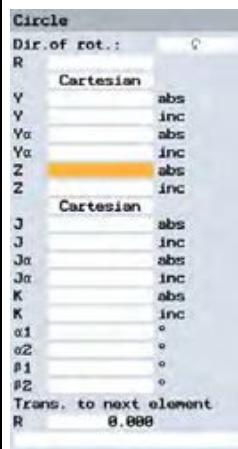
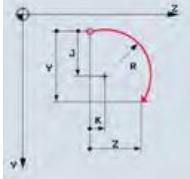
### Entering of contour elements

Parameter	Help picture	Description and hints	Notes
Face C / Face Y:			
Polar			
φ1	<p><b>Circle</b> Dir.of rot.: <math>\odot</math> R   polar L1      abs L1      inc φ1      °abs φ1      °ink   polar L2      abs L2      inc φ2      °abs φ2      °ink α1      ° α2      ° β1      ° β2      °</p> <p>Trans. to next element R      0.000</p>	<p>abs: Polar angle to the pole, end point inc: Change of polar angle at the pole, end point Incremental dimension: The sign is taken into account.</p>	
L2	<p><b>Circle</b> Dir.of rot.: <math>\odot</math> R   polar L1      abs L1      inc φ1      °abs φ1      °ink   polar L2      abs L2      inc φ2      °abs φ2      °ink α1      ° α2      ° β1      ° β2      °</p> <p>Trans. to next element R      0.000</p>	<p>abs: Distance to pole, circle centre point inc: Change of distance to the pole, circle centre point Incremental dimension: The sign is taken into account.</p>	
φ2	<p><b>Circle</b> Dir.of rot.: <math>\odot</math> R   polar L1      abs L1      inc φ1      °abs φ1      °ink   polar L2      abs L2      inc φ2      °abs φ2      °ink α1      ° α2      ° β1      ° β2      °</p> <p>Trans. to next element R      0.000</p>	<p>abs: Polar angle to the pole, circle centre point inc: Change of polar angle at the pole, circle centre point Incremental dimension: The sign is taken into account.</p>	
α1, α2, β1, β2		See Face C / Face Y, Cartesian	

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Perimeter face C:		
Cartesian		
<b>Y</b> 		End point in Y-direction - replaces the input value in Ya-input field [abs, inc] Incremental dimension: The sign will be taken into account.
<b>Ya</b> 		End angle - can be entered only if Y-input field is empty [abs, inc] Incremental dimension: The sign will be taken into account.
<b>Z</b> 		End point in Z-direction [abs, inc] Incremental dimension: The sign is taken into account.

## Section 4

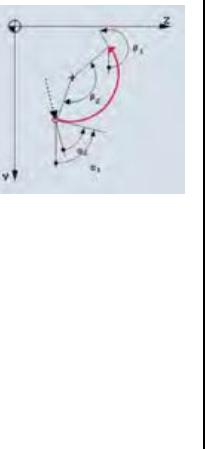
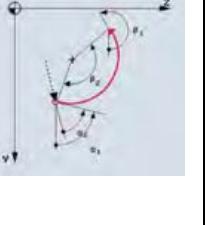
### Entering of contour elements

Parameter	Help picture	Description and hints	Notes
Perimeter face C:			
Cartesian			
J	<p><b>Circle</b> Dir.of rot.: <input checked="" type="radio"/> R R      Cartesian Y      abs Y      inc Y<sub>x</sub>      abs Y<sub>x</sub>      inc Z      abs Z      inc Cartesian J      abs J      inc Ja      abs Ja      inc K      abs K      inc α<sub>1</sub>      ° α<sub>2</sub>      ° β<sub>1</sub>      ° β<sub>2</sub>      ° Trans. to next element R      8.000</p>	<p>Circle centre point in Y-direction [abs, inc]</p> <p>Incremental dimension: The sign is taken into account.</p>	
Ja	<p><b>Circle</b> Dir.of rot.: <input checked="" type="radio"/> R R      Cartesian Y      abs Y      inc Y<sub>x</sub>      abs Y<sub>x</sub>      inc Z      abs Z      inc Cartesian J      abs J      inc Ja      abs Ja      inc K      abs K      inc α<sub>1</sub>      ° α<sub>2</sub>      ° β<sub>1</sub>      ° β<sub>2</sub>      ° Trans. to next element R      8.000</p>	<p>Angle circle centre point in Y-direction [abs, inc]</p> <p>Incremental dimension: The sign will be taken into account.</p>	
K	<p><b>Circle</b> Dir.of rot.: <input checked="" type="radio"/> R R      Cartesian Y      abs Y      inc Y<sub>x</sub>      abs Y<sub>x</sub>      inc Z      abs Z      inc Cartesian J      abs J      inc Ja      abs Ja      inc K      abs K      inc α<sub>1</sub>      ° α<sub>2</sub>      ° β<sub>1</sub>      ° β<sub>2</sub>      ° Trans. to next element R      8.000</p>	<p>Circle centre point in Z-direction [abs, inc]</p> <p>Incremental dimension: The sign is taken into account.</p>	

## Section 4

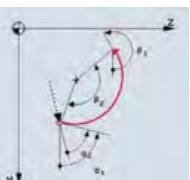
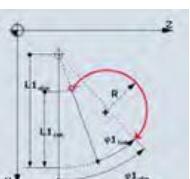
### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Perimeter face C:		
Cartesian		
<b>a1</b>		Start angle to the Y-Axis [Degrees]
<b>a2</b>		Angle to the previous element [Degrees] Tangential transition: $2 = 0$
<b>b1</b>		End angle to the Y-Axis [Degrees]

## Section 4

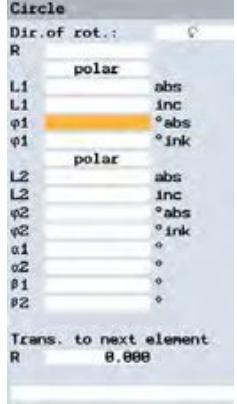
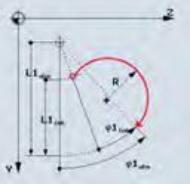
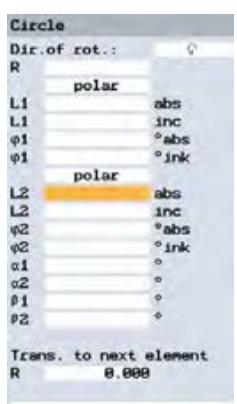
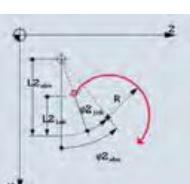
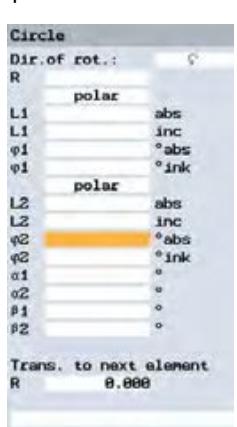
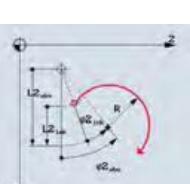
### Entering of contour elements

Parameter	Help picture	Description and hints	Notes
Perimeter face C:			
Cartesian			
$\beta_2$		Opening angle of the circle [Degrees]	
<b>Circle</b> Dir. of rot.: $\hat{z}$ R Cartesian Y abs Y inc Y $\alpha$ abs Y $\alpha$ inc Z abs Z inc Cartesian J abs J inc J $\alpha$ abs J $\alpha$ inc K abs K inc $\alpha_1$ o $\alpha_2$ o $\beta_1$ o $\beta_2$ o Trans. to next element R 0.000			
Perimeter face Y:			
kartesisch			
Y, Z, J, K, $\alpha_1, \alpha_2, \beta_1, \beta_2$		See Perimeter face C, Cartesian	
Perimeter face C / Perimeter face Y:			
Polar			
L1		abs: Distance to the pole, end point inc: Change of distance to the pole, end point Incremental dimension: The sign is taken into account.	
<b>Circle</b> Dir. of rot.: $\hat{z}$ R polar L1 abs L1 inc $\alpha_1$ °abs $\alpha_1$ °ink polar L2 abs L2 inc $\alpha_2$ °abs $\alpha_2$ °ink $\alpha_1$ o $\alpha_2$ o $\beta_1$ o $\beta_2$ o Trans. to next element R 0.000			

## Section 4

### Entering of contour elements

Notes

Parameter	Help picture	Description and hints
Perimeter face C / Perimeter face Y:		
Polar		
<b>φ1</b> 	 <p>abs: Polar angle at the pole and end point inc: Change of polar angle at the pole, end point [Degrees] Incremental dimension: The sign is taken into account.</p>	
<b>L2</b> 	 <p>abs: Distance to pole, circle centre point inc: Change of distance to the pole, circle centre point Incremental dimension: The sign is taken into account.</p>	
<b>φ2</b> 	 <p>abs: Polar angle to the pole, circle centre point inc: Change of polar angle at the pole, circle centre point Incremental dimension: The sign is taken into account.</p>	
<b>α1, α2, β1, β2</b>		See Perimeter face C, Cartesian

## Section 4

### Entering of contour elements

Parameter	Help picture	Description and hints	Notes
General parameters			
<p>FS or R</p> <pre> <b>Circle</b> Dir.of rot.:  R   polar L1      abs L1      inc φ1      *abs φ1      *ink   polar L2      abs L2      inc φ2      *abs φ2      *ink α1      * α2      * β1      * β2      * </pre> <p>Trans. to next element R <b>0.000</b></p>		<p>Transition to the previous element</p> <p>FS: Chamfer</p> <p>R: Radius</p> <p>[mm]</p>	
<p>Auxiliary commands</p> <pre> <b>Circle</b> Dir.of rot.:  R   polar L1      abs L1      inc φ1      *abs φ1      *ink   polar L2      abs L2      inc φ2      *abs φ2      *ink α1      * α2      * β1      * β2      * </pre> <p>Trans. to next element R <b>0.000</b></p>		<p>Input of auxiliary commands in the form of G-Code (max.40 characters)</p>	

## Section 5

Notes

### Alteration of contour elements

An existing contour can be changed in retrospect. Individual elements can be:

- added
- altered
- inserted or
- deleted

Hint:

If there are two contours by the same name defined in one program, then the alterations to one contour will automatically be carried out in the other contour as well.

#### 5.1 Adding a contour element



Select the contour in the work plan.

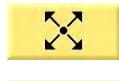


Open the contour.

The individual contour elements will be listed.



Place the cursor on the last contour element before the end.



Select the respective contour element by means of the vertical Softkey-strip, e.g. "Straight" or „Circle/circular arc“.



Enter the parameters in the input mask.



Press the VSK "Accept.".

The contour element is tagged onto the contour

## Section 5

### Alteration of contour elements

#### 5.2 Alteration of a contour element



Pre-select the contour in the work plan



Open the contour.

The individual contour elements will be listed.



Set the cursor on the contour element, that is to be altered.



By pressing the key "Cursor right" the associated input mask is opened and the selected element will be shown enlarged in the programming graphics.

Enter the required alterations.



Press the VSK "Accept.".

The current values of the contour element will be transferred and the alterations will be shown in the programming graphics.

#### 5.3 Changing the dialogue selection

If during the data input for a contour elements two differing possibilities result and the incorrect alternative was selected, this selection can be changed in retrospect. If a definite contour resulted from other parameters no further dialogue selection appears.



Open the input mask for the contour element.



Press the VSK "Change selection".

Both contour possibilities are shown.



By pressing the VSK "Dialog select" it is possible to toggle between the two alternatives.



By pressing the VSK "Dialog accept" the selected contour possibility will be transferred to the contour.

Notes

## Section 5

Notes

### Alteration of contour elements

#### 5.4 Insertion of a contour element



Pre-select the contour in the work plan



Open the contour.

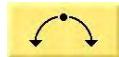
The individual contour elements will be listed.



Set the cursor on the contour element, after which the new element is to be inserted.



Select the new contour element by means of the vertical Softkey-strip, e.g. "Straight" or "Circle/circular arc".



Enter the parameters in the input mask.



Press the VSK "Accept".



The new contour element is transferred to the contour.

By selection of the next contour element the following element will be re-evaluated and displayed accordingly in the graphics window.

The status of these contour elements changes from a red to a black foreground.

Possibly the end point of the inserted element does not suit the Start point of the following element. In this case ShopTurn shows the error message "Geometrical values contradictory". If this contradiction is to be remedied, an oblique must be inserted without stating any values.

## Section 5

### Alteration of contour elements

#### 5.4 Deletion of a contour element



Select the contour in the work plan.



Open the contour.

The individual contour elements will be listed.



Set the cursor on the contour element, that is to be deleted.



Press the VSK 1 “Delete element”.



Press the VSK 8 “Delete“.

The contour element will be deleted and the contour will be matched to the new situation and shown in the graphics window.

Notes

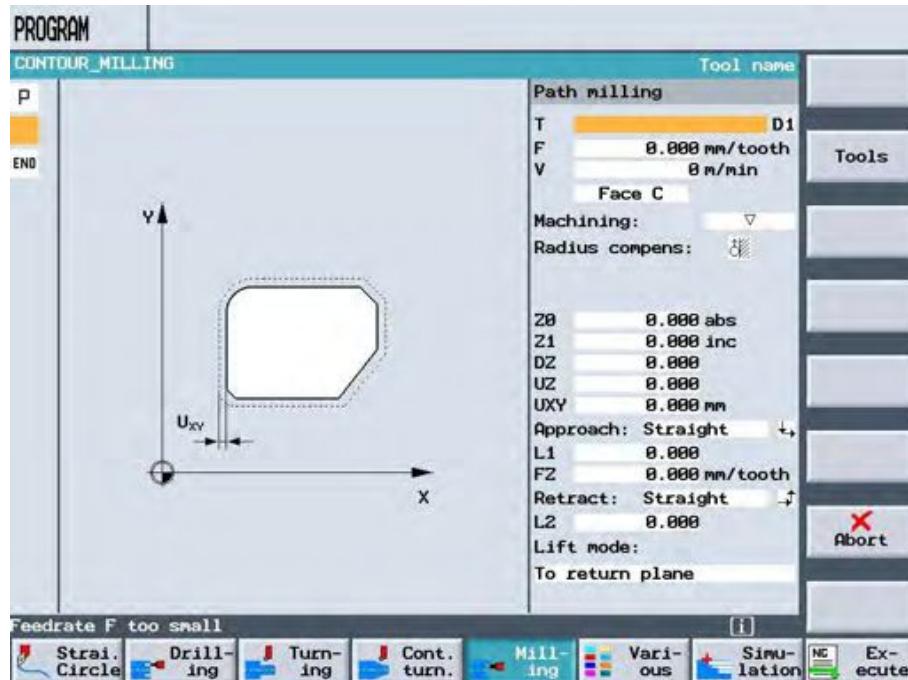
## Section 6

### Continuous path milling

Notes

Path  
milling

By pressing the VSK 2 "Path milling" the respective function for the machining is selected.



With the function "Continuous path milling" milling along any sort of programmed contour can be carried out.

The function "Continuous path milling" is characterized by the following features:

Machining is carried out using the cutter radius correction.

The contour does not have to be closed.

Internal or external machining (left or right of the contour) or with the cutter centre following the path.

Various approach and leaving strategies are available.

Machining can take place in or against the direction of the programmed contour.

For machining against the contour direction, the contours must not consist of more than 170 contour elements (incl. chamfers/radii).

Special features of the free G-code input (apart from feed rates) are not taken into account when machining against the contour direction.

## Section 6

### Continuous path milling

#### Approach/Leaving strategies

The contour can be approached or left with a quarter circle, semi-circle with a straight or vertically.

When using the quarter or semi circle the radius of the path of the cutter centre must be stated (only with radius correction).

In case of the straight the distance of the outer cutter edge to the contour start or end point respectively must be stated (only with radius correction).

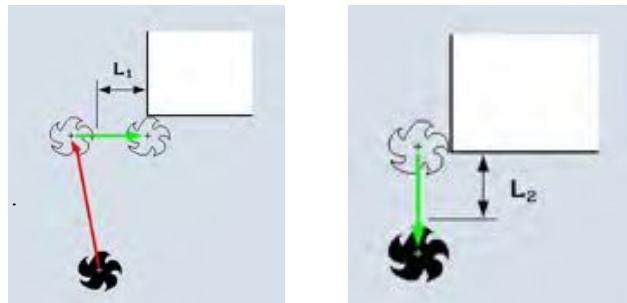
Mixed programming is possible, e.g. approach with a quarter circle, leaving with a semi circle

The approach in the plane takes place first in the Z-direction to depth and then in the XY-plane. Leaving takes place in the reverse order. In case of spatial approach/leaving the motion to depth takes place simultaneously with the motion in the plane.

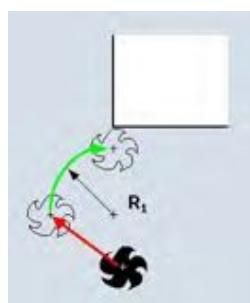
Different strategies can be selected for approach and leaving; e.g. Approach in the plane and leaving spatially.

- Motion with programmed feed rate
- Motion with rapid traverse

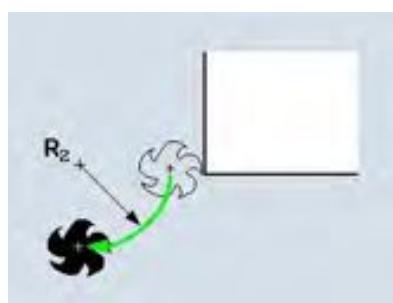
#### Approach/Leaving strategy straight



#### Approach strategy quarter circle



#### Leaving strategy quarter circle



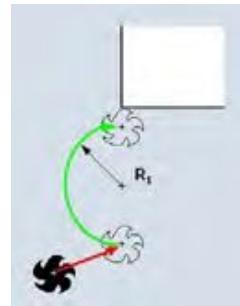
Notes

## Section 6

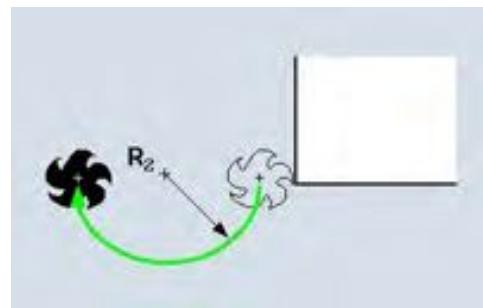
Notes

Continuous path milling

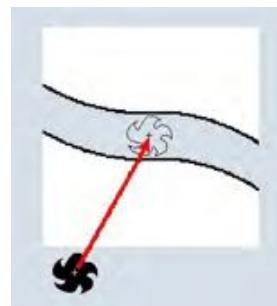
Approach strategy  
Semi circle



Leaving strategy  
Semi circle



Approach/Leaving  
strategy Vertical



**Path milling on the  
path of the cutter  
centre**

Radius compens:

A programmed contour can be machined with the path of the cutter centre if the radius correction has been disabled.

In this case the approach and leaving is possible with a straight or vertically. The vertical approach or leaving can be used for instance in case of closed contours.

## Section 6

### Continuous path milling

#### Slot wall correction

If a contour on the perimeter face (Machining plane Perimeter face C) is to be milled, machining with or without slot wall correction is possible.

##### Slot wall correction OFF

ShopTurn produces slots with parallel walls only if the tool diameter equals the slot width. The walls will not be parallel if the slot width is larger than the tool diameter.

##### Slot wall correction ON

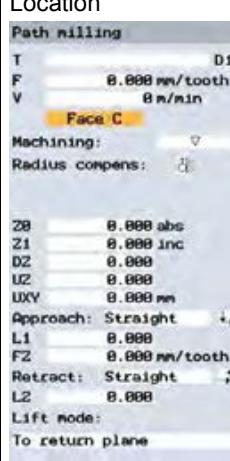
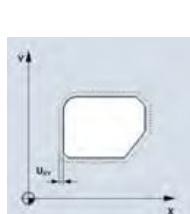
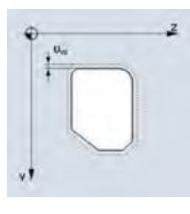
ShopTurn produces slots with parallel walls even if the slot is wider than the tool diameter

Notes

Hint:

If the slot is to be machined with the slot wall correction, the contour of the slot must not be programmed. Instead the centre path of a bolt, which would touch the slot on both walls, must be programmed. The slot width is determined by the parameter D.

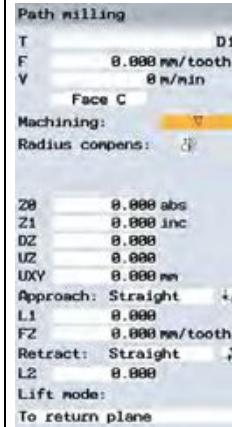
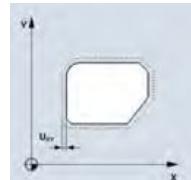
Parameters for the function „Continuous path milling“

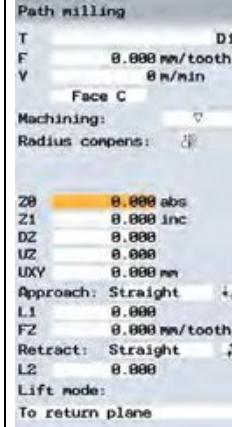
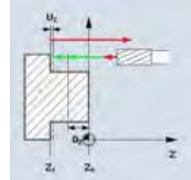
Parameter	Help picture	Description and hints
T, D, F, S / V		<i>See module A115 – “Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions“, Section 3 “Drilling centric“.</i>
Location 	 	<b>Face C</b> Face C  <b>Face Y</b> Face Y  <b>Per.surf.C</b> Perimeter face C  <b>Per.surf.Y</b> Perimeter face Y

## Section 6

### Continuous path milling

Notes

Parameter	Help picture	Description and hints
Type of machining  		Roughing
		Finishing

Face C:	
Z0  	Reference point in Z-direction [abs]  

## Section 6

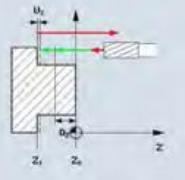
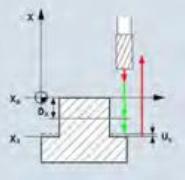
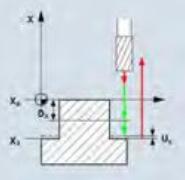
### Continuous path milling

Parameter	Help picture	Description and hints	Notes
Face C:			
Z1	<p><b>Path milling</b></p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C Machining: Radius compens: </p> <p>Z0 0.000 abs Z1 0.000 inc D2 0.000 UZ 0.000 UXY 0.000 mm Approach: Straight + L1 0.000 FZ 0.000 mm/tooth Retract: Straight ↗ L2 0.000 Lift mode: To return plane</p>	<p>Depth as referred to Z0 [abs, inc]</p>	
DZ	<p><b>Path milling</b></p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C Machining: Radius compens: </p> <p>Z0 0.000 abs Z1 0.000 inc DZ 0.000 UZ 0.000 UXY 0.000 mm Approach: Straight + L1 0.000 FZ 0.000 mm/tooth Retract: Straight ↗ L2 0.000 Lift mode: To return plane</p>	<p>Maximum infeed to depth (Z-direction)</p>	
UZ	<p><b>Path milling</b></p> <p>T D1 F 8.000 mm/tooth V 8 m/min</p> <p>Face C Machining: Radius compens: </p> <p>Z0 0.000 abs Z1 0.000 inc DZ 0.000 UZ 0.000 UXY 0.000 mm Approach: Straight + L1 0.000 FZ 0.000 mm/tooth Retract: Straight ↗ L2 0.000 Lift mode: To return plane</p>	<p>Depth of finishing allowance [mm] Only available for roughing</p>	

## Section 6

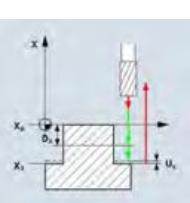
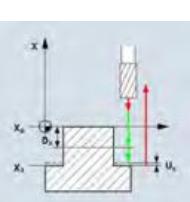
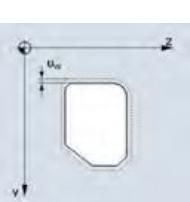
### Continuous path milling

Notes

Parameter	Help picture	Description and hints
Face C:		
UXY		Finishing allowance in the plane - only when roughing to the left and right of the contour [mm]
Perimeter face C:		
X0		Cylinder diameter $\emptyset$ [abs]
X1		Depth as referred to X0 $\emptyset$ [abs, inc]

## Section 6

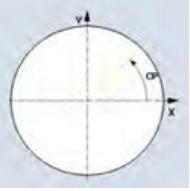
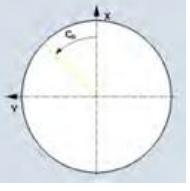
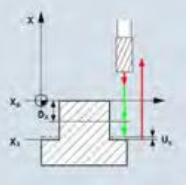
### Continuous path milling

Parameter	Help picture	Description and hints	Notes
Perimeter face C:			
DX		Maximum infeed to depth (X-direction)	
UX		Depth of finishing allowance [mm] Only available for roughing	
UYZ		Finishing allowance in the plane - only when roughing to the left and right of the contour [mm]	

## Section 6

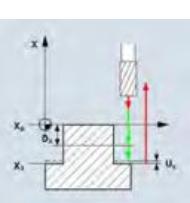
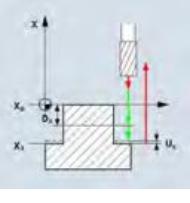
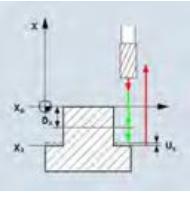
### Continuous path milling

Notes

Parameter	Help picture	Description and hints
<b>Face Y:</b>		
CP		Reference point angle to X
Path milling T D1 F 8.000 mm/tooth V 8 m/min Face Y Machining: Radius compens:  CP 0.000 ° Z0 0.000 abs Z1 0.000 inc DZ 0.000 UZ 0.000 UXY 0.000 mm Approach: Straight L1 0.000 FZ 0.000 mm/tooth Retract: Straight L2 0.000 Lift mode: To return plane		
Z0, Z1, DZ, UZ, UXY		See Face C
<b>Perimeter face Y:</b>		
C0		Reference point [Degree]
Path milling T D1 F 8.000 mm/tooth V 8 m/min Per.surf.Y Machining: Radius compens:  C0 0.000 ° X0 0.000 abs X1 0.000 inc DX 0.000 UX 0.000 UY2 0.000 mm Approach: Straight L1 0.000 FX 0.000 mm/tooth Retract: Straight L2 0.000 Lift mode: To return plane		
X0		Reference point in X-direction [abs]

## Section 6

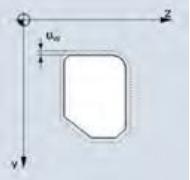
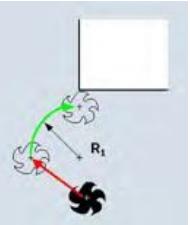
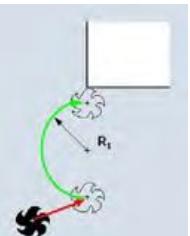
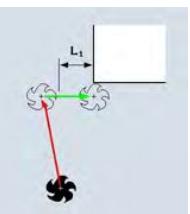
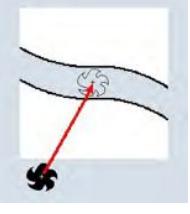
### Continuous path milling

Parameter	Help picture	Description and hints	Notes
Perimeter face Y:			
X1		Depth as referred to X0 [abs, inc]	
DX		Maximum infeed to depth (X-direction)	
UX		Depth of finishing allowance [mm] Only available for roughing	

## Section 6

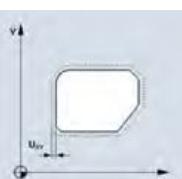
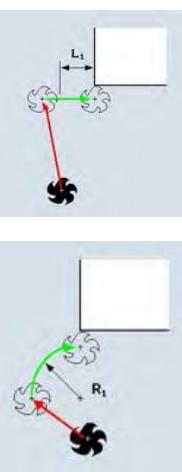
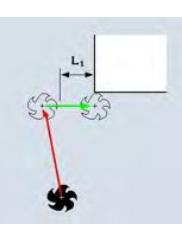
### Continuous path milling

Notes

Parameter	Help picture	Description and hints
Perimeter face Y:		
UYZ		Finishing allowance in the plane - only when roughing to the left and right of the contour [mm]
General parameters:		
Approach mode	   	Quarter circle: Part of a spiral (only for path milling left and right of the contour) Semi circle: Part of a spiral (only for path milling left and right of the contour) Straight: Oblique in space Vertical: Vertical to the path (only for path milling on the cutter centre path, only if radius correction is disabled)

## Section 6

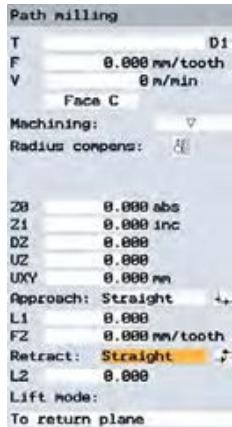
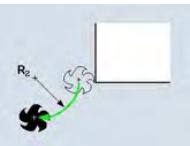
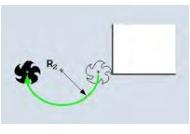
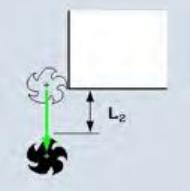
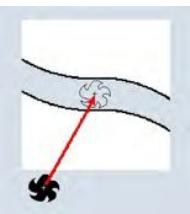
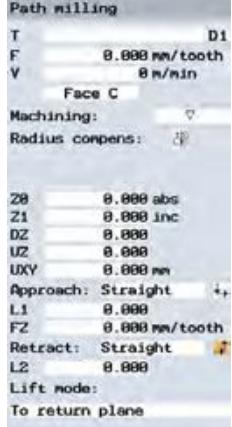
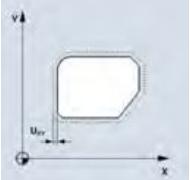
### Continuous path milling

Parameter	Help picture	Description and hints	Notes
General parameters:			
Approach strategy		Axis-by-axis	
<b>Path milling</b> T D1 F 0.000 mm/tooth V 0 m/min Face C Machining: Radius compens:  Z0 0.000 abs Z1 0.000 inc DZ 0.000 UZ 0.000 UXV 0.000 mm Approach: Straight L1 0.000 FZ 0.000 mm/tooth Retract: Straight L2 0.000 Lift mode: To return plane		In space (not for vertical approach)	
L1, R1		L1: Approach distance - only with approach mode straight  R1: Anfahrradius - only with approach mode with quarter and semi-circle	
FZ, FX		FZ: Infeed depth to depth - only for Face C and Face Y - Approach strategy axis-by-axis  FX: Infeed depth to depth - only for perimeter face C and perimeter face Y - Approach strategy axis-by-axis  [mm/tooth, mm/min]	

## Section 6

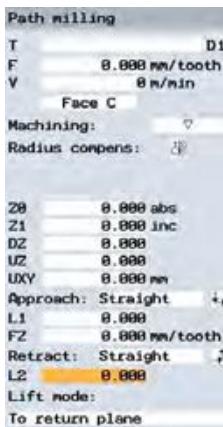
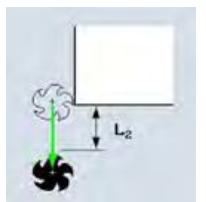
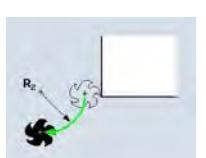
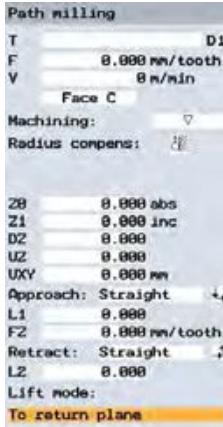
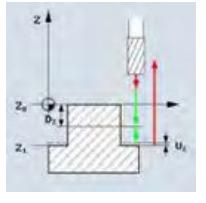
### Continuous path milling

Notes

Parameter	Help picture	Description and hints
Leaving mode	    	Quarter circle: Part of a spiral (only for path milling left and right of the contour) Semi circle: Part of a spiral (only for path milling left and right of the contour) Straight: Oblique in space Vertical: Vertical to the path (only for path milling on the cutter centre path, only with radius correction disabled)
Leaving strategy	 	Axis-by-axis
		 In space (not for vertical approach)

## Section 6

### Continuous path milling

Parameter	Help picture	Description and hints	Notes
L2, R2	  	<p>L2: Leaving distance - only for leaving mode Straight</p> <p>R2: Leaving distance - only for leaving mode with quarter and semi circle</p>	
Lift mode	 	<p>If several infeeds are necessary, state the retraction height to which the tool is to be lifted between the individual infeeds (for traverse from the end of the contour to the start).</p> <p>Z0+Safety distance - only for face C and face Y or X0+Safety distance - only for perimeter face C and perimeter face Y</p> <p>By the safety distance</p> <p>To the retraction plane</p> <p>No retraction</p>	

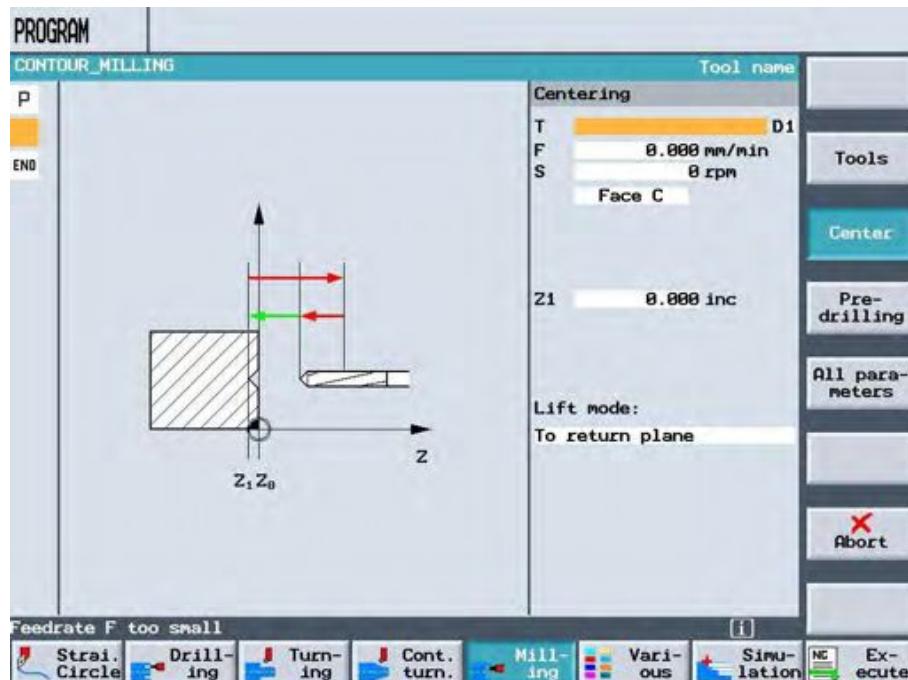
## Section 7

Notes

### Pre-drilling

Pre-drilling

By pressing the VSK 3 “Pre-drilling“ the function “Pre-drilling“ with the sub-functions  
Centring and  
Pre-drilling  
will be selected.



This function is used if for cleaning-up of contour pockets it is not possible to enter the pocket centrally with the cutter.

The number and the positions of the required pre-drillings depends on the special circumstances, such as for instance the type of contours, the tool, the infeed plane and the finishing allowance.

The drilling positions in the contour pocket cycle are already determined during the evaluation of the contour pocket. From this a special drilling program is produced which is called up for the pre-drilling cycles (centring and pre-drilling).

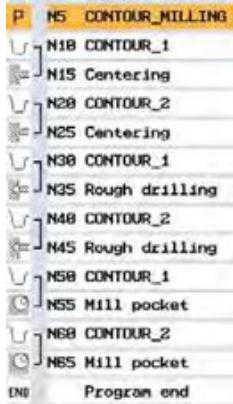
## Section 7

### Pre-drilling

#### Hints for machining

If there are several pockets to be milled, unnecessary tool changes can be avoided if firstly all pockets are centred and pre-drilled and then followed by cleaning-up. In this case the parameters, which appear when the VSK 5 “All parameters“ is pressed, must also be entered.

1 Example for programming:



1. Contour pocket 1
2. Centring
3. Contour pocket 2
4. Centring
5. Contour pocket 1
6. Pre-drilling
7. Contour pocket 2
8. Pre-drilling
9. Contour pocket 1
10. Cleaning-up
11. Contour pocket 2
12. Cleaning-up

Notes

If a pocket is completely machined, i.e. centred, pre-drilled and cleaned-up and the additional parameter were not filled in, then ShopTurn takes the parameter values from the machining step “Cleaning-up“.

#### Vertical Softkey-strip

Center

By pressing the VSK 3 “Centring“ the function “Centring“ will be called up.

Pre-drilling

By pressing the VSK 4 “Pre-drilling“ the function “Pre-drilling“ will be called up.

## Section 7

Notes

### Pre-drilling

The parameters listed below can be used for centring as well as pre-drilling.

The following parameters are shown after the VSK 5 "All parameters" is pressed:

For centring:

Face C: TR, Z0, DXY and UXY

Perimeter face C: TR, DYZ and UYZ

Face Y: TR, Z0, DXY and UXY

Perimeter face Y: TR, X0, DYZ and UYZ

For pre-drilling:

Face C: TR, Z0, Z1, DXY, UXY and UZ

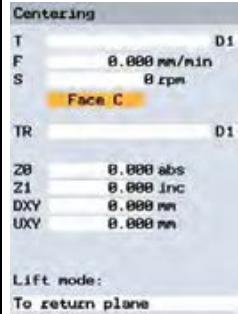
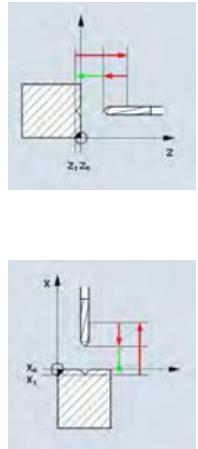
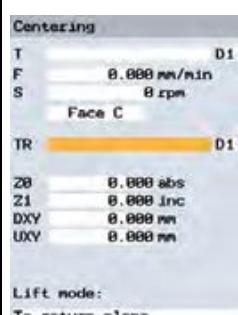
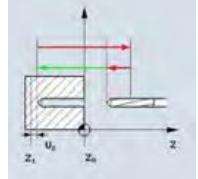
Perimeter face C: TR, X1, DYZ, UYZ and UX

Face Y: TR, Z0, Z1, DXY, UXY and UZ

Perimeter face Y: TR, X0, X1, DYZ UYZ and UX

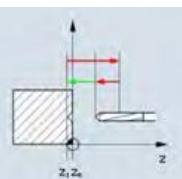
If the functions "Centring" and "Pre-drilling" are programmed within a linked chain then the programming of these parameters is not necessary. The necessary data are taken from the subsequent program steps. In this manner, for instance, the depth for pre-drilling Z1 is taken from a further milling cycle (Cleaning-up) and the depth programmed there.

If the programming of Centring and Pre-drilling takes place outside a linked chain, all available parameters called-up by pressing the VSK 5 "All parameters" must be used.

Parameter	Help picture	Description and hints
T, D, F, S / V		See module A115 – "Operating AreaProgram - Programming Centring, Drilling, Thread cutting, Positions", Section 3 "Drilling centric".
Location	 	<b>Face C</b> Face C  <b>Face Y</b> Face Y  <b>Per.surf.C</b> Perimeter face C  <b>Per.surf.Y</b> Perimeter face Y
TR	 	Reference tool for Centring

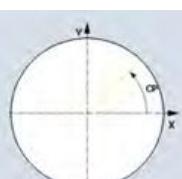
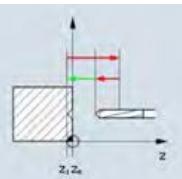
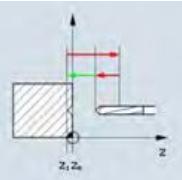
## Section 7

### Pre-drilling

Parameter	Help picture	Description and hints	Notes																		
<p>Lift mode</p> <p><b>Centering</b></p> <table border="1"> <tr><td>T</td><td>Di</td></tr> <tr><td>F</td><td>8.000 mm/min 8 rpm</td></tr> <tr><td>S</td><td></td></tr> <tr><td>Face C</td><td></td></tr> <tr><td>TR</td><td>Di</td></tr> <tr><td>ZB</td><td>8.000 abs</td></tr> <tr><td>Z1</td><td>8.000 inc</td></tr> <tr><td>DXV</td><td>8.000 mm</td></tr> <tr><td>UXY</td><td>8.000 mm</td></tr> </table> <p><b>Lift mode:</b> <b>To return plane</b></p>	T	Di	F	8.000 mm/min 8 rpm	S		Face C		TR	Di	ZB	8.000 abs	Z1	8.000 inc	DXV	8.000 mm	UXY	8.000 mm	 <p>Centring</p>	<p>Lift mode before renewed infeed</p> <p>If for the machining several entry points are necessary, state the height at which the tool is to move on to the next point of entry:</p> <p>To retraction level</p> <p>Z0 + Safety distance (Face C and Face Y) or X0 + Safety distance (Perimeter face C and Perimeter face Y)</p> <p>If within the confines of the pockets there are no islands higher than Z0 (X0), then Z0 + Safety distance (X0 + Safety distance) can be programmed as Lift mode.</p>	
T	Di																				
F	8.000 mm/min 8 rpm																				
S																					
Face C																					
TR	Di																				
ZB	8.000 abs																				
Z1	8.000 inc																				
DXV	8.000 mm																				
UXY	8.000 mm																				

Hint:

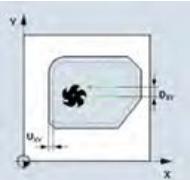
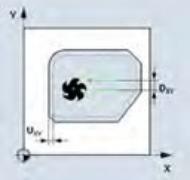
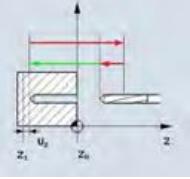
The parameters T, D, F, S / V, position, TR, as well as the lift mode are valid for all masks in the ranges Centring and pre-drilling.

Face C / Face Y:																							
<p>CP</p> <p><b>Centering</b></p> <table border="1"> <tr><td>T</td><td>Di</td></tr> <tr><td>F</td><td>8.000 mm/min 8 rpm</td></tr> <tr><td>S</td><td></td></tr> <tr><td>Face Y</td><td></td></tr> <tr><td>TR</td><td>Di</td></tr> <tr><td>OP</td><td>8.000 °</td></tr> <tr><td>ZB</td><td>8.000 abs</td></tr> <tr><td>Z1</td><td>8.000 inc</td></tr> <tr><td>DXV</td><td>8.000 mm</td></tr> <tr><td>UXY</td><td>8.000 mm</td></tr> </table> <p><b>Lift mode:</b> <b>To return plane</b></p>	T	Di	F	8.000 mm/min 8 rpm	S		Face Y		TR	Di	OP	8.000 °	ZB	8.000 abs	Z1	8.000 inc	DXV	8.000 mm	UXY	8.000 mm	 <p>Centring</p>	Reference point - only for face Y	
T	Di																						
F	8.000 mm/min 8 rpm																						
S																							
Face Y																							
TR	Di																						
OP	8.000 °																						
ZB	8.000 abs																						
Z1	8.000 inc																						
DXV	8.000 mm																						
UXY	8.000 mm																						
<p>Z0</p> <p><b>Centering</b></p> <table border="1"> <tr><td>T</td><td>Di</td></tr> <tr><td>F</td><td>8.000 mm/min 8 rpm</td></tr> <tr><td>S</td><td></td></tr> <tr><td>Face C</td><td></td></tr> <tr><td>TR</td><td>Di</td></tr> <tr><td>ZB</td><td>8.000 abs</td></tr> <tr><td>Z1</td><td>8.000 inc</td></tr> <tr><td>DXV</td><td>8.000 mm</td></tr> <tr><td>UXY</td><td>8.000 mm</td></tr> </table> <p><b>Lift mode:</b> <b>To return plane</b></p>	T	Di	F	8.000 mm/min 8 rpm	S		Face C		TR	Di	ZB	8.000 abs	Z1	8.000 inc	DXV	8.000 mm	UXY	8.000 mm	 <p>Centring</p>	Reference point in Z-direction [abs]			
T	Di																						
F	8.000 mm/min 8 rpm																						
S																							
Face C																							
TR	Di																						
ZB	8.000 abs																						
Z1	8.000 inc																						
DXV	8.000 mm																						
UXY	8.000 mm																						
<p>Z1</p> <p><b>Centering</b></p> <table border="1"> <tr><td>T</td><td>Di</td></tr> <tr><td>F</td><td>8.000 mm/min 8 rpm</td></tr> <tr><td>S</td><td></td></tr> <tr><td>Face C</td><td></td></tr> <tr><td>TR</td><td>Di</td></tr> <tr><td>ZB</td><td>8.000 abs</td></tr> <tr><td>Z1</td><td>8.000 inc</td></tr> <tr><td>DXV</td><td>8.000 mm</td></tr> <tr><td>UXY</td><td>8.000 mm</td></tr> </table> <p><b>Lift mode:</b> <b>To return plane</b></p>	T	Di	F	8.000 mm/min 8 rpm	S		Face C		TR	Di	ZB	8.000 abs	Z1	8.000 inc	DXV	8.000 mm	UXY	8.000 mm	 <p>Centring</p>	Depth as referred to Z0 [inc]			
T	Di																						
F	8.000 mm/min 8 rpm																						
S																							
Face C																							
TR	Di																						
ZB	8.000 abs																						
Z1	8.000 inc																						
DXV	8.000 mm																						
UXY	8.000 mm																						

## Section 7

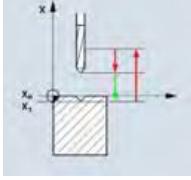
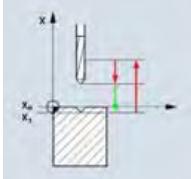
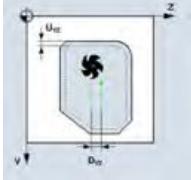
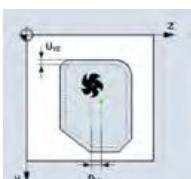
### Pre-drilling

Notes

Parameter	Help picture	Description and hints
<b>Face C / Face Y:</b>		
<b>DXY</b>	 Centring	Maximum infeed in the XY-plane Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [% , mm]
<b>UXY</b>	 Centring	Finishing allowance in the plane
<b>UZ</b>	 Pre-drilling	Finishing allowance at the bottom - only when pre-drilling

## Section 7

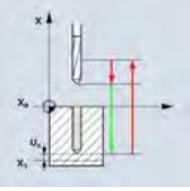
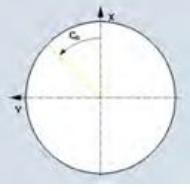
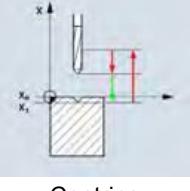
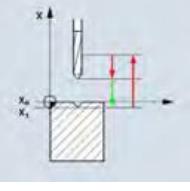
### Pre-drilling

Parameter	Help picture	Description and hints	Notes
Perimeter face C:			
X0	<p><b>Centering</b></p> <p>T Di F 8.000 mm/min S 8 rpm Per.surf.C</p> <p>TR Di</p> <p>X0 8.000 abs X1 8.000 inc DY2 8.000 mm UY2 8.000 mm</p> <p>Lift mode: To return plane</p>	 <p>Cylinder diameter <math>\emptyset</math> [abs]</p> <p>Centring</p>	
X1	<p><b>Centering</b></p> <p>T Di F 8.000 mm/min S 8 rpm Per.surf.C</p> <p>TR Di</p> <p>X0 8.000 abs X1 8.000 inc DY2 8.000 mm UY2 8.000 mm</p> <p>Lift mode: To return plane</p>	 <p>Depth as referred to X0 [inc]</p> <p>Centring</p>	
DYZ	<p><b>Centering</b></p> <p>T Di F 8.000 mm/min S 8 rpm Per.surf.C</p> <p>TR Di</p> <p>X0 8.000 abs X1 8.000 inc DY2 8.000 mm UY2 8.000 mm</p> <p>Lift mode: To return plane</p>	 <p>Maximum infeed in the YZ-plane Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%], mm</p> <p>Centring</p>	
UYZ	<p><b>Centering</b></p> <p>T Di F 8.000 mm/min S 8 rpm Per.surf.C</p> <p>TR Di</p> <p>X0 8.000 abs X1 8.000 inc DY2 8.000 mm UY2 8.000 mm</p> <p>Lift mode: To return plane</p>	 <p>Finishing allowance in the plane [mm]</p> <p>Centring</p>	

## Section 7

### Pre-drilling

Notes

Parameter	Help picture	Description and hints
<b>Perimeter face C:</b>		
UX	 Centring	Finishing allowance at the bottom - only when pre-drilling
<b>Perimeter face Y:</b>		
C0	 Centring	Reference point [Degree]
X0	 Centring	Reference point in X-direction [abs]
X1	 Centring	Depth as referred to X0 [abs, inc]
DYZ, UYZ, UX		See Perimeter face C

## Section 8

Milling of contour pockets and removal of any residue material

### 8.1 Milling of contour pockets

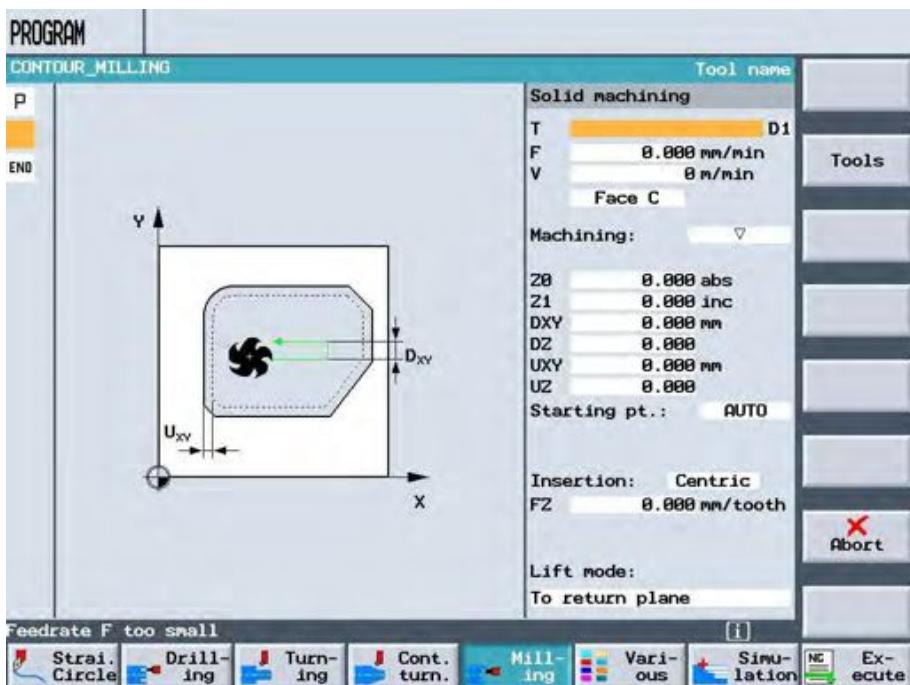
Mill pocket

By pressing the VSK 4 "Mill pocket" the function "contour pocket milling" is selected.

Pockets with and without islands can be machined. The islands may also be situated partially outside the pocket or they can overlap.

The machining direction is determined by the machining rotation (conventional or climb milling) specified in the program heading.

Notes

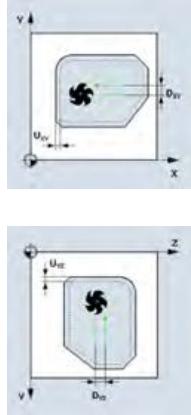
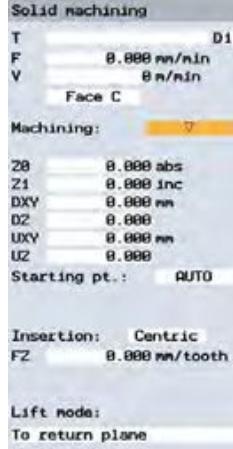
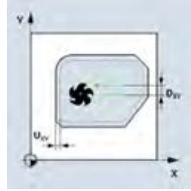
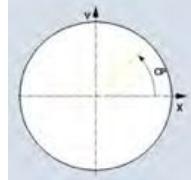


## Section 8

Notes

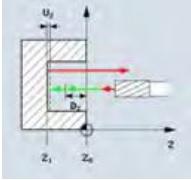
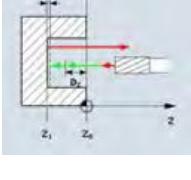
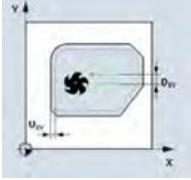
Milling of contour pockets and removal of any residue material

Parameters for the function „Contour pocket milling“

Parameter	Help picture	Description and hints
T, D, F, S / V		See module A115 – “Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions“, Section 3 “Drilling centric“.
Location	 	<b>Face C</b> Face C <b>Face Y</b> Face Y <b>Per.surf.C</b> Perimeter face C <b>Per.surf.Y</b> Perimeter face Y
Machining	 	Roughing <b>▽</b> Finishing bottom <b>VVV bottom</b> Finishing edge <b>VVV wall</b>
Face C / Face Y:	 	Reference point - only for face Y

## Section 8

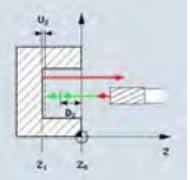
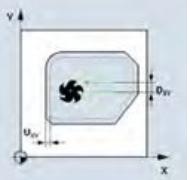
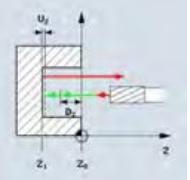
Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes																																																
Z0	<p><b>Face C / Face Y:</b></p>  <p><b>Solid machining</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/min</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td><td>B/m/min</td></tr> <tr><td>V</td><td>0.000</td><td></td></tr> <tr><td colspan="3">Face C</td></tr> <tr><td colspan="3">Machining:</td></tr> <tr><td>Z0</td><td>0.000 abs</td><td></td></tr> <tr><td>Z1</td><td>0.000 inc</td><td></td></tr> <tr><td>DXV</td><td>0.000 mm</td><td></td></tr> <tr><td>D2</td><td>0.000</td><td></td></tr> <tr><td>UXY</td><td>0.000 mm</td><td></td></tr> <tr><td>UZ</td><td>0.000</td><td></td></tr> <tr><td>Starting pt.:</td><td>AUTO</td><td></td></tr> <tr><td colspan="3">Insertion: Centric</td></tr> <tr><td>FZ</td><td>0.000 mm/tooth</td><td></td></tr> <tr><td colspan="3">Lift mode:</td></tr> <tr><td colspan="3">To return plane</td></tr> </table>	T	0.000 mm/min	D1	F	0.000 mm/min	B/m/min	V	0.000		Face C			Machining:			Z0	0.000 abs		Z1	0.000 inc		DXV	0.000 mm		D2	0.000		UXY	0.000 mm		UZ	0.000		Starting pt.:	AUTO		Insertion: Centric			FZ	0.000 mm/tooth		Lift mode:			To return plane			Reference point in Z-direction [abs]	
T	0.000 mm/min	D1																																																	
F	0.000 mm/min	B/m/min																																																	
V	0.000																																																		
Face C																																																			
Machining:																																																			
Z0	0.000 abs																																																		
Z1	0.000 inc																																																		
DXV	0.000 mm																																																		
D2	0.000																																																		
UXY	0.000 mm																																																		
UZ	0.000																																																		
Starting pt.:	AUTO																																																		
Insertion: Centric																																																			
FZ	0.000 mm/tooth																																																		
Lift mode:																																																			
To return plane																																																			
Z1	 <p><b>Solid machining</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/min</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td><td>B/m/min</td></tr> <tr><td>V</td><td>0.000</td><td></td></tr> <tr><td colspan="3">Face C</td></tr> <tr><td colspan="3">Machining:</td></tr> <tr><td>Z0</td><td>0.000 abs</td><td></td></tr> <tr><td>Z1</td><td>0.000 inc</td><td></td></tr> <tr><td>DXV</td><td>0.000 mm</td><td></td></tr> <tr><td>D2</td><td>0.000</td><td></td></tr> <tr><td>UXY</td><td>0.000 mm</td><td></td></tr> <tr><td>UZ</td><td>0.000</td><td></td></tr> <tr><td>Starting pt.:</td><td>AUTO</td><td></td></tr> <tr><td colspan="3">Insertion: Centric</td></tr> <tr><td>FZ</td><td>0.000 mm/tooth</td><td></td></tr> <tr><td colspan="3">Lift mode:</td></tr> <tr><td colspan="3">To return plane</td></tr> </table>	T	0.000 mm/min	D1	F	0.000 mm/min	B/m/min	V	0.000		Face C			Machining:			Z0	0.000 abs		Z1	0.000 inc		DXV	0.000 mm		D2	0.000		UXY	0.000 mm		UZ	0.000		Starting pt.:	AUTO		Insertion: Centric			FZ	0.000 mm/tooth		Lift mode:			To return plane			Depth as referred to Z0 [abs, inc]	
T	0.000 mm/min	D1																																																	
F	0.000 mm/min	B/m/min																																																	
V	0.000																																																		
Face C																																																			
Machining:																																																			
Z0	0.000 abs																																																		
Z1	0.000 inc																																																		
DXV	0.000 mm																																																		
D2	0.000																																																		
UXY	0.000 mm																																																		
UZ	0.000																																																		
Starting pt.:	AUTO																																																		
Insertion: Centric																																																			
FZ	0.000 mm/tooth																																																		
Lift mode:																																																			
To return plane																																																			
DXY	 <p><b>Solid machining</b></p> <table border="1"> <tr><td>T</td><td>0.000 mm/min</td><td>D1</td></tr> <tr><td>F</td><td>0.000 mm/min</td><td>B/m/min</td></tr> <tr><td>V</td><td>0.000</td><td></td></tr> <tr><td colspan="3">Face C</td></tr> <tr><td colspan="3">Machining:</td></tr> <tr><td>Z0</td><td>0.000 abs</td><td></td></tr> <tr><td>Z1</td><td>0.000 inc</td><td></td></tr> <tr><td>DXV</td><td>0.000 mm</td><td></td></tr> <tr><td>D2</td><td>0.000</td><td></td></tr> <tr><td>UXY</td><td>0.000 mm</td><td></td></tr> <tr><td>UZ</td><td>0.000</td><td></td></tr> <tr><td>Starting pt.:</td><td>AUTO</td><td></td></tr> <tr><td colspan="3">Insertion: Centric</td></tr> <tr><td>FZ</td><td>0.000 mm/tooth</td><td></td></tr> <tr><td colspan="3">Lift mode:</td></tr> <tr><td colspan="3">To return plane</td></tr> </table>	T	0.000 mm/min	D1	F	0.000 mm/min	B/m/min	V	0.000		Face C			Machining:			Z0	0.000 abs		Z1	0.000 inc		DXV	0.000 mm		D2	0.000		UXY	0.000 mm		UZ	0.000		Starting pt.:	AUTO		Insertion: Centric			FZ	0.000 mm/tooth		Lift mode:			To return plane			Maximum infeed in the XY-plane Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%], [mm]	
T	0.000 mm/min	D1																																																	
F	0.000 mm/min	B/m/min																																																	
V	0.000																																																		
Face C																																																			
Machining:																																																			
Z0	0.000 abs																																																		
Z1	0.000 inc																																																		
DXV	0.000 mm																																																		
D2	0.000																																																		
UXY	0.000 mm																																																		
UZ	0.000																																																		
Starting pt.:	AUTO																																																		
Insertion: Centric																																																			
FZ	0.000 mm/tooth																																																		
Lift mode:																																																			
To return plane																																																			

## Section 8

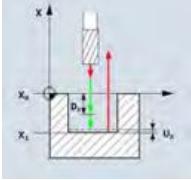
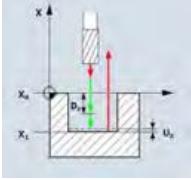
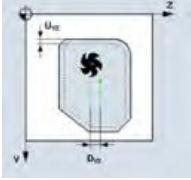
Milling of contour pockets and removal of any residue material

Notes

Parameter	Help picture	Description and hints
<b>Face C / Face Y:</b>		
<b>DZ</b>		Maximum infeed to depth (Z-direction)
<b>UXY</b>		Finishing allowance in the plane [mm]
<b>UZ</b>		Finishing allowance at the bottom

## Section 8

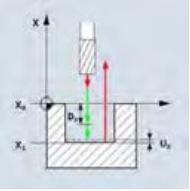
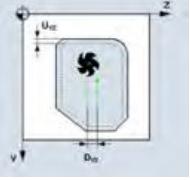
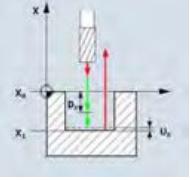
Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes
X0  Solid machining T D1 F 0.000 mm/min V 0 m/min Per.surf.C Machining: X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000 Starting pt.: AUTO  Insertion: Centric FX 0.000 mm/tooth  Lift mode: To return plane		Perimeter face C: Cylinder diameter $\emptyset$ [abs]	
X1  Solid machining T D1 F 0.000 mm/min V 0 m/min Per.surf.C Machining: X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000 Starting pt.: AUTO  Insertion: Centric FX 0.000 mm/tooth  Lift mode: To return plane		Depth as referred to X0 [abs, inc]	
DYZ  Solid machining T D1 F 0.000 mm/min V 0 m/min Per.surf.C Machining: X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000 Starting pt.: AUTO  Insertion: Centric FX 0.000 mm/tooth  Lift mode: To return plane		Maximum infeed in the YZ-plane Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%], mm]	

## Section 8

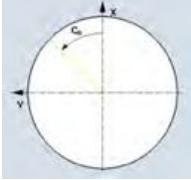
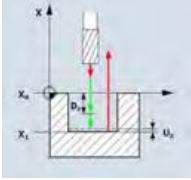
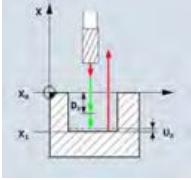
Milling of contour pockets and removal of any residue material

Notes

Parameter	Help picture	Description and hints
<b>Perimeter face C:</b>		
<b>DX</b>		Maximum infeed to depth (X-direction)
<b>UYZ</b>		Finishing allowance in the plane [mm]
<b>UX</b>		Finishing allowance at the bottom

## Section 8

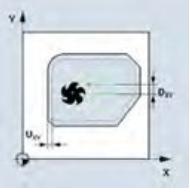
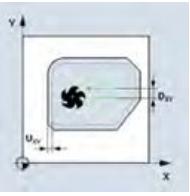
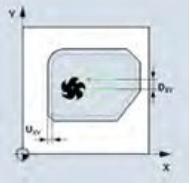
Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes
C0  Solid machining T D1 F 0.000 mm/min V 0 mm/min Per.surf.V Machining: C0 0.000 ° X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000 Starting pt.: AUTO  Insertion: Centric FX 0.000 mm/tooth  Lift mode: To return plane		Reference point [Degree]	
X0  Solid machining T D1 F 0.000 mm/min V 0 mm/min Per.surf.V Machining: C0 0.000 ° X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000 Starting pt.: AUTO  Insertion: Centric FX 0.000 mm/tooth  Lift mode: To return plane		Reference point in X-direction [abs]	
X1  Solid machining T D1 F 0.000 mm/min V 0 mm/min Per.surf.V Machining: C0 0.000 ° X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000 Starting pt.: AUTO  Insertion: Centric FX 0.000 mm/tooth  Lift mode: To return plane		Depth as referred to X0 [abs, inc]	
DYZ, DX, UYZ, UX		See Perimeter face C	

## Section 8

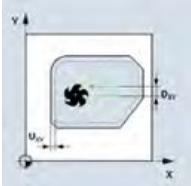
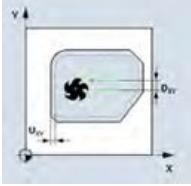
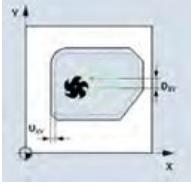
Milling of contour pockets and removal of any residue material

Notes

Parameter	Help picture	Description and hints
General parameters:		
Start point  <b>Solid machining</b> T D1 F 0.000 mm/min V 0 mm/min Face C Machining: Z0 0.000 abs Z1 0.000 inc DXY 0.000 mm D2 0.000 UXY 0.000 mm UZ 0.000 Starting pt.: Manual X 0.000 abs Y 0.000 abs Insertion: Centric FZ 0.000 mm/tooth  Lift mode: To return plane		<p>Specify whether Start point <b>automatic</b> or Start point <b>manual</b> is used.</p> <p>In case of manual input the Start point can also be located outside the pocket, in which case there will be a straight machining into the pocket, e.g.. for a laterally open pocket without plunging. - available only for roughing and finishing the bottom.</p> <p><b>Manual</b> Input for co-ordinates in X- and Y-direction</p> <p><b>AUTO</b> The Start point will be determined automatically</p>
X, Y  <b>Solid machining</b> T D1 F 0.000 mm/min V 0 mm/min Face C Machining: Z0 0.000 abs Z1 0.000 inc DXY 0.000 mm D2 0.000 UXY 0.000 mm UZ 0.000 Starting pt.: Manual X 0.000 abs Y 0.000 abs Insertion: Centric FZ 0.000 mm/tooth  Lift mode: To return plane		<p>X: Start point X – only for Face C and Face Y</p> <p>Y: Start point Y – only for perimeter face C and perimeter face Y</p> <p>[abs]</p> <p>Only for Start point manual.</p>
Y, Z  <b>Solid machining</b> T D1 F 0.000 mm/min V 0 mm/min Face C Machining: Z0 0.000 abs Z1 0.000 inc DXY 0.000 mm D2 0.000 UXY 0.000 mm UZ 0.000 Starting pt.: Manual X 0.000 abs Y 0.000 abs Insertion: Centric FZ 0.000 mm/tooth  Lift mode: To return plane		<p>Y: Start point Y – only for Face C and Face Y</p> <p>Z: Start point Z – only for perimeter face C and perimeter face Y</p> <p>[abs]</p> <p>Only available with Start point manual.</p>

## Section 8

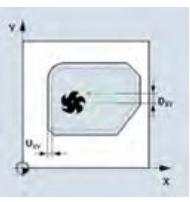
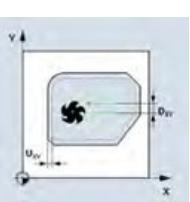
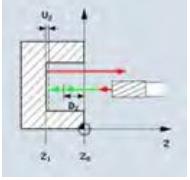
Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes
Plunging		<p>Only available for "Roughing " and "Finishing bottom".</p> <p><b>Centric</b> Plunging in centre of pocket</p> <p><b>helical</b> Plunging with a spiral path</p> <p><b>oscillat</b> Plunging with an oscillating motion along the centre line of the pocket</p>	
FZ		<p>FZ: Infeed feed rate to depth - only for Face C and Face Y</p> <p>FX: Infeed feed rate to depth - only for - only with perimeter face C and perimeter face Y [mm/tooth, mm/min]</p> <p>Only available for centric.</p>	
EP		<p>Max. pitch of helix [mm/rev]</p> <p>Only available for Helical.</p>	

## Section 8

### Milling of contour pockets and removal of any residue material

Notes

Parameter	Help picture	Description and hints
<b>ER</b> 		Radius of the helix [mm] Only available for Helical
<b>EW</b> 		Plunging angle [degrees] Available only with oscillating.
<b>Lift mode</b> 		Lift mode before a renewed infeed If for the machining several entry points are necessary, state the height at which the tool is to move on to the next point of entry: To retraction level Z0 + Safety distance (Face C and Face Y) or X0 + Safety distance (Perimeter face C and Perimeter face Y) If within the confines of the pockets there are no islands higher than Z0 (X0), then Z0 + Safety distance (X0 + Safety distance) can be programmed as Lift mode.

## Section 8

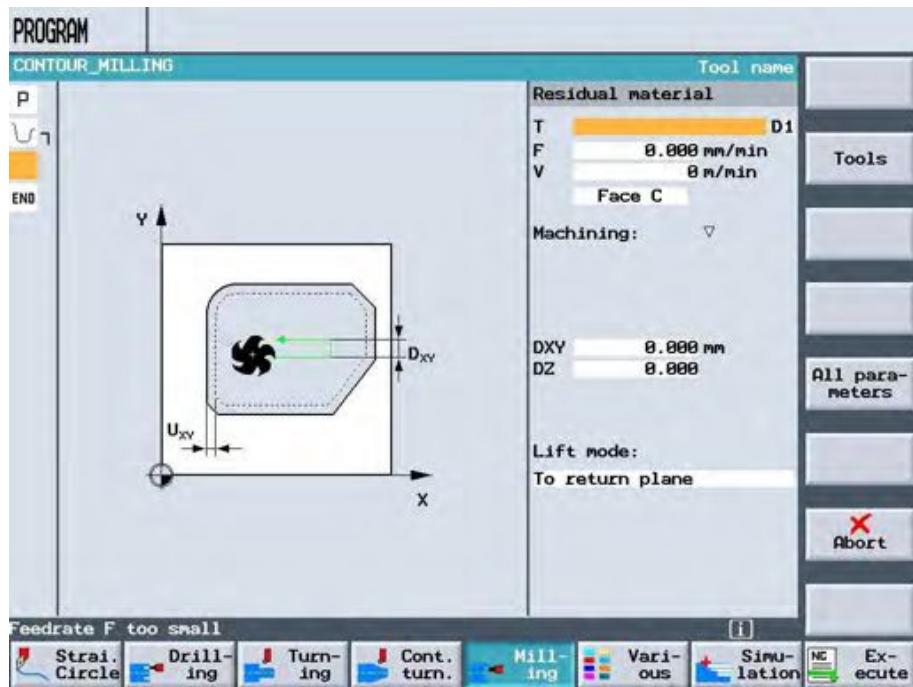
Milling of contour pockets and removal of any residue material

### 8.2 Clearing of residue material (Option)

Pocket  
res.mat.

By pressing the VSK 5 "Pocket res.mat." the function "Cleaning-up of rest material in contour pocket" is selected.

Notes



If a pocket (with/without islands) has been cleaned-up and any rest material has been left, ShopMill recognizes this automatically. Using a suitable tool this residue material can be removed without having to re-machine the entire pocket, i.e. unnecessary dummy passes are avoided. The rest material is evaluated on the basis of the cutter used for cleaning up.

Hints:

Material, which has been left due to the finishing allowance, is not rest material.

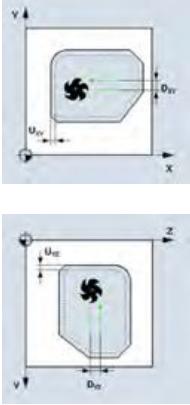
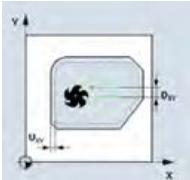
If several pockets have to be milled and unnecessary tool changes are to be avoided, it is sensible to clear out all the pockets and to remove the rest material thereafter. In this case the tool, that was used for clearing the pocket must be entered into the parameter "Reference tool TR" when clearing the residue material. For this the VSK 5 "All parameters" must be pressed.

## Section 8

Notes

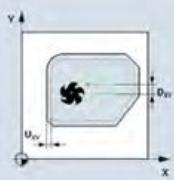
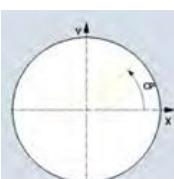
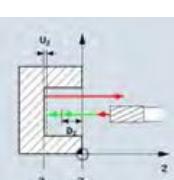
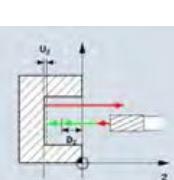
Milling of contour pockets and removal of any residue material

Parameter for the function “Clearing of residue material”

Parameter	Help picture	Description and hints
T, D, F, S / V		See module A115 – “Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions”, Section 3 “Drilling centric“.
Location		<b>Face C</b> Face C <b>Face Y</b> Face Y <b>Per.surf.C</b> Perimeter face C <b>Per.surf.Y</b> Perimeter face Y
Machining		Roughing Display only, no input is possible.
TR		Reference tool for residue material Only available for VSK 5 „All parameters.“

## Section 8

Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes
D  Residual material T Di F 8.000 mm/min V 8 m/min Face C Machining: TR D1 ZB 0.000 abs Z1 0.000 inc DXY 0.000 mm DZ 0.000 UXY 0.000 mm UZ 0.000 Lift mode: To return plane		Cutting edge number  Only available for VSK 5 "All parameters"	
Face C / Face Y:  CP  Residual material T Di F 8.000 mm/min V 8 m/min Face Y Machining: TR D1 CP 8.000 ° ZB 0.000 abs Z1 0.000 inc DXY 0.000 mm DZ 0.000 UXY 0.000 mm UZ 0.000 Lift mode: To return plane		Reference point - only for face Y	
Z0  Residual material T Di F 8.000 mm/min V 8 m/min Face C Machining: TR D1 ZB 0.000 abs Z1 0.000 inc DXY 0.000 mm DZ 0.000 UXY 0.000 mm UZ 0.000 Lift mode: To return plane		Reference point in Z-direction  [abs]  Only available for VSK 5 "All parameters."	
Z1  Residual material T Di F 8.000 mm/min V 8 m/min Face C Machining: TR D1 ZB 0.000 abs Z1 0.000 inc DXY 0.000 mm DZ 0.000 UXY 0.000 mm UZ 0.000 Lift mode: To return plane		Depth as referred to Z0  [abs, inc]	

## Section 8

Milling of contour pockets and removal of any residue material

Notes

Parameter	Help picture	Description and hints
<b>Face C / Face Y:</b>		
<b>DXY</b>		Maximum infeed in the XY-plane Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%], [mm]
<b>DZ</b>		Maximum infeed to depth (Z-direction)
<b>UXY</b>		Finishing allowance in the plane [mm]
<b>UZ</b>		Finishing allowance at the bottom

## Section 8

Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes
X0	Perimeter face C:		
	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.C</p> <p>Machining:</p> <p>TR D1</p> <p>X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	<p>Cylinder diameter <math>\emptyset</math> [abs]</p>	
X1		Depth as referred to X0 $\emptyset$	
	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.C</p> <p>Machining:</p> <p>TR D1</p> <p>X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	<p>[abs, inc]</p>	
DYZ		Maximum infeed in the YZ-plane	
	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.C</p> <p>Machining:</p> <p>TR D1</p> <p>X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	<p>Plane infeed in %: Ratio of plane infeed (mm) to cutter diameter (mm) [%], mm]</p>	
DX		Maximum infeed to depth (X-direction)	
	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.C</p> <p>Machining:</p> <p>TR D1</p> <p>X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UY2 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>		

## Section 8

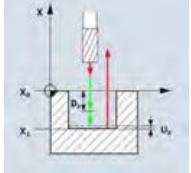
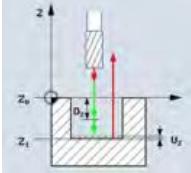
Milling of contour pockets and removal of any residue material

Notes

Parameter	Help picture	Description and hints
Perimeter face C:		
UYZ	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.C</p> <p>Machining:</p> <p>TR D1</p> <p>X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UVZ 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	Finishing allowance in the plane [mm]
UX	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.C</p> <p>Machining:</p> <p>TR D1</p> <p>X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UVZ 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	Finishing allowance at the bottom
Perimeter face Y:		
C0	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.Y</p> <p>Machining:</p> <p>TR D1</p> <p>CB 0.000 ° X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UVZ 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	Reference point [Degree]
X0	<p><b>Residual material</b></p> <p>T D1 F 0.000 mm/min V 0 m/min Per.surf.Y</p> <p>Machining:</p> <p>TR D1</p> <p>CB 0.000 ° X0 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UVZ 0.000 mm UX 0.000</p> <p>Lift mode: To return plane</p>	Reference point in X-direction [abs]

## Section 8

Milling of contour pockets and removal of any residue material

Parameter	Help picture	Description and hints	Notes
Perimeter face Y:			
X1  Residual material T D1 F 0.000 mm/min V 0 m/min Per.surf.Y Machining: TR D1 CB 0.000 ° XB 0.000 abs X1 0.000 inc DY2 0.000 mm DX 0.000 UYZ 0.000 mm UX 0.000  Lift mode: To return plane		Depth as referred to X0 [abs, inc]	
DYZ, DX, UYZ, UX		See Perimeter face C	
General parameters:			
Lift mode  Residual material T D1 F 0.000 mm/min V 0 m/min Face C Machining: TR D1 ZB 0.000 abs Z1 0.000 inc DXY 0.000 mm DZ 0.000 UXY 0.000 mm UZ 0.000  Lift mode: To return plane		Lift mode before a renewed infeed  If for the machining several entry points are necessary, state the height at which the tool is to move on to the next point of entry:  To retraction level  Z0 + Safety distance (Face C and Face Y) or X0 + Safety distance (Perimeter face C and Perimeter face Y)  If within the confines of the pockets there are no islands higher than Z0 (X0), then Z0 + Safety distance (X0 + Safety distance) can be programmed as Lift mode	
Hints for machining:			
Finishing a contour pocket		If when clearing the pocket a finishing allowance was programmed to be left on the bottom and/or the edge, the pocket still has to be finished.  For finishing and/or the edge a single block must be programmed for each. The pocket will be machined only once..  If an island is present, ShopTurn will take this into account in the same way as for roughing .	

## Section 9

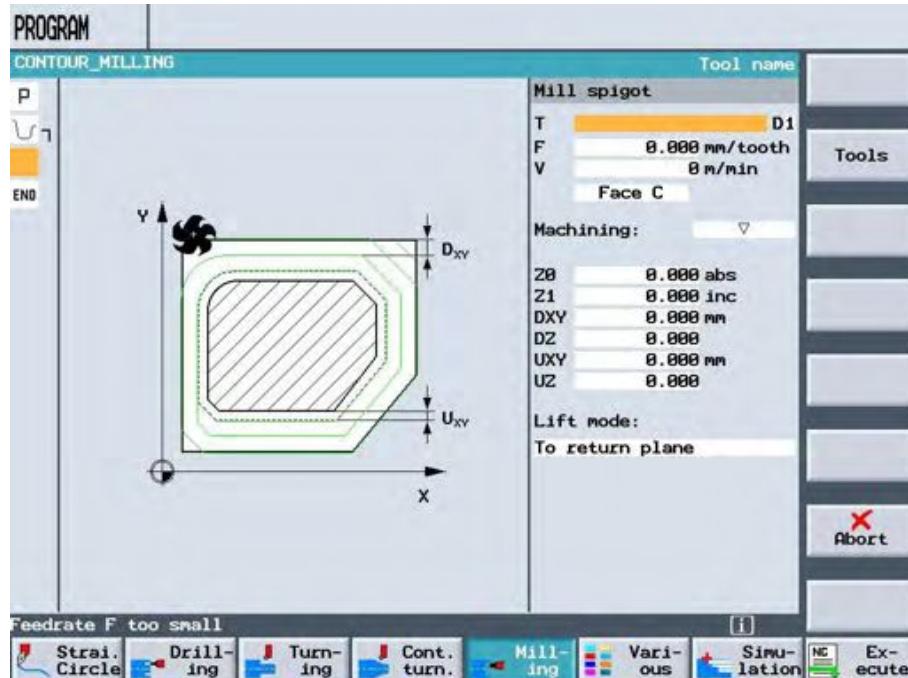
Milling of spigots and removal of any residue material

Notes

### 9.1 Milling a spigot

Mill  
spigot

By pressing the VSK 6 "Mill spigot" the function for milling a spigot contour is selected.



#### Hints for machining

1. Enter the raw material contour  
The raw material contour describes the range outside of which there is no more material; i.e. outside this range motions take place with rapid traverse .
2. Enter one or more spigot contours.  
During machining the material between the raw material and the spigot contour is removed.

If only one raw material contour is programmed and no second contour for a spigot, the raw material contour can be milled as a planar face.

## Section 9

Milling of spigots and removal of any residue material

### Approach/Leaving strategies

1. The tool traverses at rapid traverse rate at the height of the retraction level to the Start point and then approaches to the safety distance. The Start point is calculated by Shop-Mill.
2. Firstly the tool approaches to the machining depth and then the spigot contour from the side with a quarter circle at machining feed rate.
3. The spigot is then cut out parallel to the contour from the outside inwards. The direction is determined by the machining rotation (conventional or climb milling).
4. Once the spigot is cleared at one level the tool leaves the contour with a quarter circle and the infeed to the next machining level ensues.
5. The spigots is again approached with a quarter circle and cut out parallel to the contour from the outside inwards.
6. Step 4 and 5 are repeated until the programmed spigot depth is reached.
7. The tool retracts with rapid traverse to the

Notes

Parameters for the function “Milling of spigots”

Parameter	Help picture	Description and hints
T, D, F, S / V		<i>See module A115 – “Operation sector Program - Programming Centring, Drilling, Thread cutting, Positions“, Section 3 “Drilling centric“.</i>
Location, Machining, Z0, Z1, DXY, DZ, UXY, UZ, X0, X1, DYZ, DX, UYZ, UX, CP, C0 and Lift mode		<i>See 8.1 Contour pocket milling</i>

## Section 9

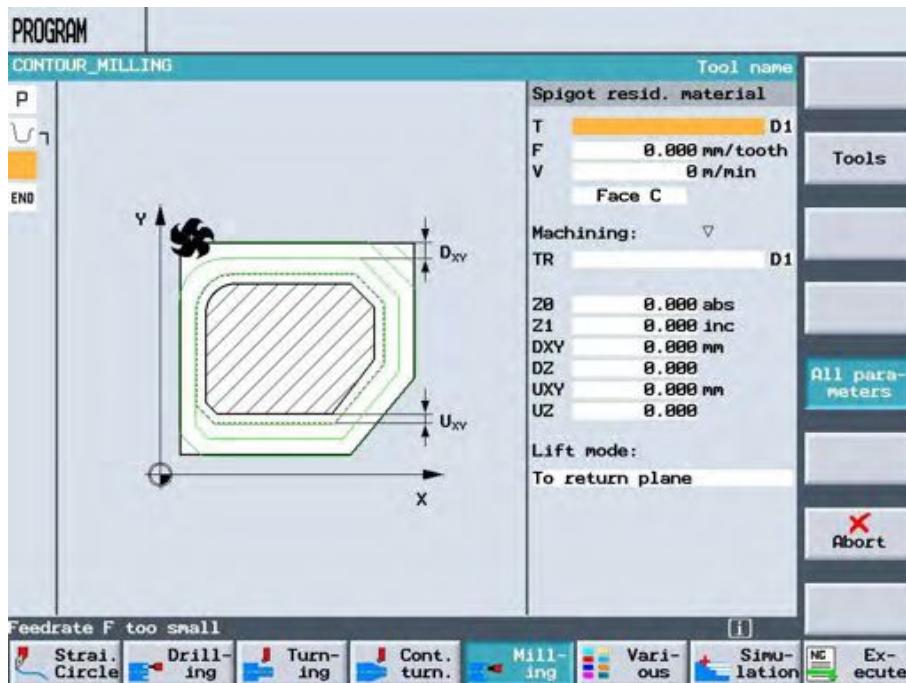
Milling of spigots and removal of any residue material

Notes

### 9.2 Clearing of residue material (Option)

Spigot  
res. mat.

By pressing the VSK 7 "Spigot res.mat." the function "Cleaning-up of residue material on spigot contour" is selected.



If a spigot has been milled and residue material has been left, ShopTurn will recognize this automatically. With a suitable tool this residue material can be removed without having to re-machine the entire spigot, i.e. unnecessary dummy runs are avoided. The calculation of the residue material is carried out on the basis of the milling cutter used for cutting out the spigot.

#### Hints:

Material, which has been left due to the finishing allowance is no residue material.

If several spigots have to be milled and unnecessary tool changes are to be avoided, it is sensible to cut out all spigots followed by the removal of the residue material. In this case the tool, that was used for clearing the spigot must be entered into the parameter "Reference tool TR" when clearing the residue material. For this the VSK 5 "All parameters" must be pressed.

## Section 9

Milling of spigots and removal of any residue material

Parameter for the function “Clearing of residue material”

Notes

Parameter	Help picture	Description and hints
T, D, F, S / V		<i>See module A115 – “Operating Area Program - Programming Centring, Drilling, Thread cutting, Positions“, Section 3 “Drilling centric“.</i>
Location, Machining, TR, D, Z0, Z1, DXY, DZ, UXY, UZ, X0, X1, DYZ, DX, UYZ, UX, CP, C0 and Lift mode		<i>See 8.2 Clearing of residue material</i>

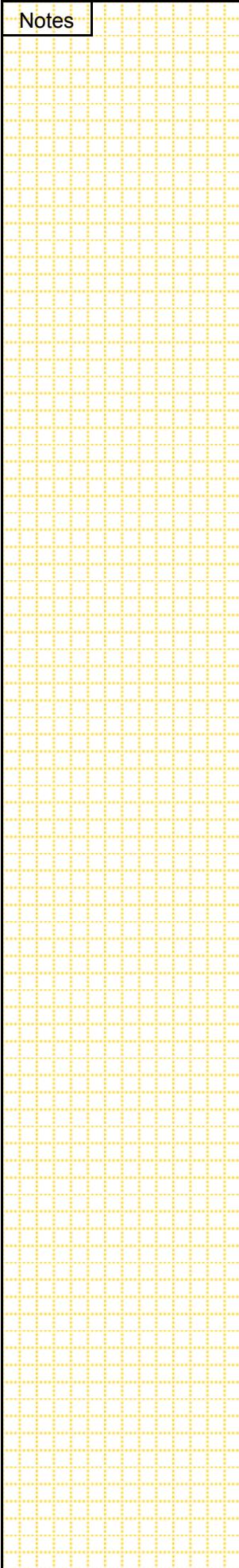
Example for programming:

The screenshot shows a CNC program editor with the following structure:

- P N5 CONTOUR\_MILLING**
- N10 CONTOUR\_BLANK\_1**
- N15 CONTOUR\_SPIGOT\_1**
- N20 Mill spigot**
- N25 CONTOUR\_BLANK\_2**
- N30 CONTOUR\_SPIGOT\_2**
- N35 Mill spigot**
- N70 CONTOUR\_BLANK\_1**
- N75 CONTOUR\_SPIGOT\_1**
- N50 Spig. res. mat.**
- N85 CONTOUR\_BLANK\_2**
- N90 CONTOUR\_SPIGOT\_2**
- N65 Spig. res. mat.**
- END Program end**

1. Contour Blank 1
2. Contour Spigot 1
3. Mill Spigot 1
4. Contour Blank 2
5. Contour Spigot 2
6. Mill Spigot 2
7. Contour Blank 1
8. Contour Spigot 1
9. Residue material spigot 1 clearing
10. Contour Blank 2
11. Contour Spigot 2
12. Residue material spigot 2 clearing

Notes



# Operating Area Program Editor - Programming of Various “Set mark / Sub-program”

## 1 Brief description

### Objective of the module:

This module explains the functions 'Set mark', "Repetition" and "Sub-program" in the range "Various".

### Description of the module:

The 1st section of this module deals with the general functions under "Various" in order to help understanding. The other functions available in this mask are described in separate modules:

Counter spindle (See *Module A131 Operating Area Program - Programming Various - Counter spindle*)

Settings (See *Module A133 Operating Area Program -Settings*)

Transformations (See *Module A134 Operating Area Program - Programming Transformation*)

This module deals only with the functions "Set mark", "Repetition" and "Sub-program".

These functions are helpful, if certain program steps must be carried out repeatedly when machining a work piece (set mark and repetition) or on different work pieces (sub-program).

### Content of the module:

#### Functions under "Various"

- Pre-selection "Various"

- Vertical Softkey strip

Set mark and repetition

Sub-program

Functions under "Various"  
Pre-selection "Various"  
Vertical Softkey strip

Set mark and repetition

Sub-program



Section 2



Section 3



Section 4

## Section 2

Notes

Functions under “Various“

### 2.1 Pre-selection “Various“



By pressing the HSK 5 “Various“ the range for the programming of additional programming functions is called up.



### 2.2 Vertical Softkey strip



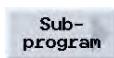
By pressing the VSK 1 “Set mark“ a mark for a repetition can be set.

See Section 3 “Set mark and Repetition“.



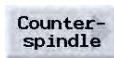
By pressing the VSK 2 “Repetition“ a program section can be taken over.

See Section 3 “Set mark and Repetition“.



By pressing the VSK 3 “Sub-program“ a Sub-program in the present program can be called up.

See Section 4 “Sub-program“.



By pressing the VSK 5 “Counter-spindle“ the process for the machining of the rear end of the work piece can be called up.

See Module A131 Operating Area Program - Programming Various - Counter spindle“.

## Section 2

### Functions under “Various“

Settings

By pressing the VSK 7 “Settings“ the window for setting of the:  
retraction parameter  
tool changing point  
safety distance  
Speed limits and  
The machining sense of rotation  
will be called up.

Here the parameters of the program header with the exception of the raw material description can be changed.

*See Module A133 - “Operating Area Program - Programming Various - Settings“.*

Notes

Transformations

By pressing the VSK 8 “Transformations“ the following transformations can be called up:

- Zero point offset
- Offset
- Rotation
- Scaling
- Mirroring
- Rotation C-axis

*See module A134 - “Operating Area Program - Programming Various - Transformations“.*

## Section 3

Notes

### Set mark and repetition

If for a machining sequence of a work piece certain steps have to be executed several times it is sufficient to program these just once. ShopTurn permits subsequently to repeat any steps that have already been programmed.

#### Hits:

The program blocks that are to be repeated must be identified by a Start and an End mark.

These program blocks can then be repeated within a program up to 9999 times.

The marks must all feature unmistakable, i.e. different names.

Marks and Repetitions can be added as an afterthought, but not within linked program blocks (e.g. contour milling).

It is possible to use the same mark as an end-mark of previous program blocks and also as start-mark for subsequent program blocks.



## Section 4

### Sub-program

If the same machining steps are required for the programming of other work pieces, these steps can be defined as an individual Sub-program. This Sub-program can then be called up in any other program. Thus the multiple programming of identical machining steps is unnecessary.

Notes

#### Hints:

ShopTurn does not differentiate between main and Sub-programs; i.e. a linked step or G-code program can be called up in another linked step program as a Sub-program.

Within one Sub-program it is possible to call up yet another Sub-program. The maximum depth of encapsulation is 8 Sub-programs.

No Sub-programs may be inserted within linked blocks. If a linked step program is to be called up as a Sub-program, the program must have been evaluated previously (the program must be loaded or simulated in the mode of operation "Machine Auto").

The Sub-program must always be stored in the NC working memory (in a separate directory "Test\_1" or else in one of the directories "ShopTurn", "Part programs", "Sub-programs").

The path does not need to be stated in the input mask for the Sub-program unless the Sub-program is not stored in the same directory as the main program.

#### Paths to be stated

#### Directory

ShopTurn

Own directory

Part programs

Sub-programs

#### Path to be stated

ShopTurn

Test\_1

MPF

SPF

If a sub-program must be called up, which is stored on another medium (e.g. external hard disk), the G-code command "EXTCALL" can be used.

## Section 4

### Sub-program

Notes

The file endings (\*.mpf or \*.spf) need only be stated, if the sub-program does not have the file ending that has been defaulted for the directory in which the Sub-program is stored.

#### Preset file endings

#### Directory

#### Preset file ending

ShopTurn	*.mpf
Own directory	*.mpf
Part programs	*.mpf
Sub-programs	*.spf

# Operating Area Program - Programming Various - Counter spindle

## 1 Brief description

### Objective of the module:

This module explains the programming of the function “Counter spindle” in the section “Various”.

### Description of the module:

This module describes only the function “Counter spindle”.

This function is useful if work pieces must be machined at both ends with turning, drilling and milling operations without the need of manual re-loading of the work piece.

### Content of the module:

Functions under “Various”

- Selection “Various”
  - Vertical Softkey-strip
- Counter spindle
- Gripping
  - Pulling
  - Rear end
  - Front end
  - Complete

Functions under “Various”  
Selection “Various”  
Vertical Softkey-strip



Counter spindle  
Gripping  
Pulling  
Rear end  
Front end  
Complete



## Section 2

Notes

Functions under “Various“

### 2.1 Selection “Various“



By pressing the HSK 5 “Various“ the range for the programming of additional programming functions is called up.



### 2.2 Vertical Softkey-strip



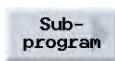
By pressing the VSK 1 “Set mark“ a mark for a repetition can be set.

See Module A127 “Operating Area Program - Programming Various - Set mark / Sub-program“.



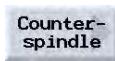
By pressing the VSK 2 “Repetition“ a repetition of a previously marked program section can be inserted.

See Module A127 “Operating Area Program - Programming Various - Set mark / Sub-program“.



By pressing the VSK 3 “Sub-program“ a sub-program in the current program can be called up.

See Module A127 “Operating Area Program - Programming Various - Set mark / Sub-program“.



By pressing the VSK 5 “Counter spindle“ the process for the machining of the rear end of a work piece can be called up.

See Section 3 “Counter spindle“.

## Section 3

### Counter spindle

If the turning machine (lathe) features a counter spindle, the work piece can be machined on both the front and the rear end with turning, drilling and milling functions without having to re-load the work piece manually. Before machining the rear end, the work piece must be gripped, pulled out of the main spindle chuck and removed to the new machining position. These operations can be programmed with the function "Counter spindle". For programming ShopTurn provides the following 5 steps:

Gripping: grip the work piece with the counter spindle

Pulling: pull the work piece with the counter spindle from the main spindle

Rear end: Move the work piece with the counter spindle to the new machining position

Front end: Work offset for the machining of the new front end (in case of bar material)

Complete: The steps Gripping, Pulling (if necessary with parting) and Rear end

When a program is started for the counter spindle machining, the counter spindle is first of all retracted to the retraction position specified by a machine datum.

For the functions "Pulling" and "Rear end" it must be stated into which work offset ShopTurn is to store the associated co-ordinate system; i.e. this work offset needs not to be defined beforehand. In case of the "Front end" the work offset, that is to be used, must be defined separately.

#### Caution

The work offsets stored in the functions "Pulling" and "Rear end" will also be changed if just a search run or simulation is started. These work offsets should therefore not be used at the same time for other purposes, since this could lead to problems. In order to ease the programming the following programming suggestions are provided for three typical uses:

Machining with the main spindle – transfer of the work piece – machining with the counter spindle

Machining with the counter spindle (without a previous work piece transfer)

Machining of bar material

Notes

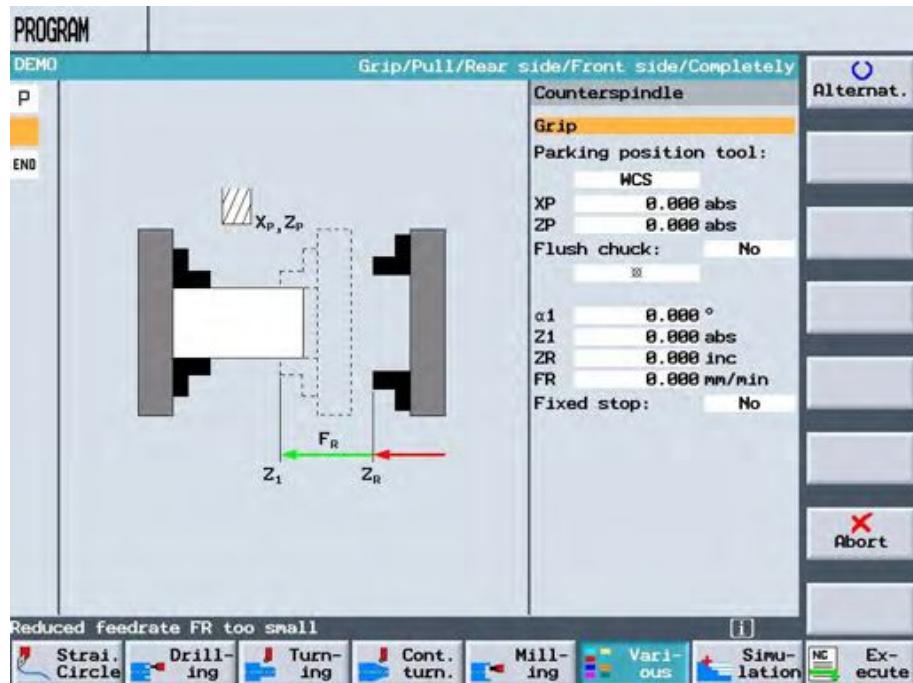
## Section 3

Notes

### Counter spindle

#### 3.1 Gripping

First of all ShopTurn synchronizes the main and the counter spindle and positions the tool at the selected parking position XP and ZP. Then the counter spindle moves with rapid traverse to the programmed position ZR towards the work piece and then with reduced feed rate FR to the transfer position Z1. Whether the counter spindle is to approach with front edge or the positioning edge to this position must be defined in the mask „Spindles“. Alternatively to this the counter spindle moves to a fixed stop as from a certain distance. This distance and the respective feed rate are specified in the machine data.



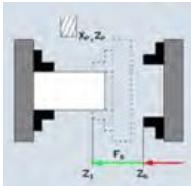
#### Hints:

The documentation by the machine manufacturer must be observed.

If an angular offset  $\alpha_1$  is stated such that the work piece can be gripped better, this will not have any effect on the subsequent machining of the rear end of the work piece.

## Section 3

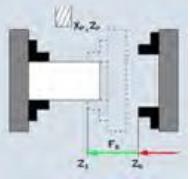
### Counter spindle

Parameter	Help picture	Description and Hints	Notes
WCS / MCS		Parking position: WCS / MCS  Hint: In case of the selection MCS it is possible to teach the parking position by means of the VSK 6 "Teach park pos.".	
XP, ZP		Parking position of the tool in X direction, Parking position of the tool in Z direction  [abs]	
Chuck flush		Flush the counter spindle chuck YES or NO	
Direction of rotation		Direction of rotation of the main spindle (and counter spindle):  <span style="background-color: orange; padding: 2px;">✖</span> Spindles are stationary <span style="background-color: orange; padding: 2px;">↺</span> Anti-clockwise <span style="background-color: orange; padding: 2px;">↻</span> Clockwise	
S		Spindle speeds (main spindle and counter spindle)	

## Section 3

### Counter spindle

Notes

Parameter	Help picture	Description and Hints
$\alpha_1$		Angular offset of the counter spindle being Gripping
Z1		Transfer position [abs]
ZR		Position from which the approach is carried out with reduced feed rate [abs, inc]
FR		Reduced feed rate
Fixed stop		<p>YES: The counter spindle stops at a certain distance before the transfer position Z1 and carries on from there with a fixed feed rate to the fixed stop.</p> <p>NO: The counter spindle moves on to the transfer position Z1.</p>

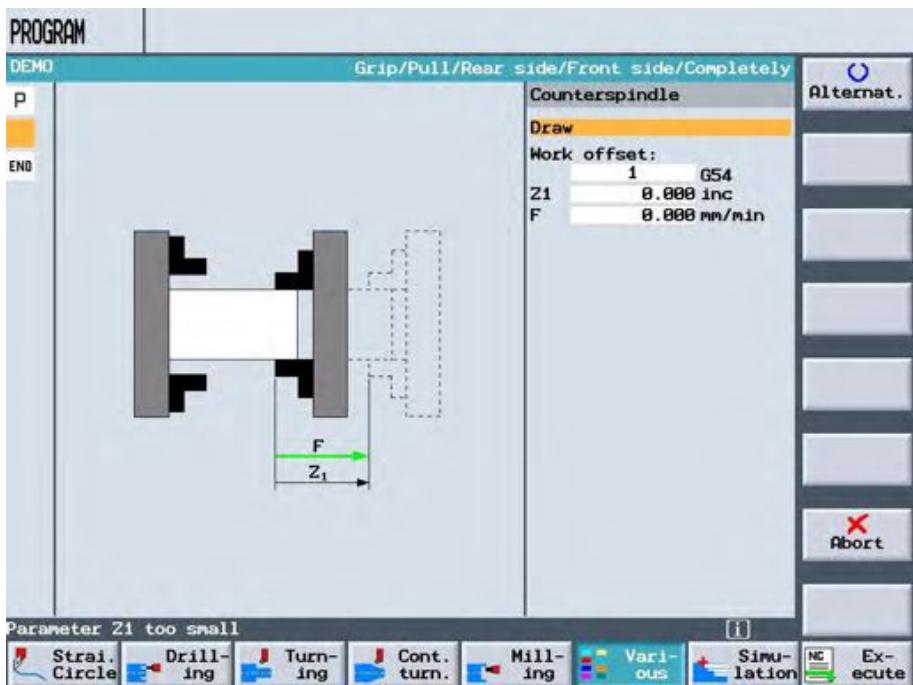
## Section 3

### Counter spindle

#### 3.2 Pulling

The counter spindle pulls the work piece by the amount Z1 from the main spindle. ShopTurn takes along the co-ordinate system and stores the offset in the selected work offset.

Notes



Parameter	Help picture	Description and Hints
Work offset.		Work offset in which the co-ordinate system, shifted by Z1, is to be stored.
Z1		Amount by [inc]
F		Feed rate, with which the work piece is to be pulled from the main spindle.

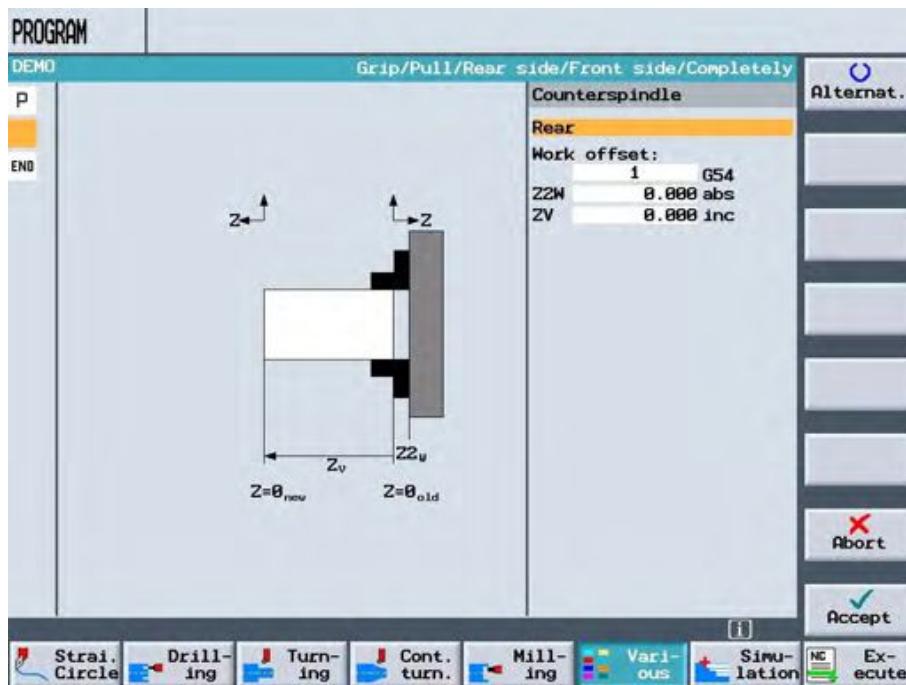
## Section 3

Notes

### Counter spindle

#### 3.3 Rear end

The counter spindle moves with the work piece at rapid traverse rate to the new machining position Z2W. The work piece zero point is taken along in this case and shifted by the amount ZV (with sign) from the front end to the rear end of the work piece. Following this the co-ordinate system for the machining is mirrored to the rear end and stored in the selected work offset. The synchronization for the two spindles is cancelled. The counter spindle is now the master spindle.



#### Hints:

ShopTurn automatically mirrors the co-ordinate system for the machining of the rear end; i.e. the programming for the rear end is carried out exactly the same way as for the front end.

The following points must be borne in mind for the program step "rear end", if the machining is carried out with the counter spindle without a preceding transfer of the work piece. The work offset, which is selected in the parameter mask will be simply transferred and not evaluated; i.e. the work piece zero point should be stored in the work offset for the counter spindle machining. Furthermore the parameter ZV will not be used.

## Section 3

### Counter spindle

Parameter	Help picture	Description and Hints	Notes
Work offset.		Work offset in which the co-ordinate system , which has been shifted by ZV to Z2W, is to be stored.	
Z2W		Machining position additional axis [abs]	
ZV		Offset of the work piece zero point in the Z direction [inc]	

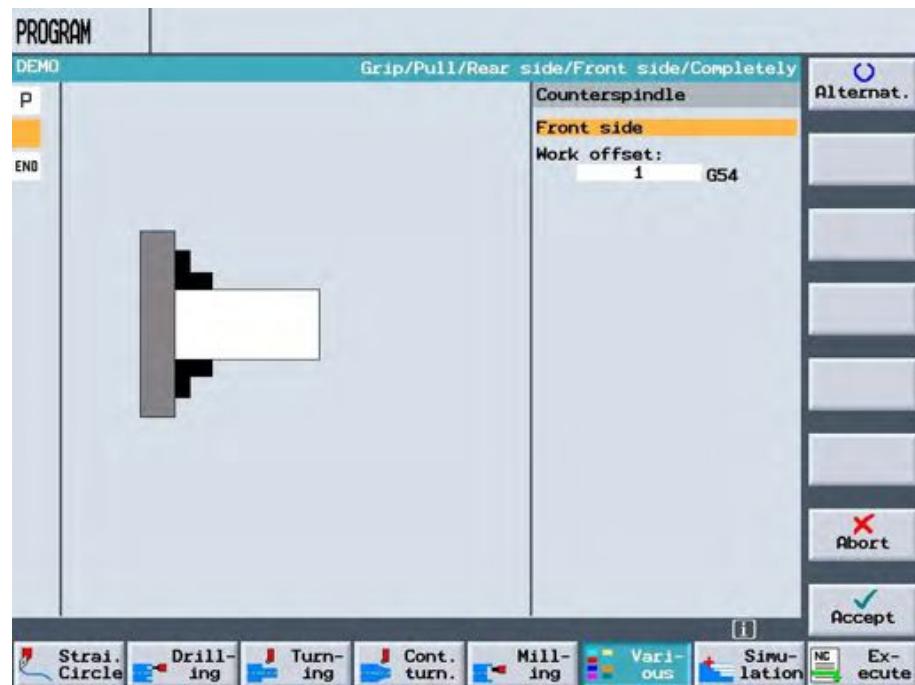
## Section 3

### Counter spindle

Notes

#### 3.4 Front end

Once the machining on the rear end of a work piece has been completed, The machining of the next work piece on the front end commences. Between these a work offset for the machining on the front end can be called up with the function "Front end". It is usual to use the work offset that was operative before gripping was activated. The main spindle is once more the master spindle and the previously mirrored Z-axis is now re-instated.



Parameter	Help picture	Description and Hints
Work offset.  Counterspindle Front side Work offset: 1 G54		Work offset for the machining of the next front end.

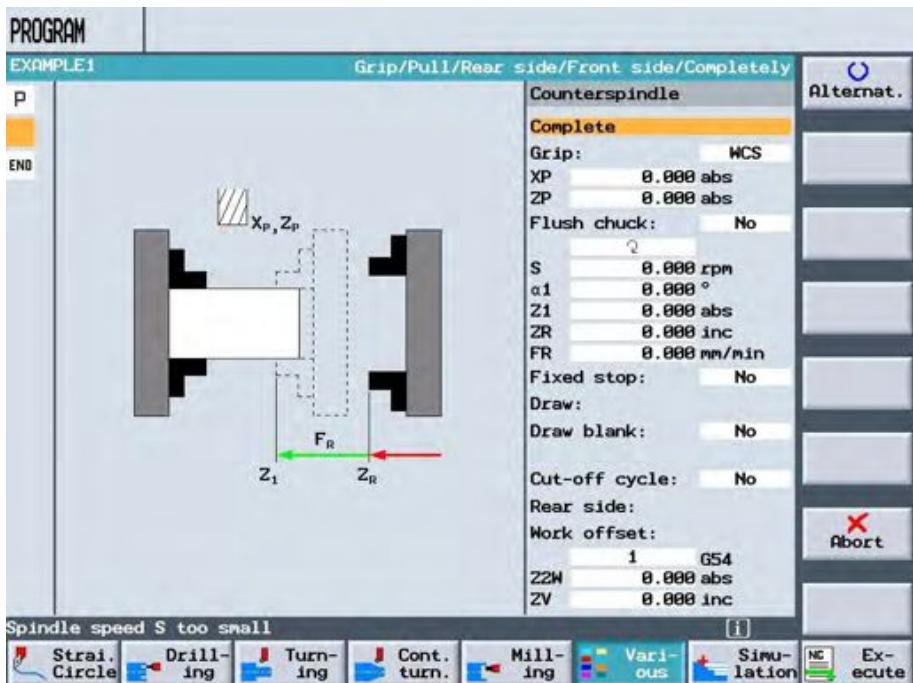
## Section 3

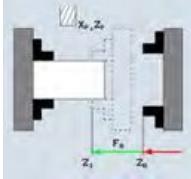
### Counter spindle

#### 3.5 Complete

If the program step "Complete" is programmed and if the raw material used is bar material, it will be necessary to enter "Raw material pulling: YES" and "Parting cycle: Yes" for the step "Pulling". Following this the function „Parting“ must be programmed. The parting of the work piece will then follow after gripping and pulling from the main spindle. The amount by which the work piece is to be pulled out - it will be evaluated from the parameters entered for the parting cycle. The two program blocks "Complete" and "Parting" will be linked in the work plan.

Notes

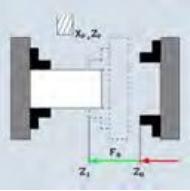


Parameter	Help picture	Description and Hints
<b>WCS / MCS</b>  <b>Counterspindle</b> <b>Complete</b> Grip: <b>WCS</b> XP: 0.000 abs ZP: 0.000 abs Flush chuck: No S: 0.000 rpm alpha1: 0.000 ° Z1: 0.000 abs ZR: 0.000 inc FR: 0.000 mm/min Fixed stop: No Draw: No Draw blank: No Cut-off cycle: No Rear side: No Work offset: 1 G54 Z2W: 0.000 abs ZV: 0.000 inc		Parking position: WCS / MCS  Hint: In case of the selection MCS it is possible to teach the parking position by means of the VSK 6 "Teach park pos.".

## Section 3

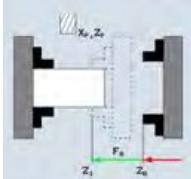
### Counter spindle

Notes

Parameter	Help picture	Description and Hints
<b>XP, ZP</b>		Parking position of the tool in X direction, Parking position of the tool in Z direction [abs]
<b>Chuck flush</b>		Counter spindle chuck flush YES or NO
<b>Direction of rotation</b>		Direction of rotation of the main spindle (and counter spindle):   Spindles are stationary  Anti-clockwise  Clockwise

## Section 3

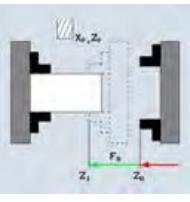
### Counter spindle

Parameter	Help picture	Description and Hints	Notes
S		Spindle speed (main spindle and counter spindle)	
$\alpha_1$		Angular offset of the counter spindle for gripping	
Z1		Transfer position [abs]	

## Section 3

### Counter spindle

Notes

Parameter	Help picture	Description and Hints
ZR		Position from which onwards the motion takes place with reduced feed rate [abs or inc]
FR		Reduced feed rate
Fixed stop		<p>YES: The counter spindle stops at a certain distance before the transfer position Z1 and carries on from there with a fixed feed rate to the fixed stop.</p> <p>NO: The counter spindle moves on to the transfer position Z1.</p>

## Section 3

### Counter spindle

Parameter	Help picture	Description and Hints	Notes
<p>Raw material pulling</p> <p><b>Counterspindle</b></p> <p><b>Complete</b></p> <p>Grip: WCS</p> <p>XP 0.000 abs</p> <p>ZP 0.000 abs</p> <p>Flush chuck: No</p> <p>S 0.000 rpm</p> <p><math>\alpha_1</math> 0.000 °</p> <p>Z1 0.000 abs</p> <p>ZR 0.000 inc</p> <p>FR 0.000 mm/min</p> <p>Fixed stop: No</p> <p>Draw:</p> <p>Draw blank: No</p> <p>Cut-off cycle: No</p> <p>Rear side:</p> <p>Work offset:</p> <p>1 G54</p> <p>Z2H 0.000 abs</p> <p>ZV 0.000 inc</p>		<p>YES: Pulling the raw material by the raw material length (preparation for the next work piece)</p> <p>NO: No pulling of raw material</p> <p>Hint: The function "Raw material pulling" is available only in conjunction with "Parting cycle: YES".</p>	
F		<p>Feed rate for pulling</p> <p>[mm/min]</p> <p>Hint: The value entered here will only be activated if subsequently "Parting cycle: YES" is selected.</p>	
Parting cycle		<p>YES: After pulling the work piece will be parted off.</p> <p>NO: There will be no automatic parting.</p>	

## Section 3

### Counter spindle

Notes

Parameter	Help picture	Description and Hints
<b>Work offset</b> <pre> Counterspindle Complete Grip:           WCS XP:            0.000 abs ZP:            0.000 abs Flush chuck:   No S:             0.000 rpm a1:            0.000 ° Z1:            0.000 abs ZR:            0.000 inc FR:             0.000 mm/min Fixed stop:    No Draw: Draw blank:   No Cut-off cycle: No Rear side: Work offset:   1 GS4 Z2W:          0.054 abs ZV:            0.000 inc </pre>	<p>Work offset in which the co-ordinate system , which has been shifted by ZV to Z2W, is to be stored.</p>	
<b>Z2W</b> <pre> Counterspindle Complete Grip:           WCS XP:            0.000 abs ZP:            0.000 abs Flush chuck:   No S:             0.000 rpm a1:            0.000 ° Z1:            0.000 abs ZR:            0.000 inc FR:             0.000 mm/min Fixed stop:    No Draw: Draw blank:   No Cut-off cycle: No Rear side: Work offset:   1 GS4 Z2W:          0.000 abs ZV:            0.000 inc </pre>	<p>Machining position additional axis (abs)</p> <p>Special case: Z2W = Ø In this case the counter spindle will be traversed at rapid traverse rate to the retraction position specified in a machining datum.</p>	
<b>ZV</b> <pre> Counterspindle Complete Grip:           WCS XP:            0.000 abs ZP:            0.000 abs Flush chuck:   No S:             0.000 rpm a1:            0.000 ° Z1:            0.000 abs ZR:            0.000 inc FR:             0.000 mm/min Fixed stop:    No Draw: Draw blank:   No Cut-off cycle: No Rear side: Work offset:   1 GS4 Z2W:          0.000 abs ZV:            0.000 inc </pre>	<p>Offset of the work piece zero point in Z direction (inc, the sign will also be taken into account)</p>	

## 1 Brief description

**Objective of the module:**

This module explains the programming of the functions “Settings” in the sector “Various”.

**Description of the module:**

All parameters defined in the program heading, with the exception of the dimensional units, can be altered at any place in the program. Additionally there is the possibility to change the basic setting for the sense of rotation for milling.

The settings in the program heading are self-maintaining, i.e. they remain active until such time as they are altered. Alterations of the settings for the raw material can be sensible for instance to vary the visible section of the simulation.

Further uses for changing the settings are the functions:

- Zero point offsets
- Co-ordinate transformations
- Cylinder mantle transformation

**Content of the module:**

- Selection and parameters of the function “Settings”
  - Selection of the function “Settings”
  - parameters for the function “Settings”

Selection and parameters for the function “Settings”

Selection of the function  
“Settings”

Parameters for the function  
“Settings”



Section 2

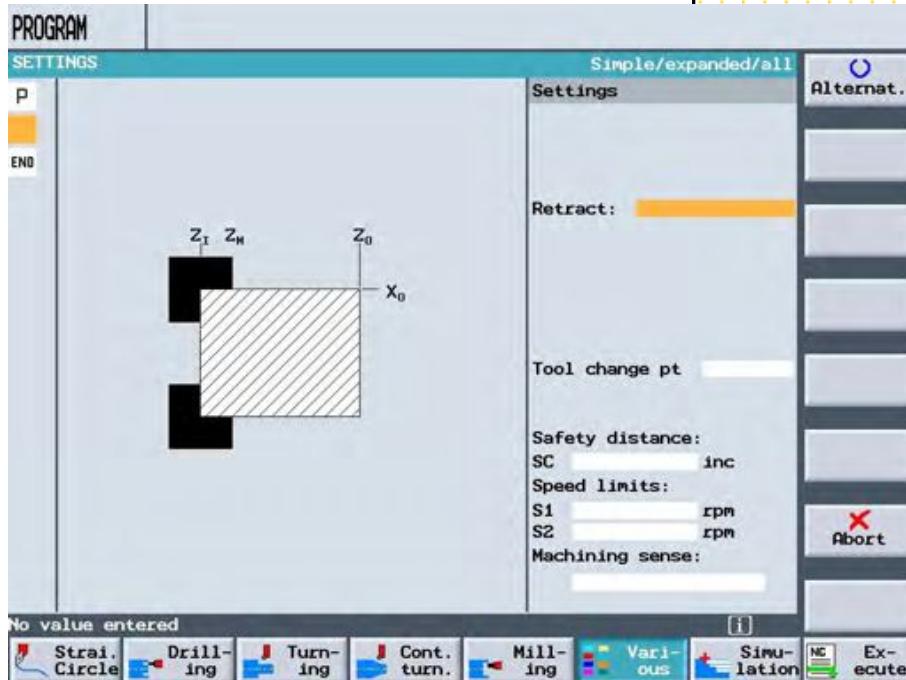
## Section 2

### Selection and parameters for the function “Settings”

#### 2.1 Selection of the function “Settings”

**Settings**

By pressing the VSK 7 "Settings" select the range alteration of the settings.



#### 2.2 Parameters for the function “Settings”

Parameter	Help picture	Description and Hints
<b>Retraction</b> 		<p>Retraction levels - dependent on the shape of the raw material and the type of retraction (simple, extended or all)</p> <p>XRA: external retraction level in X-direction [abs]</p> <p>XRI: internal retraction level in X-direction [abs, inc]</p> <p>ZRA: external retraction level in the X-direction Z-direction [abs]</p> <p>ZRI: internal retraction level in X-direction Z-direction [abs, inc]</p> <p>Hint: If a new retraction level is defined in a program, it will not be taken into account until after the next tool change.</p>

## Section 2

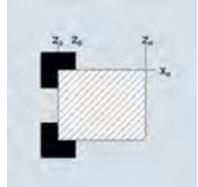
### Selection and parameters for the function “Settings“

Parameter	Help picture	Description and Hints	Notes
Tool changing point		XT, ZT: Co-ordinates of the tool changing point  If the machine co-ordinate system (MCS) has been selected, it is possible to “teach” the tool changing point with the VSK 6 “Tool chg.pt teach“.	
Safety distance		SC: Safety distance [inc]	
Speed limitation		S1: Speed limit main spindle S2: Speed limit counter spindle [rev/min]	

## Section 2

Notes

### Selection and parameters for the function “Settings“

Parameter	Help picture	Description and Hints
Machining sense of rotation		<p>Climb milling / Conventional milling</p> <p>ShopMill takes both the machining sense of rotation into account and also the direction of spindle rotation from the tool list when a pocket, a longitudinal slot or a spigot is being machined. The pocket is then machined either clockwise or anti-clockwise.</p> <p>Hint: The sense of rotation (conventional or climb milling) is the cutting direction of the milling cutter tooth compared to the feed direction of the work piece. This means that ShopTurn takes into account the parameter “Sense of rotation” together with the direction of spindle rotation when milling, with the exception of path milling. The basic setting for the sense of rotation is selected in a machine datum.</p> <p><i>See also Module A123 - "Operating Area - Programming Contour milling", Section 4 "Entering contour elements".</i></p>

# Operating Area Program editor - Programming various - Transformations

## 1 Brief description

### Objective of the module:

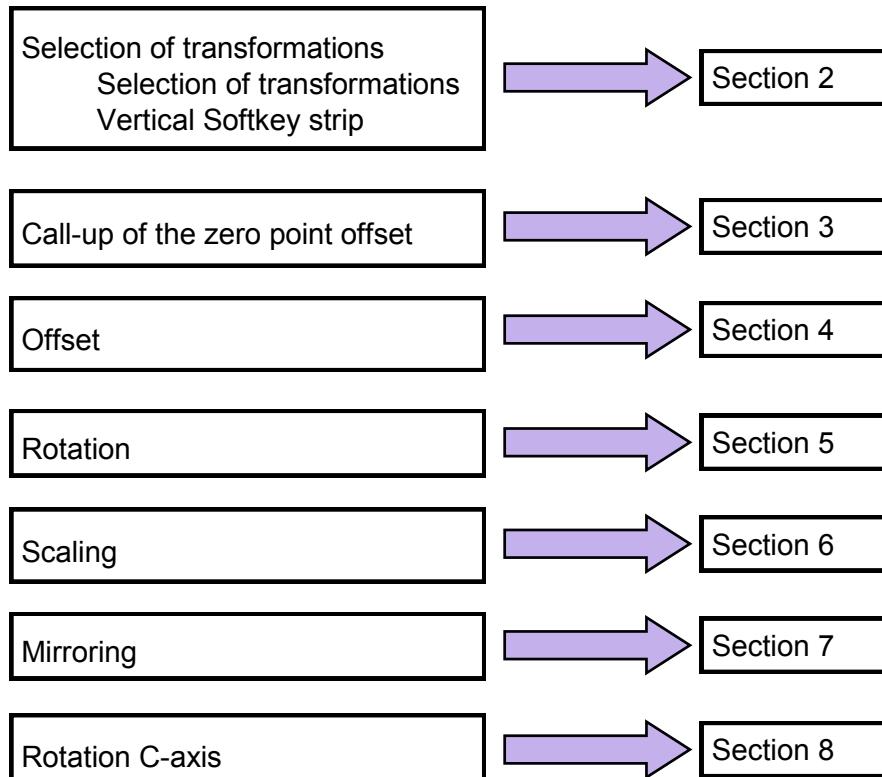
This module explains the use of the function “Settings” in the range “Various”.

### Description of the module:

In this module the functions “Transformations” in the range “Various” under ShopTurn will be explained and the programming described in general terms. The following transformations are available for programming:

### Content of the module:

Selection of transformations  
- Selection of transformations  
- Vertical Softkey strip  
Call-up of the zero point offset  
Offset  
Rotation  
Scaling  
Mirroring  
Rotation C-axis



## Section 2

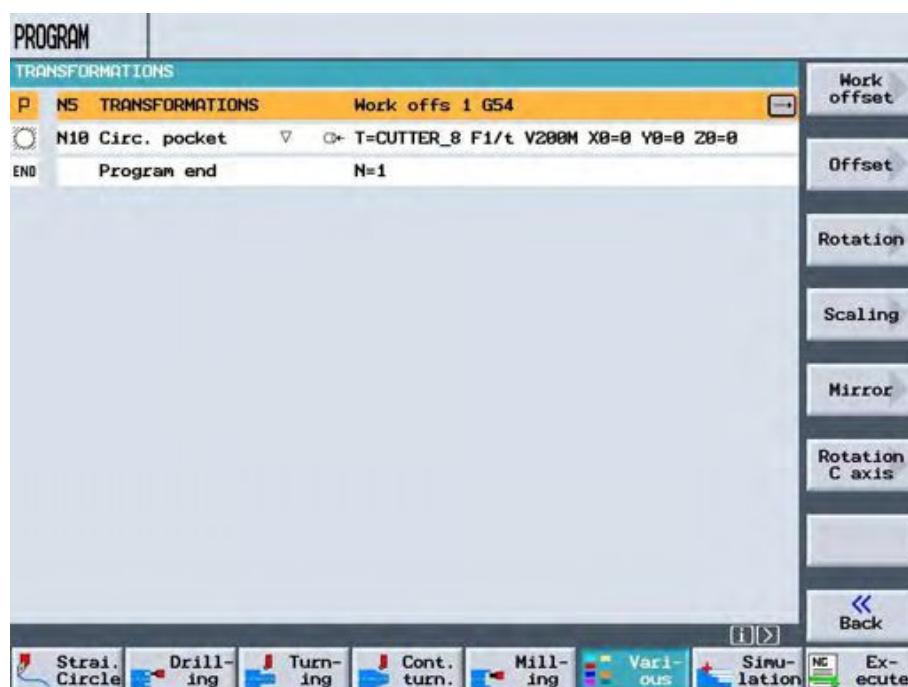
Notes

### Selection of transformations

Transformations

By pressing the VSK 8 “Transformations“ the range for the following transformations can be called up:

- Zero point offset
- Offset
- Rotation
- Scaling
- Mirroring
- Rotation C-axis



### 2.2 Vertical Softkey strip

Work offset

By pressing the VSK 1 “Work offset.“ the input mask for the selection of a zero point offset will be called up.

See Section 3 “Zero point offset“.

Offset

By pressing the VSK 2 “Offset“ the input mask for the programming of an offset will be called up.

See Section 4 “Offset“.

Rotation

By pressing the VSK 3 “Rotation“ the input mask for the programming of a rotation will be called up.

See Section 5 “Rotation“.

## Section 2

### Selection of transformations

Scaling

By pressing the VSK 4 "Scaling" the input mask for the programming of scaling will be called up.  
See Section 6 "Scaling".

Notes

Mirror

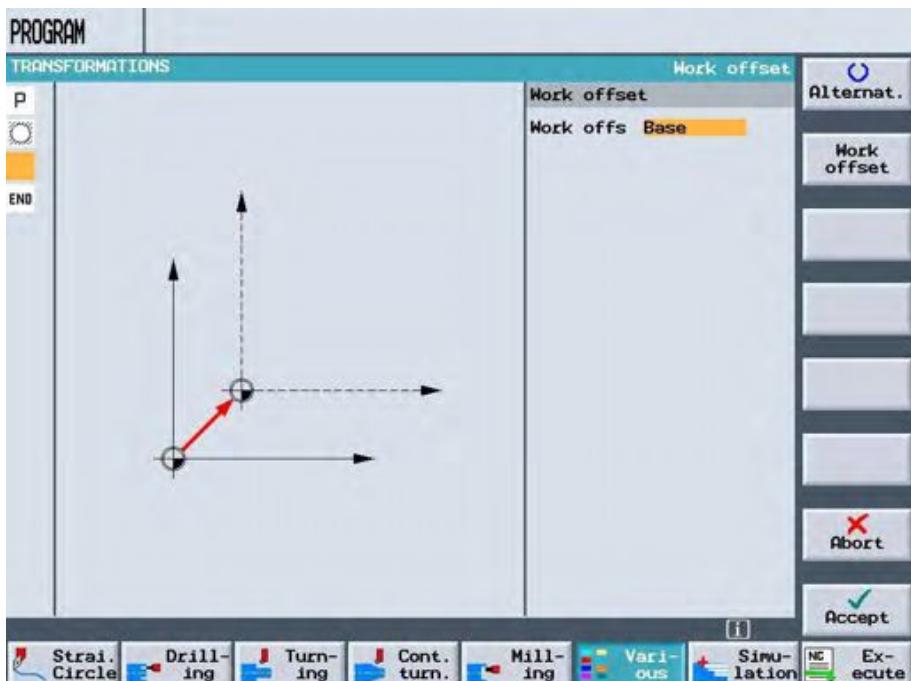
By pressing the VSK 5 "Mirror" the input mask for the programming of a mirroring will be called up.  
See Section 7 "Mirroring".

Rotation  
C axis

By pressing the VSK 6 "Rotation C-axis" the input mask for the programming for "Rotation of the C-axis" is called up.  
See Section 8 "Rotation C-axis".

Work  
offset

By pressing the VSK 1 „Work offset.“ the input mask for the selection of a zero point offset will be called up.



Zero point offsets (G54 etc.) can be called up from any program.

These offsets can be used for instance to machine work pieces with differing raw material dimensions with one and the same program. The offset in this case adapts the work piece zero point to the new raw material.

Alternat.

By pressing the VSK 1 "Alternat." the basic-ZPO (G500), ZPO 1 (G54) etc. can be called up.

Work  
offset

By pressing the VSK 2 "Work offset" the list of zero point offsets will be called up. With the cursor a zero point offset can be selected and transferred to the program by means of VSK 2 "Into program".

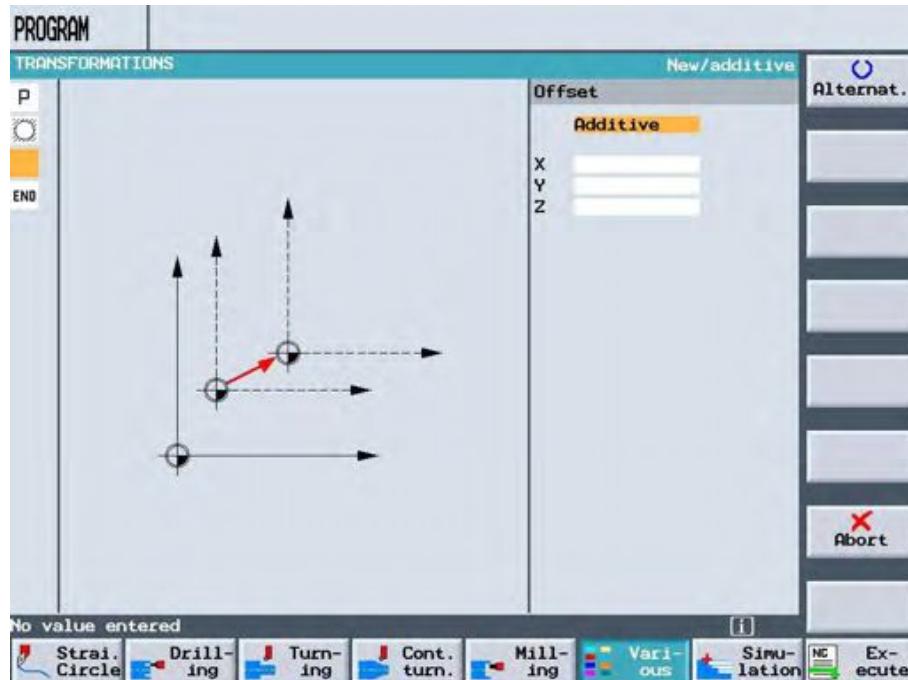
## Section 4

Notes

Offset

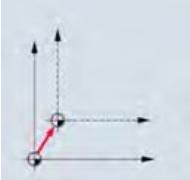
Offset

By pressing the VSK 2 “Offset“ the input mask for the programming of an offset will be called up.



Alternat.

By pressing the VSK 1 “Alternat.“ the function Offset “New“ or “Additive“ will be called up.

Parameter	Help picture	Description and Hints
Offset	 	New Activated zero point offsets will be deactivated, the zero point will be shifted to the original ZPO. Hint: If “New“ is selected, all previous transformations will be deleted.  Additive The zero point will be shifted from the current position by the entered value.
X, Y, Z		Offset X, Y or Z

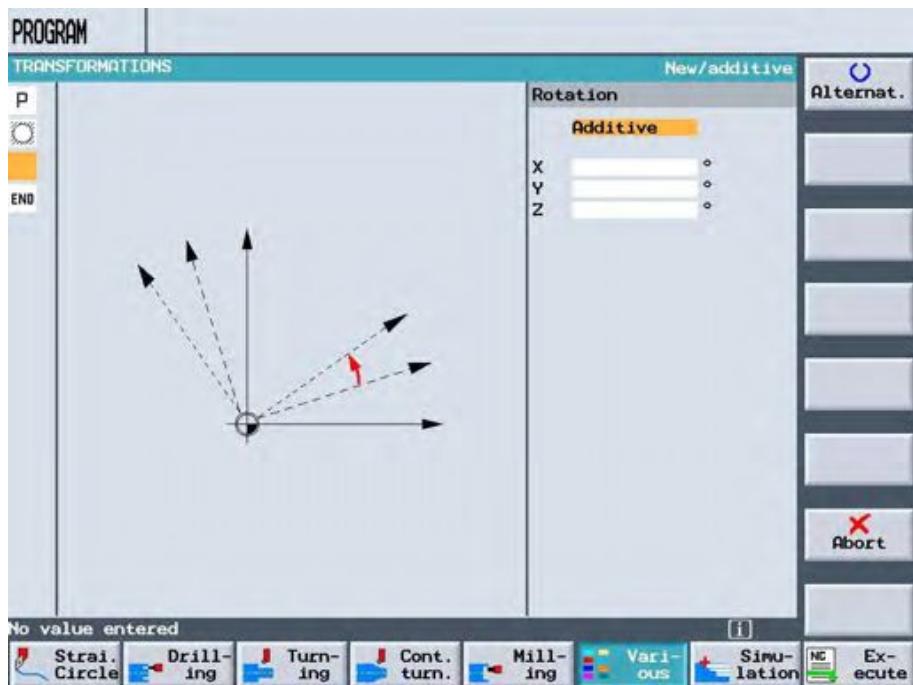
## Section 5

### Rotation

Rotation

By pressing the VSK 3 “Rotation“ the input mask for the programming of a rotation will be called up.

Notes



Alternat.

By pressing the VSK 1 “Alternat.“ the function Rotation “New“ or “Additive“ will be called up.

Parameter	Help picture	Description and Hints
Rotation		<p>New</p> <p>The activated rotation will be de-activated, the zero point will be rotated to the original WO.</p> <p>Hint: If “New“ is selected, all previous transformations will be deleted.</p> <p>Additive</p> <p>The zero point will be rotated by the entered value in addition to the present rotation.</p>
X, Y, Z		Rotation about X, Y or Z.

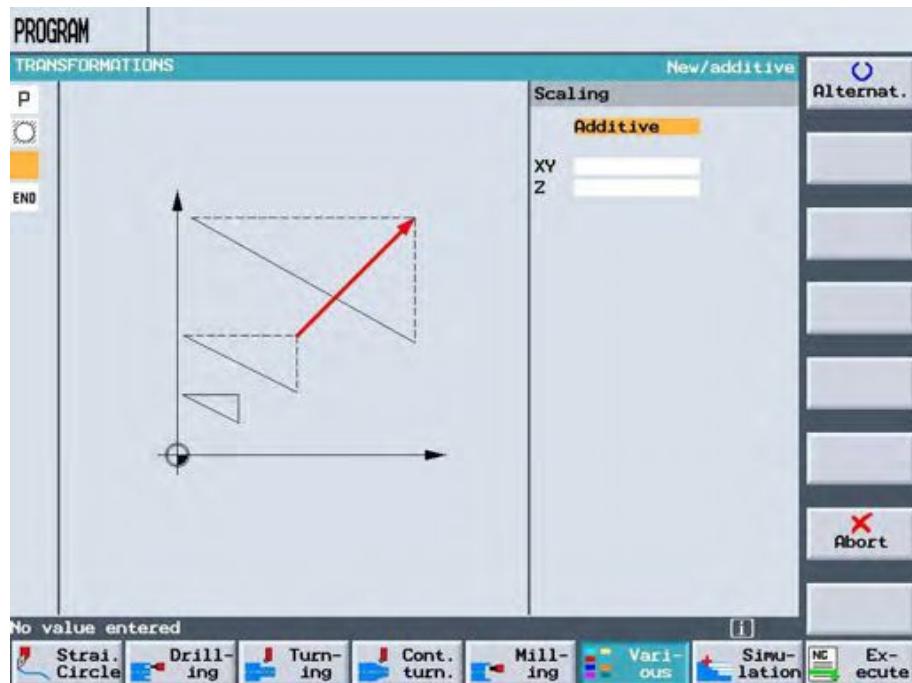
## Section 6

Notes

### Scaling

Scaling

By pressing the VSK 4 "Scaling" the input mask for the programming of scaling will be called up.



Alternat.

By pressing the VSK 1 "Alternat." the function Scaling "New" or "Additive" can be called up.

Parameter	Help picture	Description and Hints
Scaling		<p>New</p> <p>A scale factor can be entered for the active machining plane as well as for the tool axis. Any activated scaling will be de-activated. The programmed co-ordinates will then be multiplied by the stated factor.</p> <p>Hints:</p> <ul style="list-style-type: none"> <li>The scaling always refers to the zero point of the work piece.</li> <li>Hint: If "New" is selected, all previous transformations will be deleted.</li> </ul> <p>Additive</p> <p>The scaling is additive to the existing scaling by the entered amount.</p>
XY, Z		Scaling in XY or Z

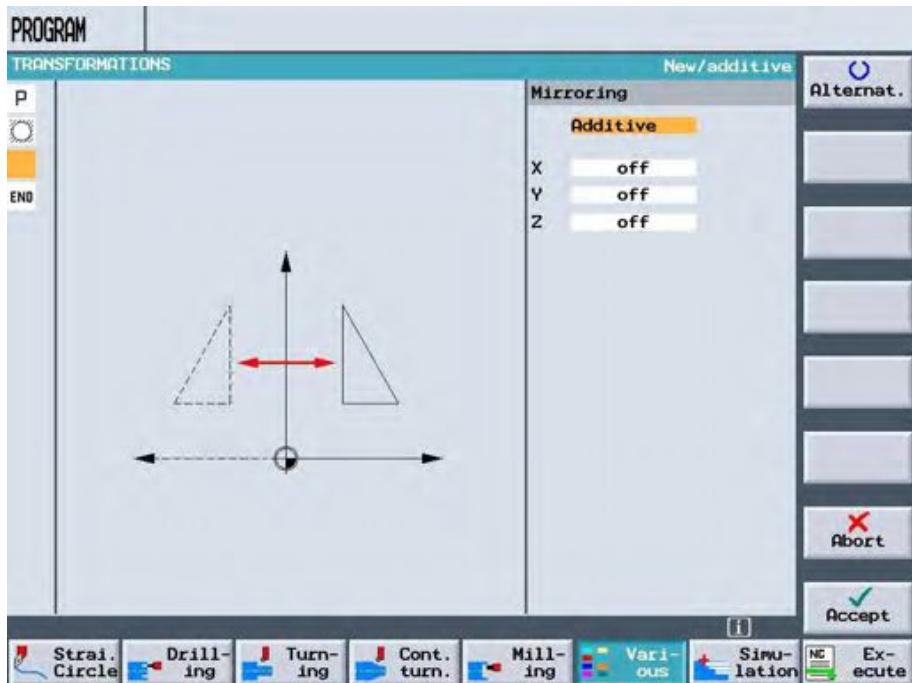
## Section 7

### Mirroring

**Mirror**

By pressing the VSK 5 "Mirror" the input mask for the programming for mirroring will be called up.

Notes



**Alternat.**

By pressing the VSK 1 "Alternat." the function Mirroring "New" or "Additive" will be called up.

Parameter	Help picture	Description and Hints
Mirroring		<p>New</p> <p>The machining direction will be mirrored about the selected axis. Any activated mirroring will be de-activated.</p> <p>Hints:</p> <ul style="list-style-type: none"> <li>the machining direction (conventional / climb milling) will also be mirrored.</li> <li>If "New" is selected, all previous transformations will be deleted.</li> </ul> <p>Additive</p> <p>The mirroring will be switched over as referred to any existing mirroring</p>
X, Y, Z		<p>Mirroring in X, Y or Z</p> <p>The mirroring for relevant axis can be switched ON or OFF.</p>

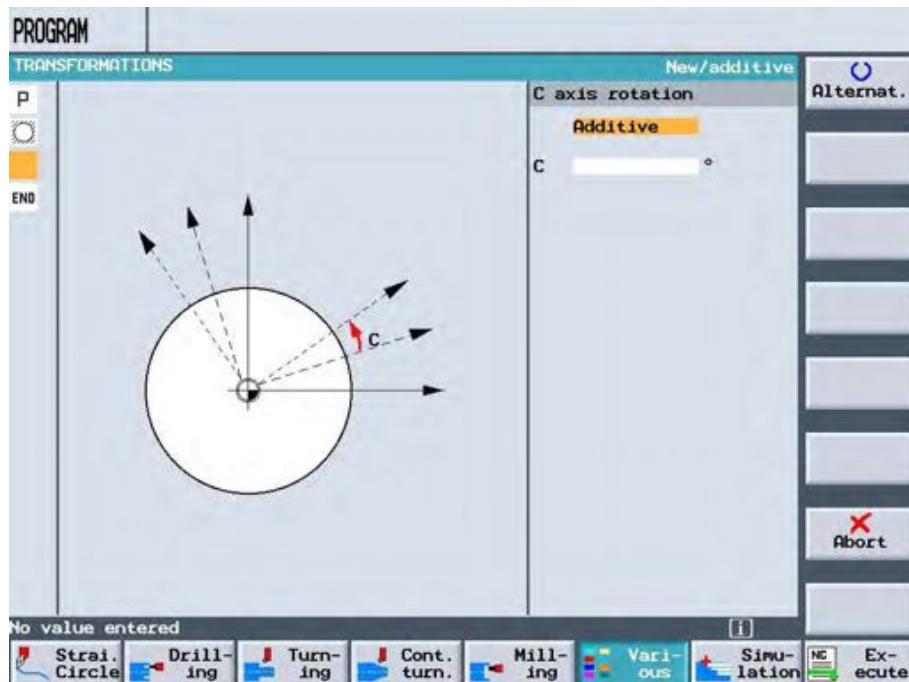
## Section 8

Notes

### Rotation C-axis

Rotation  
C axis

By pressing the VSK 6 "Rotation C-axis" the input mask for the programming of a rotation about the C-axis will be called up.



Alternat.

By pressing the VSK 1 "Alternat." the function Mirroring "New" or "Additive" will be called up.

Parameter	Help picture	Description and Hints
Rotation C-axis		New  The activated rotation will be de-activated, the zero point will be rotated back to the original WO.
C		Additive  The zero point will be rotated by the entered value in addition to the present rotation.

## 1 Brief description

**Objective of the module:**

This module explains the usage of the G-Code editor under ShopTurn.

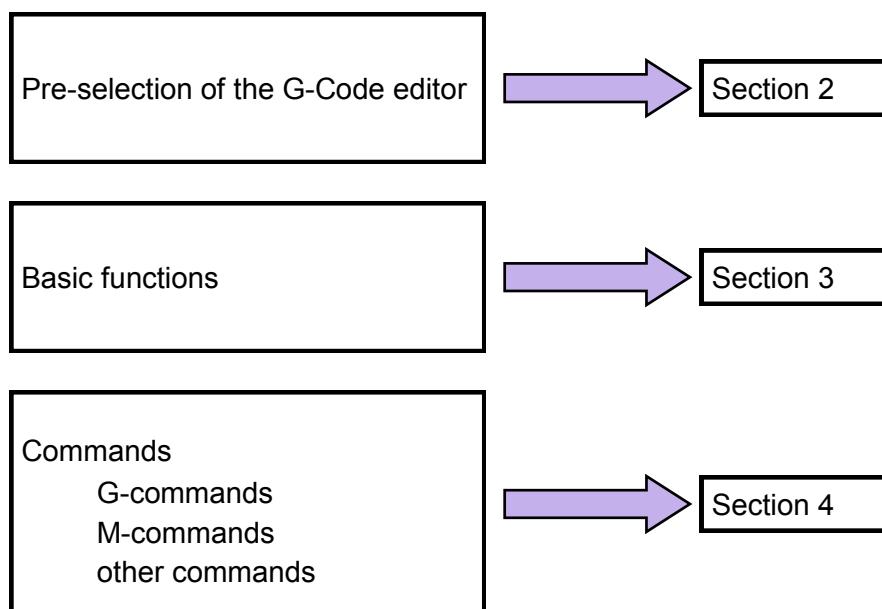
**Description of the module:**

Described in this module are the general functions and means of the G-Code editor, such as for example:

- Departure commands
- Machine commands
- Call-up of cycles

**Content of the module:**

- Pre-selection of the G-Code editor
- Basic functions
- Commands
  - G-commands
  - M-commands
  - other commands



## Section 2

Notes

Pre-selection of the G-Code editor

Creating a new G-Code program as described under *Module A107-“Operating Area Program - Program manager“ Section 3.2. “New NC-part program“*.

The G-Code editor opens.



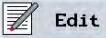
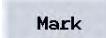
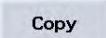
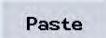
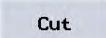
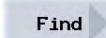
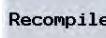
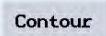
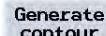
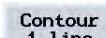
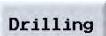
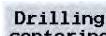
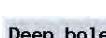
- ① Horizontal Softkey strip
- ② Vertical Softkey strip
- ③ Current line number

## Section 3

### Basic functions

The following basic functions are available in the G-Code editor.

Notes

HSK	VSK	Description
 Edit	 Mark	Marks program lines.
	 Copy	Copies marked program lines.
	 Paste	Inserts copied/cut-out program lines.
	 Cut	Cuts out marked program lines.
	 Find	Search or replacement of text in the program.
	 Continue	The following functions can be selected: Selection of a tool Positioning the cursor at the beginning or the end of the program Re-marking of program lines
	 Recompile	Permits the re-compilation of cycles back into the respective input mask
 Contour	 Generate contour	Opens the input mask for the range “Generate contour”.
	 Contour 1 line	Opens the input mask for the range “Contour 1 line”.
	 Contour 2 lines	Opens the input mask for the range “Contour 2 lines”.
	 Contour 3 lines	Opens the input mask for the range “Contour 3 lines”.
	 <<	Pages back to the basic mask of G-Code editor
 Drilling	 Drilling centering	Opens the input mask for the range “Drilling/ Centring”.
	 Deep hole drilling	Opens the input mask for the range “Deep hole drilling”.
	 Boring	Opens the input mask for the range “Boring”.
	 Tapping	Opens the input mask for the range “Thread cutting”.
	 Hole pattern pos.	Opens the input mask for the range “Position pattern”.
	 Deselect modal	Unselects the function “Modal”.

## Section 3

### Basic functions

Notes

HSK	VSK	Description
<b>Milling</b>	<b>Face milling</b>	Opens the input field for the section "Face milling".
	<b>Path milling</b>	Opens the input field for the section "Path milling".
	<b>Standard pockets</b>	Opens the input field for the section "Standard pockets".
	<b>Grooves</b>	Opens the input field for the section "Grooves".
	<b>Spigot</b>	Opens the input field for the section "Spigots".
	<b>&gt;&gt;</b>	Opens the second VSK-strip.
	<b>Thread milling</b>	Opens the input field for the section "Thread milling".
	<b>HighSpeed settings</b>	Opens the input field for the section "HighSpeed Settings". Hint: Optional - the documentation by the machine manufacturer must be observed.
	<b>&lt;&lt;</b>	Opens the first VSK-strip.

## Section 4

### Commands

Notes

The following standard G-commands are available in the G-Code editor .  
Hint: Un-named G-commands are freely selectable. The documentation by the machine manufacturer must be observed.

- G 00 Rapid traverse command
- G 01 Linear interpolation (Feed motion)
- G 02 Circular interpolation anti-clockwise
- G 03 Circular interpolation clockwise
- G 04 Dwell - Hint: Additional parameter inputs are required
- G 17 Plane selection XYZ
- G 18 Plane selection ZXY
- G 19 Plane selection YZX
- G 54 – Selection of the zero points
- G 57 Hint: More zero points may be available.  
The documentation by the machine manufacturer must be observed.
- G 90 Absolute dimensions
- G 91 Incremental dimensions

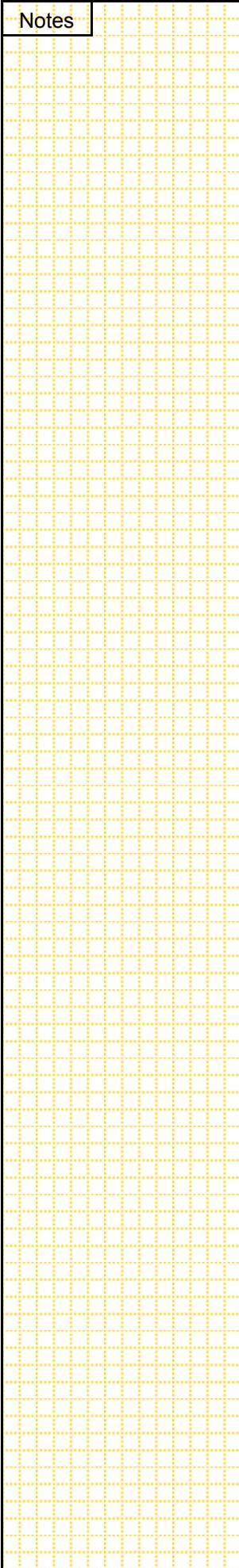
The following standard G-commands are available in the G-Code editor .  
Hint: Un-named M-commands are freely selectable. The documentation by the machine manufacturer must be observed.

- M00 Programmed Halt
- M01 Optional Stop, see also M00
- M02 End of Program
- M03 Spindle Start clockwise
- M04 Spindle Start anti-clockwise
- M05 Spindle Stop
- M06 Tool change
- M08 Coolant ON
- M09 Coolant OFF
- M19 Defined spindle stop
- M30 End of Program (see also M02)

The following standard “other“ commands are available in the G-Code editor. Hint: The documentation by the machine manufacturer must be observed.

- T Tool call-up (Tool)
- S Speed (Speed)
- F Feed rate (Feed)

Notes



## 1 Brief description

**Objective of the module:**

This module provides the means to carry out further exercises regarding the programming with Shop-Turn and to consolidate the programming knowledge so far attained.

**Description of the module:**

This module contains all the drawings that were used in the individual modules so far and additional work piece drawings as exercises for the consolidation of the acquired knowledge.

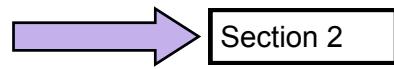
**Content of the module:**

## Shop drawings

- Bolt\_1
- Bolt\_2
- Shaft\_1
- Spigot\_1
- Pin
- Stepped shaft
- Contour spigot
- Joint shaft
- Fitting shaft
- Bowling pin
- Core
- Kidney
- Example counter spindle
- Hub

## Drawings for the programming examples

Shop drawings  
Bolt  
Shaft\_1  
Spigot\_1  
Pin  
Stepped shaft  
Contour spigot  
Joint shaft  
Fitting shaft  
Bowling pin  
Core  
Kidney  
Example counter spindle  
Hub



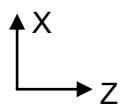
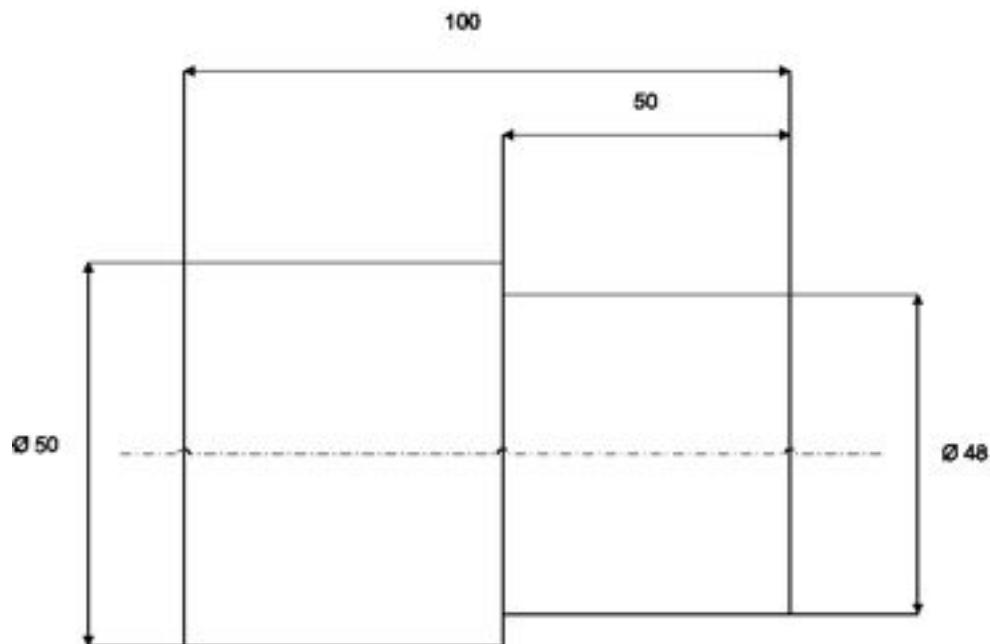
**Section 2**

## Section 2

### Shop drawings

Notes

2.1 Bolt\_1

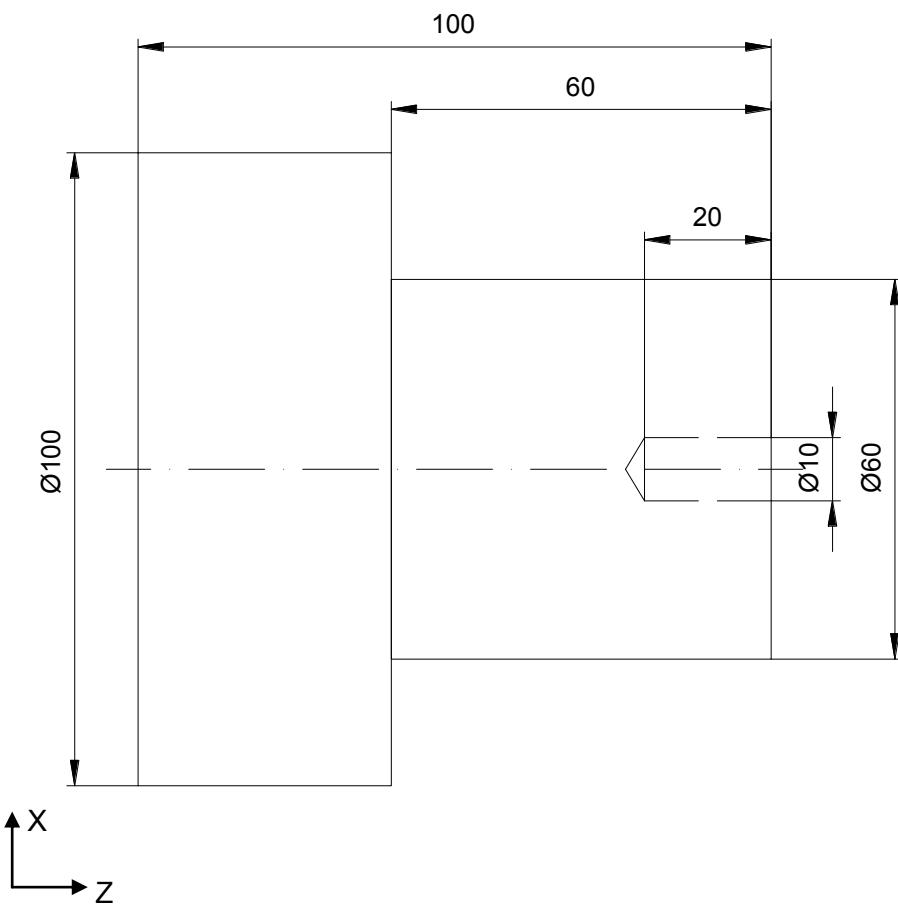


## Section 2

### Shop drawings

Notes

#### 2.1 Bolt\_2

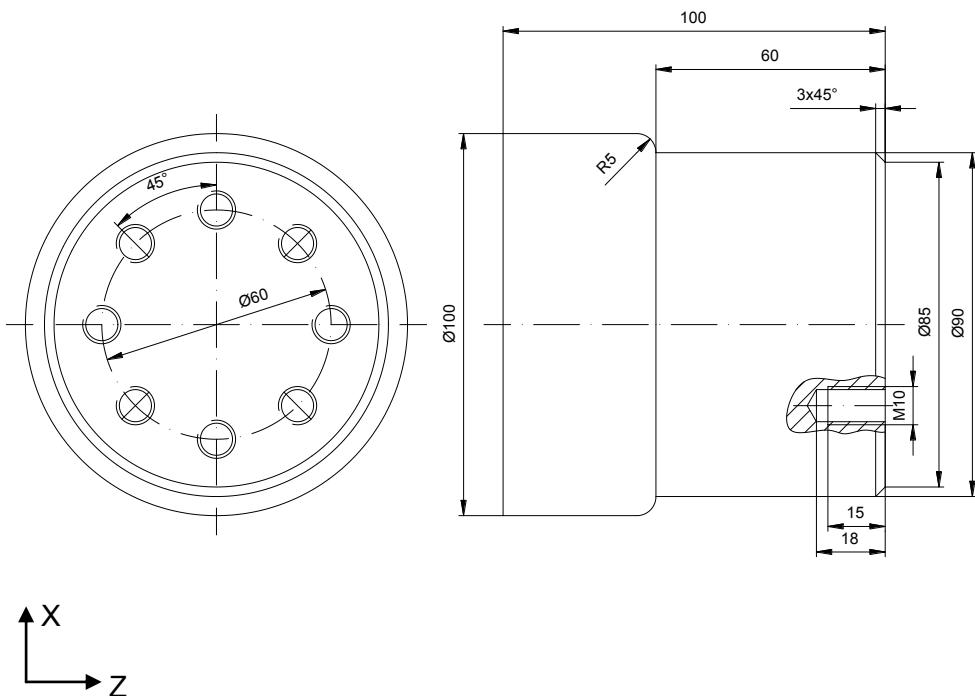


## Section 2

### Shop drawings

#### 2.2 Shaft\_1

Notes



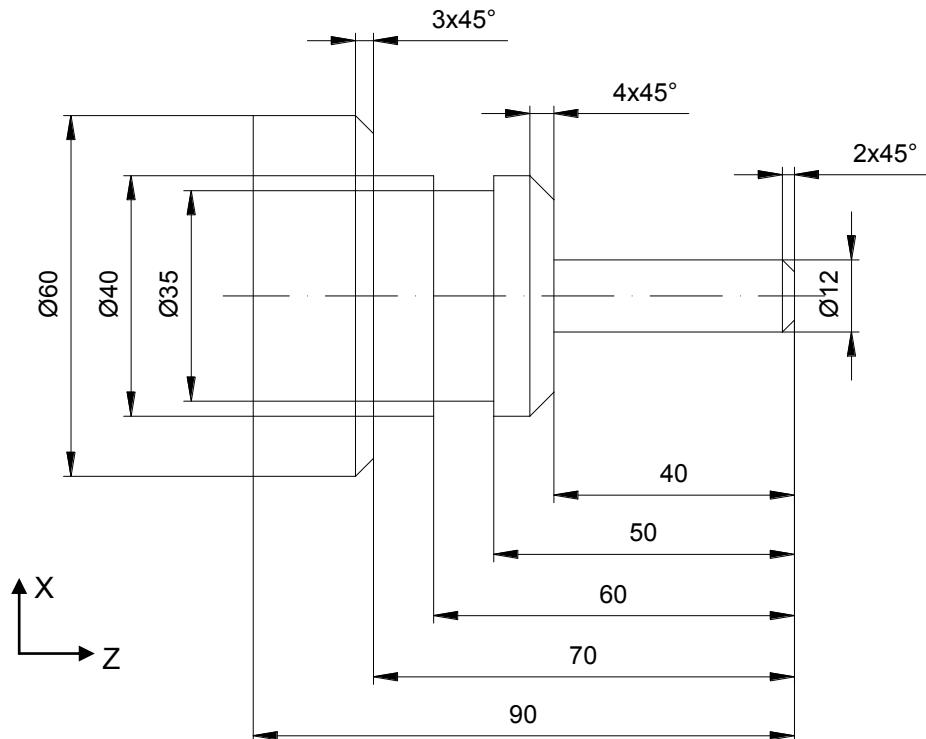
X  
Z

## Section 2

### Shop drawings

Notes

#### 2.4 Pin

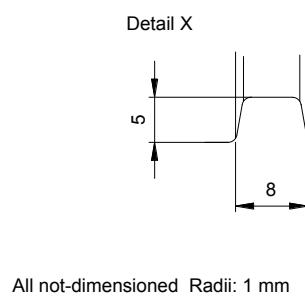
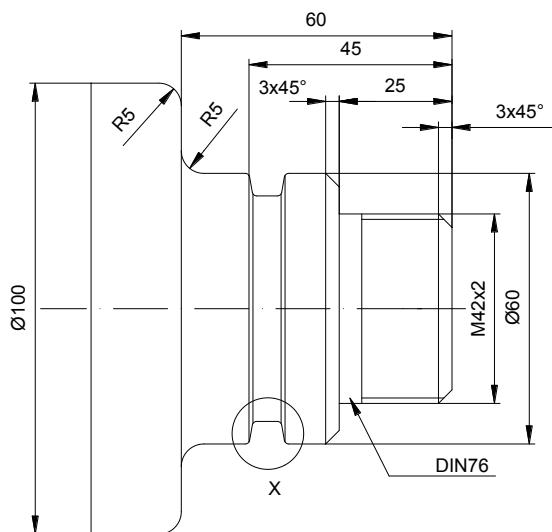


## Section 2

### Shop drawings

#### 2.3 Spigot\_1

Notes



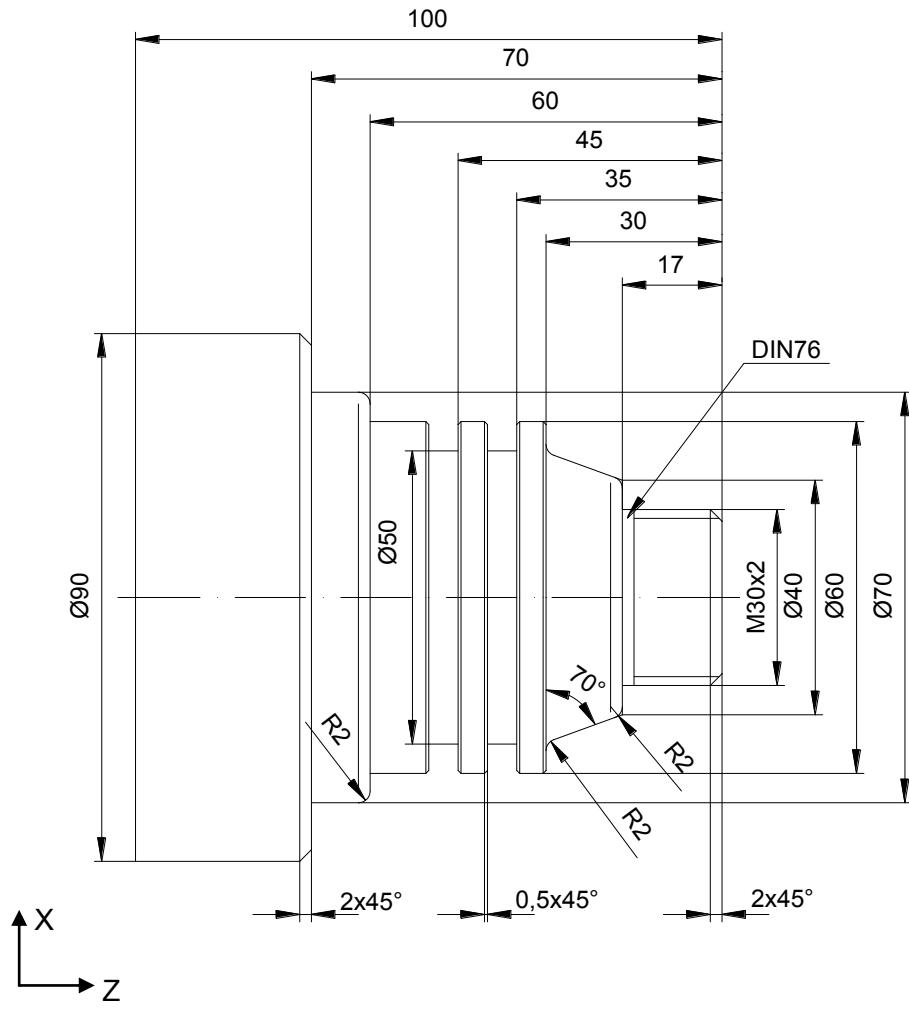
X  
Z

## Section 2

### Shop drawings

Notes

#### 2.5 Stepped shaft

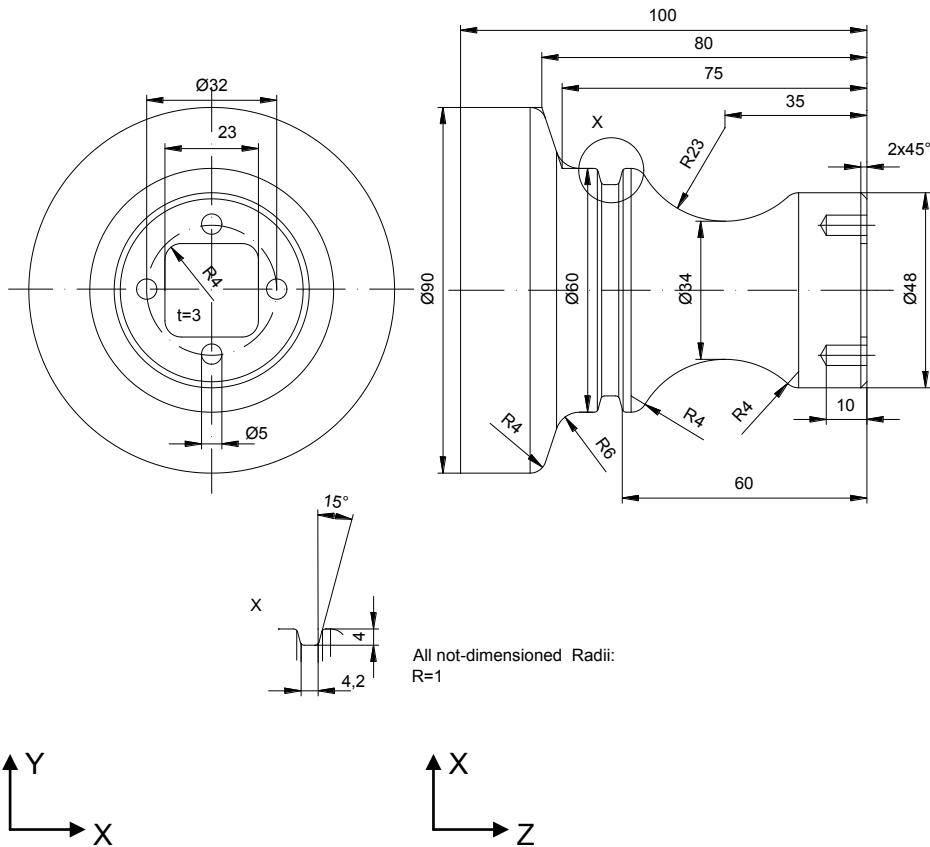


## Section 2

### Shop drawings

Notes

#### 2.6 Contour spigot

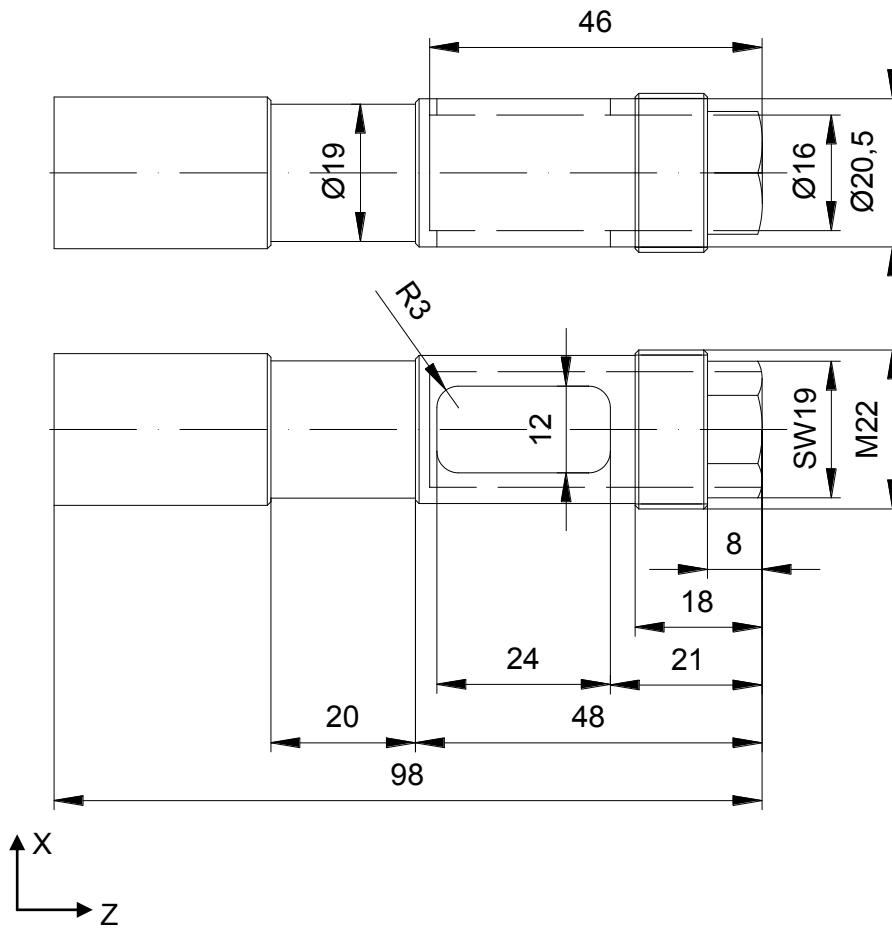


## Section 2

### Shop drawings

Notes

#### 2.7 Joint shaft

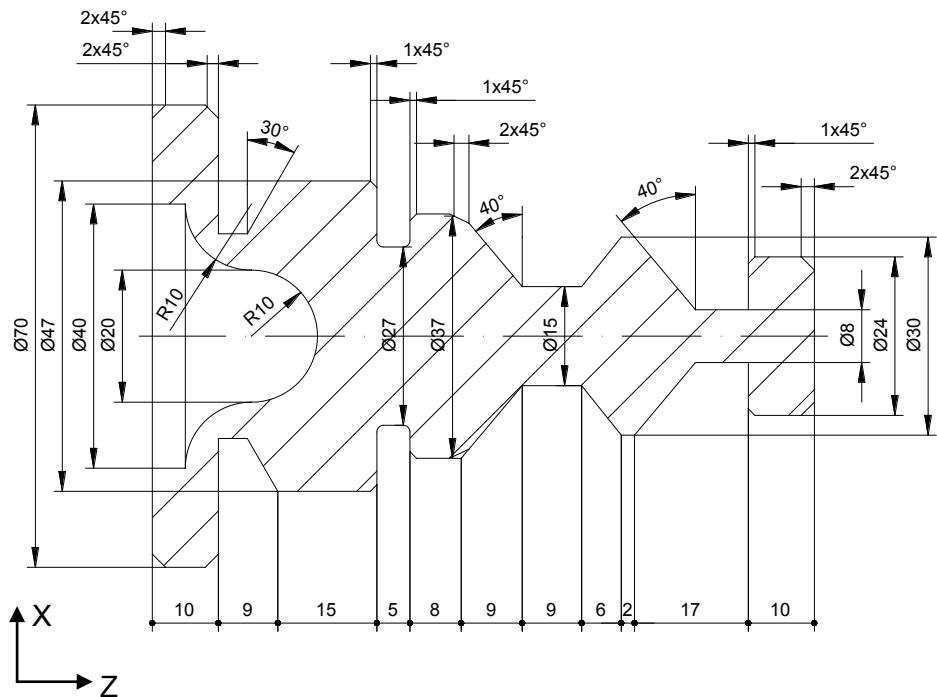


## Section 2

### Shop drawings

Notes

#### 2.8 Fitting shaft

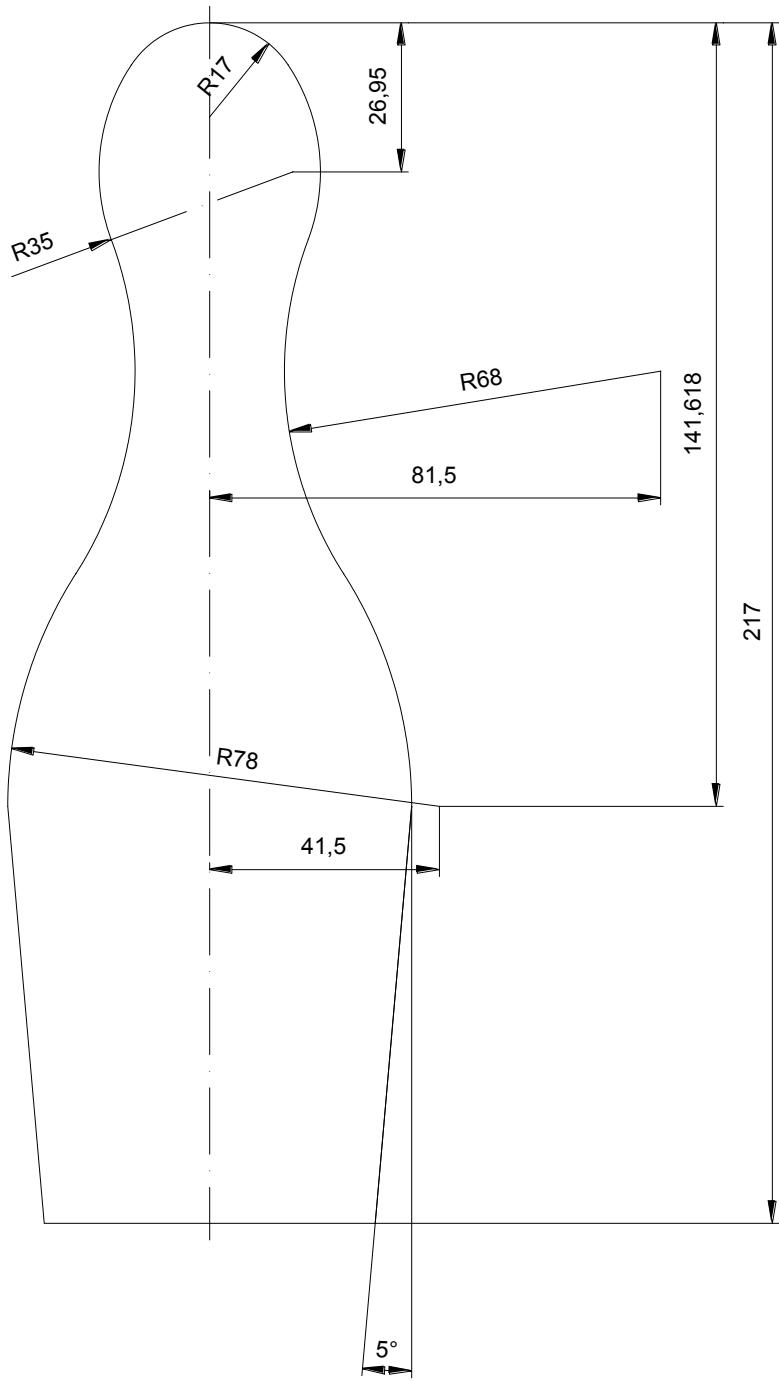


## Section 2

### Shop drawings

Notes

#### 2.9 Bowling pin

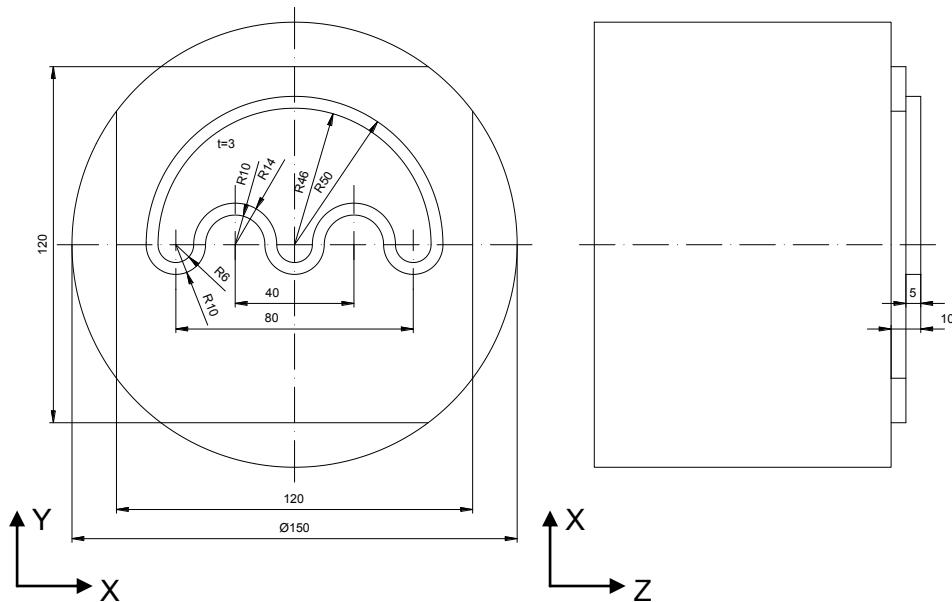


## Section 2

### Shop drawings

Notes

#### 2.10 Core

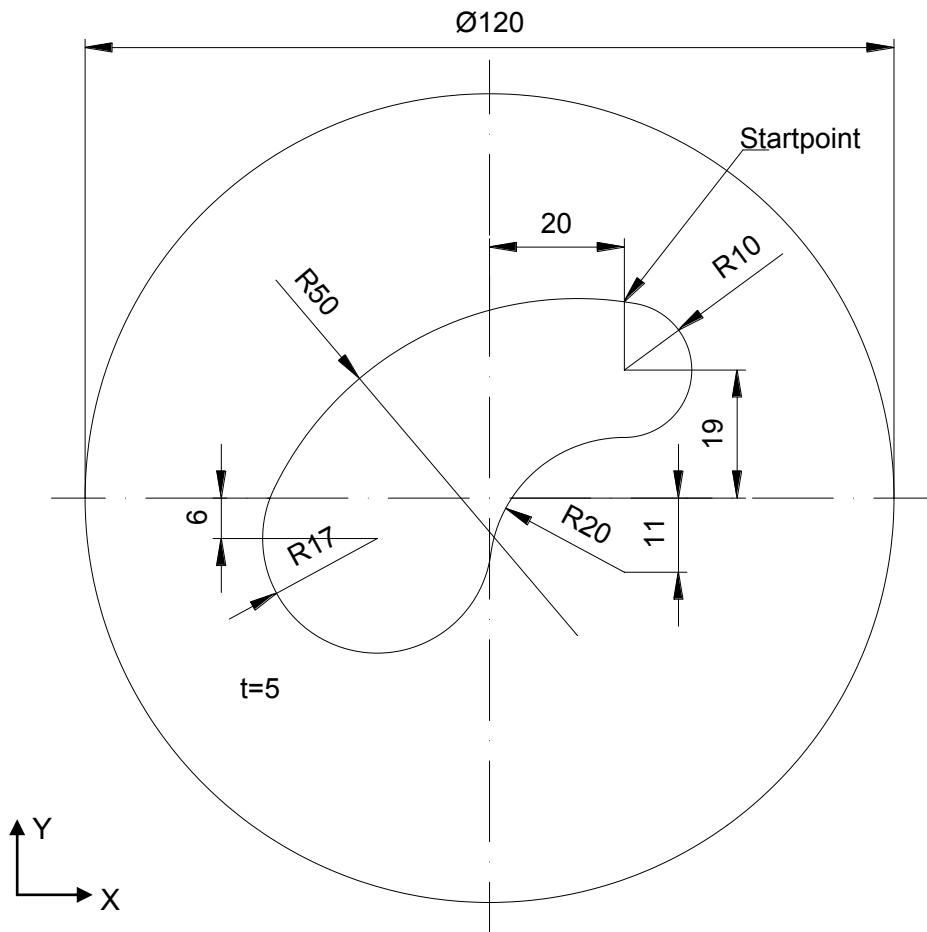


## Section 2

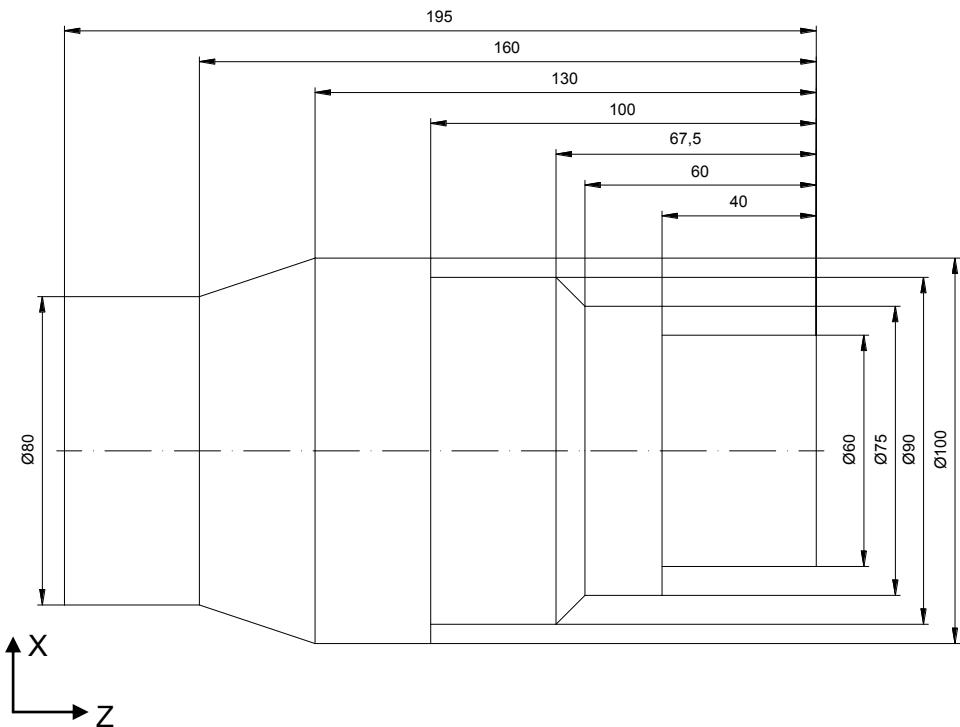
### Shop drawings

#### 2.11 Kidney

Notes



## 2.12 Example counter spindle



## Section 2

Notes

Shop drawings

2.13 Hub

