



HEIDENHAIN



VTC

Technical Manual (Supplement)

With VT 12x Vision System

English (en)
07/2024


Contents


1	Introduction.....	3
1.1	Meaning of the symbols used in this manual.....	3
1.2	Would you like any changes, or have you found any errors?.....	4
1.3	General information.....	5
2	Fundamentals.....	7
3	Function.....	9
4	Hardware.....	10
5	Initial setup.....	11
5.1	Installing the software.....	12
5.2	Configuring the Python process.....	14
5.3	Activating the cycles.....	15
5.4	Creating NC macros.....	16
5.5	VT121.TAB vision system table.....	25
5.6	VTC.TAB configuration table.....	25
5.7	VTC_CALIBRATION.TAB calibration data table.....	29
5.8	Setting up the VTC application.....	30
6	VTC on the programming station.....	31

1 Introduction


1.1 Meaning of the symbols used in this manual


 DANGER
<p>Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard will result in death or severe injury.</p>


 WARNING
<p>Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard could result in death or serious injury.</p>

 CAUTION
<p>Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard could result in minor or moderate injury.</p>

NOTICE
<p>Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard could result in property damage.</p>

 The information symbol indicates a tip . A tip provides important additional or supplementary information.

 The book symbol indicates a cross reference . A cross reference leads to external documentation, for example the documentation of your machine manufacturer or other supplier.

 The gear symbol indicates a function that depends on the machine . The function described depends on the machine if, for example: <ul style="list-style-type: none">■ Your machine features a certain software or hardware option■ The behavior of the functions depends on the configurable machine settings

1.2 Would you like any changes, or have you found any errors?

We are continuously striving to improve our technical documentation for you. Please help us by sending your requests.

Contact us via the following e-mail address:

tnc-oemdoc@heidenhain.de

1.3 General information

Supplementary documents

This manual is a supplement to the Technical Manual of your control. Therefore, please note and follow all the information and safety precautions contained in the current edition of the Technical Manual of your control. These also apply to the VT 12x and the related VTC (Visual Tool Check) software.



Technical Manual

Please refer to the current Technical Manual for your control.

User's Manual

Please refer to the current VTC User's Manual (ID 1322445-xx).

Operating Instructions

Please refer to the current VT 121 Operating Instructions (ID 1322444 -xx).

Please refer to the current VT 122 Operating Instructions (ID 1422124-xx).

Notes on unpacking and handling

CAUTION

Risk of injury from knives or sharp edges

Inappropriate tools or sharp edges of the housing of the devices can cause cuts during unpacking.

- ▶ Open the packaging with a suitable tool
- ▶ Wear suitable gloves during handling

NOTICE

General information

- Open the HEIDENHAIN packaging with a suitable tool. Make sure that you do not damage the device or injure yourself
- Read and observe all notes and mounting instructions that may be provided together with the devices in the packaging. They are intended as supplements to the Technical Manuals
- Ensure that you have enough room for mounting and installation in your working area.

To unpack the product:

- ▶ Open the packaging
- ▶ Remove the packaging materials. Take care not to lose any of the accessories contained in the packaging
- ▶ Remove contents, use suitable means of transport, if necessary
- ▶ Check the delivery for completeness
- ▶ Check the delivery for damage

In case of damage in transit:

- ▶ Have the shipping agent confirm the damage
- ▶ Keep the packaging material for inspection
- ▶ Notify the sender of the damage

Information for commissioning

Pay attention to the notes and safety precautions in the Technical Manual of your control.

Notes on the disposal of HEIDENHAIN components

NOTICE

Incorrect disposal

Pollution

The devices described in this manual must be disposed of or recycled in accordance with the applicable national regulations.
Consult a disposal company for electronic devices as needed.

2 Fundamentals

The VT 12x vision system and the VTC (Visual Tool Check) software (software: ID 1280600) enable you to perform a visual inspection of the currently inserted tool during program run.

The following software options need to be available on the HEIDENHAIN control in order to use VTC:

- **Python** software option (option 46)
- **Remote Desktop Manager** software option (option 133)

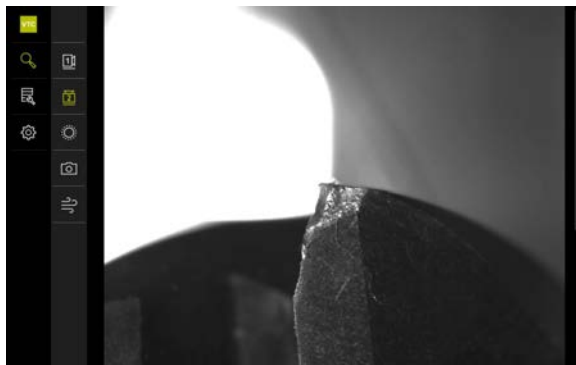
The VT 12x vision system is equipped with two cameras and thus allows you to inspect your tool from different viewing angles. Thanks to the high sturdiness of the vision system, the VT 12x can be mounted inside the working space. Cleaning jets remove chips and liquid contamination from the cameras. A LED ring light illuminates the scene while images are being taken.



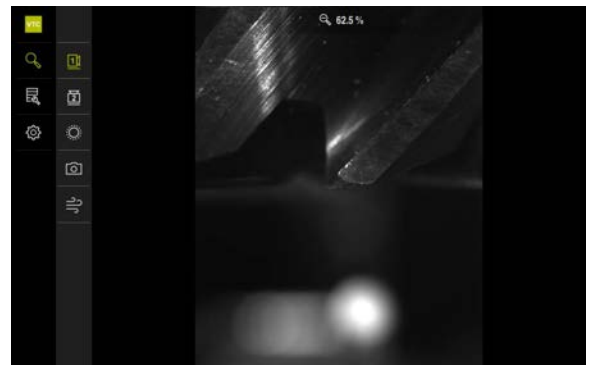
The VT 12x allows you to perform the following tool inspections:

- Checking the quality of the cutting edges
- Checking the tool for breakage
- Tool measurement (VT 122 only)

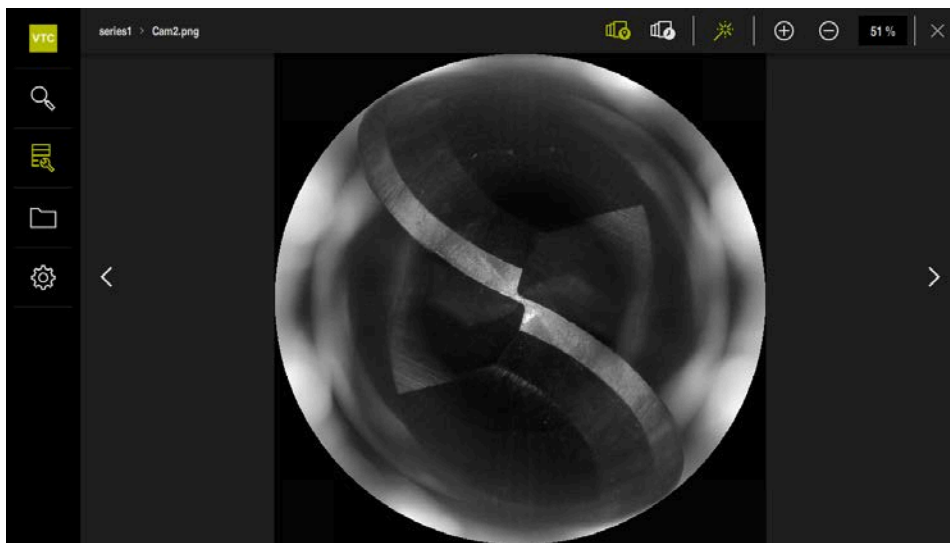
The VTC (Visual Tool Check) software allows you to manage your images in an image database or to inspect your tool by means of a live image. VTC can create single shots of the tool cutting edges as well as panoramic images of the tools. Panoramic images show the entire tool circumference in unfolded view. The integrated image processing detects tool breakage and initiates the defined reaction.



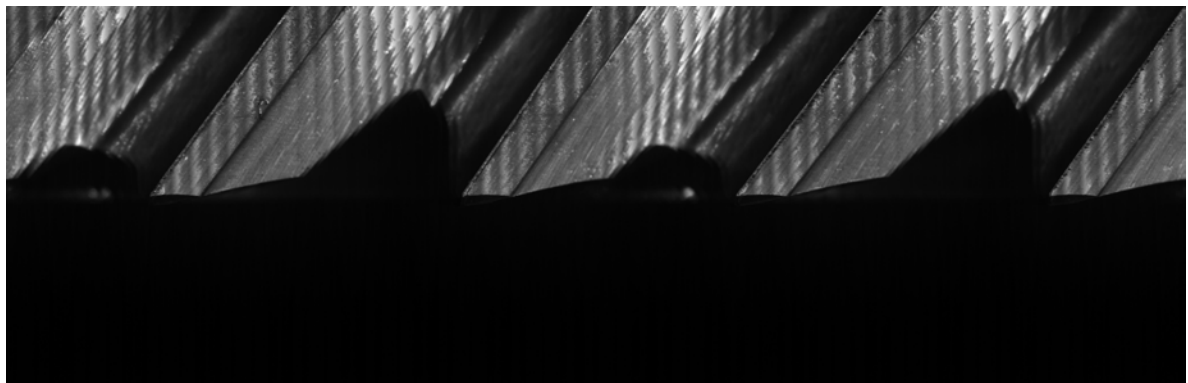
Camera 2: image of the tool from below



Camera 1: image of the tool from the side



Camera 2: mosaic image



Camera 1: panoramic image

3 Function

Cycles let you specify multiple locations within the NC program at which the control takes images of the inserted tool or inspects the tool for breakage. With the help of Q parameters, users can adapt the cycle to their requirements as usual.

The cycles are selected via the **TCH PROBE** menu, which is expanded to include the **VTC VISION SYSTEM** soft key after the VTC software has been installed.



For a list of the cycles and more detailed explanation on the sequence and operation of the cycles, please refer to the VTC User's Manual.

4 Hardware

With the VT 12x vision system, HEIDENHAIN provides the right hardware for the VTC software.

Part number (ID) of the vision system:

- VT 121: ID 1249466-xx
- VT 122: ID 1373589-xx

The VT 12x vision system for tool monitoring (ID 1249466-xx) consists of:

- Protective housing
- Upper camera (camera 2)
- Lateral camera (camera 1)
- Ring lights
- Measuring LED (VT 122 only)
- Jets
- Compressed-air inlet
- Connection for voltage supply
- Camera interface

The vision system is suitable for mounting in the working space. In order to achieve high-quality images and measurements, the vision system is configured and calibrated by the control. For this purpose, the vision system is probed using touch probe cycles and a calibration tool is measured. When mounting the vision system, therefore please ensure sufficient distance between the vision system and the limit switches in its proximity.



Further specifications for the VT 122 vision system can be found in the VT 122 Installation Instructions (ID 1422124).

Further specifications for the VT 121 vision system can be found in the VT 121 Operating Instructions (ID 1322444).

NOTICE

Vision system as a collision object

Collision between tool and vision system

- ▶ Choose the mounting orientation such that the tool will move towards the jet blocks of the vision system when approaching the imaging position
- ▶ When tilting the tool into position, ensure sufficient distance between the tool and the vision system

5 Initial setup

For successful initial setup of the VT 12x vision system and the VTC software, perform the following steps in the sequence given below:

- Installation
- Installing the software
- Setting up the Python process
- Activating the cycles
- Creating NC macros
- Adjusting the VTC.TAB configuration table
- Adjusting the VTC_CALIBRATION.TAB table

For more detailed information on how to perform each individual step, read the corresponding section of this manual.



For more information about setting up the vision system and the VTC software, please refer to the VTC User's Manual.

5.1 Installing the software

The setup file is available as a ZIP file on the HEIDENHAIN filebase. The name of the setup file is: VTC_<VTC version>_TNC_<NC software>-<software version>.zip.

Example: VTC_1.3.0_TNC_340590-10.zip

The software can also be installed while SELinux is active.



Please note the following regarding the installation of the software:

- The VTC setup software version 1.4 or higher is required.
- After an NC software update, you also need to reinstall the VTC with a VTC software package that matches the NC software
- The installation of the VTC software package does not change the existing PLC project
- After updating the VTC software, the control will issue an error message when rebooting after the RMP installer has finished. You can acknowledge this error message with OK.



HEIDENHAIN recommends:

Make a backup of your control data, for example with TNCbackup (included in TNCremo), before updating the NC software.



The tables and directories for VTC will only be created on the control after the corresponding Python process has been executed for the first time. Please refer to Configuring the Python process.

Installing the software on a TNC7

- ▶ Connect the appropriate network drive or the USB memory medium
- ▶ Enter the code number **231019**, or the keyword **SETUP** or **setup**.
- > The **Load NC software** window opens

Continue with "Running the update"

Installing the software on a TNC 640

To update the software manually:

- ▶ Connect the appropriate network drive or the USB memory medium containing the files necessary for the update (**setup.zip**).
- If the network drive is not displayed, please proceed as follows (if it is, then you can skip these sub-steps):
 - Switch to the **Programming** operating mode
 - Press the **PGM MGT** key to open the file manager
 - Press the **NETWORK** soft key (second soft-key row)
 - Select the PC to be connected and confirm it by pressing the **CONNECT DRIVE** soft key
 - To exit the list of network drives, press the **END** soft key
 - To exit the file manager, press the **END** soft key
- ▶ While in the **Programming** operating mode, press the **MOD** key
- ▶ Enter the code number **231019**, or the keyword **SETUP** or **setup**
- > The **Load NC software** window opens

Running the update

- ▶ Enter the path of the setup file in the following window
- ▶ In the selection dialog, select the **setup.zip** file and press the **SELECT** soft key
- ▶ Press the **YES** soft key to confirm the message **Do you really want to switch off?**
- ▶ The NC software is now automatically shut down and then the actual update program is started
- ▶ Select the appropriate conversational language in the update program and confirm with the **ENT** key
- ▶ The control displays an overview of the previous NC software version and the NC software version to be installed. If too little memory capacity is available on the memory medium, the control offers to delete an older NC software version. Press the **Yes** button if you want to continue the update process
- ▶ Wait until the update program has installed the new NC software. You will be informed about the installation progress in a status window with progress bar
- ▶ If you used a USB device to run the update:
Remove the USB memory medium after the update has been completed and confirm the restart of the control
- ▶ During the first restart of the control after the software update, important firmware updates of connected devices may be necessary. In this case, follow the instructions on the screen.
- ▶ Read-in any files that you had saved to a PC by using TNCremo
- ▶ The **COPY SAMPLE FILES** soft key is used to copy the HEIDENHAIN standard tables and kinematics configuration examples to the corresponding directories
- > End of the NC software update



The tables and directories for VTC will only be created on the control after the corresponding Python process has been executed for the first time. Please refer to Configuring the Python process.

5.2 Configuring the Python process

Python processes are managed in the configuration editor under System\PLC\PythonSettings. The Python process is automatically created during installation of the VTC software.

To configure the VTC Python process:

- ▶ Configure the "VTC" data object under Process\CfgPythonScript (118300) as follows:
 - **parameter** (118301):
 - Enter the parameters successively, separated by space characters
 - --application=VTC
 - (Optional) --opcuaPortNr <port number>
If port number 19050 is already being used, you can assign an alternative number here. Default = 19050
 - --opcuaHost=<server IP> | <server name>
To be able to use the VTC cycles of the control, insert this parameter. Adapt the server IP or the host name as needed.

In order for the Python process to be executed during start-up of the control, also perform the following steps:

- ▶ Insert the data object System\PLC\PythonSettings\Process\CfgPyJobToActivate\atPLCStart (127401)
- ▶ Create a new element
- ▶ The control opens a window with a selection menu listing the defined Python processes.
- ▶ Select the Python process of the VTC vision system (e.g., VTC)
- ▶ Confirm your selection with OK
- ▶ Restart the control
- ▶ Load the PLC program
- ▶ The control has created the VTC.TAB and VTC_CONFIGURATION.TAB tables under PLC:\VTC, as well as the VTC-TOOLS.TAB table under TNC:\table.

5.3 Activating the cycles

During the installation of VTC, the control saves the "VTC.cdf" file. In this file, the NC cycles for VTC are defined. To be able to use the NC cycles, you need to tell the control which cycles are contained in the file.

Proceed as follows:

- ▶ Open the following path in the configuration editor: System\Path\CfgOemPath (102000)
- ▶ Create a new element under **cycleSubTreeFiles** (102004)
- ▶ Press the **ENTER VALUE DIRECTLY** soft key
- ▶ Enter the following path: SYS:\jhcyc\VTC.cdf



To be able to use the newly added cycles, the control must be restarted.

5.4 Creating NC macros

For the VTC cycles, pre- and post-macros are required. Please note the following steps:

- ▶ Create pre-macro VTC_PRE.H (see "Example of VTC_PRE.H")
- ▶ Create the post-macro VTC_POST.H (see "Example of VTC_POST.H")
- ▶ Save the VTC_PRE.H and VTC_POST.H macros under PLC:\VTC
- ▶ Enter the name of the pre-macro (VTC_PRE.H) in the VTC.TAB table in the OEM_MACRO_VTC_PRE row
- ▶ Enter the name of the post-macro (VTC_POST.H) in the VTC.TAB table in the OEM_MACRO_VTC_POST row

Example of VTC_PRE.H

The NC macro indicated here only serves as an example and needs to be adapted to the individual machine.

```

0 BEGIN PGM VTC_PRE MM
;-----
;Output:
; Q601 = 0 Execute cycle
;       = 1 Don't execute cycle
;-----
;DON'T WRITE TO: QL0-QL2 !!!
;-----
;Read cycle number
FN 18: SYSREAD QL1 = ID71 NR2 IDX0
;Turning mode
FN 18: SYSREAD QL2 = ID992 NR20
FN 9: IF +QL1 EQU +620 GOTO LBL "VTC_620"
FN 9: IF +QL1 EQU +621 GOTO LBL "VTC_621"
FN 9: IF +QL1 EQU +622 GOTO LBL "VTC_622"
FN 9: IF +QL1 EQU +623 GOTO LBL "VTC_623"
FN 9: IF +QL1 EQU +624 GOTO LBL "VTC_624"
FN 9: IF QL1 EQU +625 GOTO LBL "VTC_625"
FN 9: IF QL1 EQU +626 GOTO LBL "VTC_626"
FN 9: IF QL1 EQU +627 GOTO LBL "VTC_627"
FN 9: IF QL1 EQU +628 GOTO LBL "VTC_628"
FN 9: IF QL1 EQU +629 GOTO LBL "VTC_629"
FN 9: IF QL1 EQU +630 GOTO LBL "VTC_630"
FN 9: IF QL1 EQU +631 GOTO LBL "VTC_631"
;Unknown cycle number
QL0 = 1
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 620 VT SETUP
;=====
LBL "VTC_620"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_620"
;Milling
;-----
;
QL0 = 0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_620"
;Turning
;-----
;
QL0 = 1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"

```

```

;=====
;Cycle 621 MANUAL INSPECTION
;=====
LBL "VTC_621"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_621"
;Milling
;-----
;
QL0 = 0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_621"
;Turning
;-----
;
QL0 = 1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 622 IMAGES
;=====
LBL "VTC_622"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_622"
;Milling
;-----
;
QL0 = 0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_622"
;Turning
;-----
;
QL0 = 1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 623 BREAKAGE DETECTION
;=====
LBL "VTC_623"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_623"
;Milling
;-----
;
QL0 = 0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_623"
;Turning
;-----
;
QL0 = 1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 624 MEASURING CUTTING EDGE ANGLES
;=====
LBL "VTC_624"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_624"
;Milling

```

```

;-----
;
QL0 = 0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_624"
;Turning
;-----
;
QL0 = 1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 625 CALIBRATION
;=====
LBL "VTC_625"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_625"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_625"
;Turning
;-----
;
QL0 = +1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 626 TEMPERATURE COMPENSATION
;=====
LBL "VTC_626"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_626"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_626"
;Turning
;-----
;
QL0 = +1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 627 TOOL LENGTH
;=====
LBL "VTC_627"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_627"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_627"
;Turning

```

```

;-----
;
QL0 = +1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 628 TOOL RADIUS
;=====
LBL "VTC_628"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_628"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_628"
;Turning
;-----
;
QL0 = +1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 629 TOOL RADIUS 2
;=====
LBL "VTC_629"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_629"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_629"
;Turning
;-----
;
QL0 = +1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 630 MEASURE TOOL
;=====
LBL "VTC_630"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_630"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_630"
;Turning
;-----
;
QL0 = +1
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 631 MEASURE TOOL TIP

```

```

;=====
LBL "VTC_631"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_631"
;Milling
;-----
;
QL0 = +0
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_631"
;Turning
;-----
;
QL0 = +1
;-----
          LBL "COMMON"
;Common program part
;-----
;
;-----
Q601= QL0
END PGM VTC_PRE MM

```



If the VTC_PRE.H macro is not executed due to Q601 = 1, the VTC_POST macro will not be executed, either.

Example of VTC_POST.H

The NC macro indicated here only serves as an example and needs to be adapted to the individual machine.

```

0 BEGIN PGM VTC_POST MM
;-----
;DON'T WRITE TO QL1-QL2 !!!
;-----
;Read cycle number
FN 18: SYSREAD QL1 = ID71 NR2 IDX0
;Turning mode
FN 18: SYSREAD QL2 = ID992 NR20
FN 9: IF +QL1 EQU +620 GOTO LBL "VTC_620"
FN 9: IF +QL1 EQU +621 GOTO LBL "VTC_621"
FN 9: IF +QL1 EQU +622 GOTO LBL "VTC_622"
FN 9: IF +QL1 EQU +623 GOTO LBL "VTC_623"
FN 9: IF +QL1 EQU +624 GOTO LBL "VTC_624"
FN 9: IF QL1 EQU +625 GOTO LBL "VTC_625"
FN 9: IF QL1 EQU +626 GOTO LBL "VTC_626"
FN 9: IF QL1 EQU +627 GOTO LBL "VTC_627"
FN 9: IF QL1 EQU +628 GOTO LBL "VTC_628"
FN 9: IF QL1 EQU +629 GOTO LBL "VTC_629"
FN 9: IF QL1 EQU +630 GOTO LBL "VTC_630"
FN 9: IF QL1 EQU +631 GOTO LBL "VTC_631"
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 620 VT SETUP
;=====
LBL "VTC_620"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_620"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_620"

```

```

;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 621 MANUAL INSPECTION
;=====
LBL "VTC_621"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_621"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_621"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 622 IMAGES
;=====
LBL "VTC_622"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_622"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_622"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 623 BREAKAGE DETECTION
;=====
LBL "VTC_623"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_623"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_623"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 624 MEASURING CUTTING EDGE ANGLES
;=====
LBL "VTC_624"
;Turning mode ?
FN 9: IF +QL2 EQU +1 GOTO LBL "TURNING_624"
;Milling
;-----

```

```

;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_624"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 625 CALIBRATION
;=====
LBL "VTC_625"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_625"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_625"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 626 TEMPERATURE COMPENSATION
;=====
LBL "VTC_626"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_626"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_626"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 627 TOOL LENGTH
;=====
LBL "VTC_627"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_627"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_627"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 628 TOOL RADIUS
;=====
LBL "VTC_628"

```

```

;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_628"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_628"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 629 TOOL RADIUS 2
;=====
LBL "VTC_629"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_629"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_629"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 630 MEASURE TOOL
;=====
LBL "VTC_630"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_630"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_630"
;Turning
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
;=====
;Cycle 631 MEASURE TOOL TIP
;=====
LBL "VTC_631"
;Turning mode ?
FN 9: IF QL2 EQU +1 GOTO LBL "TURNING_631"
;Milling
;-----
;
;-----
FN 9: IF +1 EQU +1 GOTO LBL "COMMON"
LBL "TURNING_631"
;Turning
;-----
;
;-----
LBL "COMMON"

```

```
;Common program part  
;-----  
;  
;-----  
END PGM VTC_POST MM
```



Information on the creation of NC programs can be found in the Klartext Programming User's Manual.

5.5 VT121.TAB vision system table

The VT121.TAB vision system table contains the vision system data. The control uses these data for the VTC cycles. The table is stored in the PLC:\VTC directory.



Always consult with HEIDENHAIN before making any changes to the VT121.TAB vision system table.

5.6 VTC.TAB configuration table

The VTC.TAB configuration table contains the data for the machine-specific sequence of the cycles. The table can be found in the PLC:\VTC directory. Enter here the machine-specific data of your machine.


The configuration table consists of the following columns:

Column name	Description
NAME	Name of of the configuration datum
VALUE	Numerical configuration datum
STRVALUE	Textual configuration datum
DOC	Documentation on configuration datum

The configuration table consists of the following rows:

Row name	Description
VERSION	Version index of the table This value may change in case of changes to the table format and is checked by the cycles.
FEED_PLANE	Cycle feed rate in the working plane (numerical value or FMAX string in case of rapid-traverse movement)
FEED_TOOL_DIRECTION	Cycle feed rate in tool-axis direction (numerical value or FMAX string in case of rapid-traverse movement)
BREAKAGE_TOLERANCE	Global breakage tolerance Effective if tool-specific breakage tolerance LBREAK=0. The value 0.0 returns the result "breakage."
BREAKAGE_LOCK_TOOL	In the event of tool breakage, lock the tool in TOOL.T If value = 1.0, then the tool will be locked in the event of tool breakage If value = 0.0 or no entry, then the tool will continue to be used
BREAKAGE_NC_STOP	NC stop in the event of tool breakage If value = 1.0, then an NC stop will be triggered in the event of tool breakage If value = 0.0 or no entry, then the NC program will be continued
WORN_TOLERANCE	Global wear tolerance No entry: Use LTOL or RTOL from TOOL.T

Row name	Description
WORN_LOCK_TOOL	Lock the tool in TOOL.T upon exceedance of wear limit If value = 1.0, then an NC stop will be triggered If value = 0.0 or no entry, then the NC program will be continued
WORN_NC_STOP	NC stop upon exceedance of wear limit If value = 1.0, then an NC stop will be triggered If value = 0.0 or no entry, then the NC program will be continued
PICTURE_L_OFFSET	Global tool offset in tool-axis direction on the lateral camera with "live image" and "images" cycles No entry corresponds to the value 0.0. The value is limited to 10 mm in negative tool-axis direction.
PICTURE_R_OFFSET	Global radial tool offset on the upper camera with "live image" and "images" cycles No entry corresponds to the value 0.0
USE_TOOLT_LR_OFFS	Length offset and radial offset from TOOL.T (for imaging only) <ul style="list-style-type: none"> ■ Value = 0.0 or no entry (recommended) VTC uses the value PICTURE_L_OFFSET for camera 1 (lateral) and the value PICTURE_R_OFFSET for camera 2 (upper). The values from the TOOL.T table will not be used. ■ Value = 1.0 VTC uses the linear offset (L-OFFS) for camera 1 (lateral) and the radial offset (R-OFFS) for camera 2 (upper) from the TOOL.T table. The diameter of the tool influences its placement in front of the camera. Tool diameter $D < 7.5$ mm: <ul style="list-style-type: none"> ■ R_OFFS = 0: Center placement in front of the camera ■ R_OFFS = Radius or no entry: Placement in front of the camera is eccentric by the radius Tool diameter $D > 7.5$ mm: <ul style="list-style-type: none"> ■ R_OFFS = 0: Placement in front of the camera is eccentric by the radius ■ R_OFFS = Radius or no entry: Center placement in front of the camera <p>The values PICTURE_L_OFFSET and PICTURE_R_OFFSET have an additive effect, respectively.</p>
REFERENCE_SPINDLE_ANGLE	Reference angle of the spindle nose relative to the camera orientation X+ for externally determined cutting edge angles
SPINDLE_SPEED_BREAKAGEDTECT	Nominal spindle speed for Cycle 623 BREAKAGE DETECTION Ideal value: 10 000 rpm
SPINDLE_SPEED_SPIN_OFF	Spindle speed for self-cleaning of the tool through high spindle speed (Cycle 622). Active until the tool is positioned in front of the camera. No entry: no self-cleaning

Row name	Description
OEM_MACRO_VTC_POST	Name of the post-macro for the VTC cycles The macro must be in the PLC:\VTC directory
OEM_MACRO_VTC_PRE	Name of the pre-macro for the VTC cycles The macro must be in the PLC:\VTC directory
PLC_MARKER_BLOW	Name of the global PLC marker that is set by the cycles for cleaning the cameras with compressed air ¹⁾
PLC_MARKER_CYCLE_ACTIVE	Name of the global PLC marker that indicates an active VTC cycle ¹⁾
PLC_WORD_TOOL_BROKEN	This PLC word represents the event parameter Q601 of Cycle 623 ¹⁾
COLUMN_NAME_TOOLID	Tool ID from TOOL.T Enter here the column of the TOOL.T table in which the tool IDs are saved. An empty field means: the IDs are assigned by VTC
SERIES_TIME_FORMAT	Specify the folder name format in this row: %Y: year as four digits %m: month as a number from 1 to 12 %d: day as a number from 1 to 31 %H: hour as a number from 0 to 23 %M: minute as a number from 0 to 59 %S: second as a number from 0 to 59 Example 1: (default) Format string %Y%m%d%H%M%S: October 21, 2021 at 15:37 hours and 44 seconds = 20211021153744 Example 2: Format string %Y-%m-%d_%H-%M-%S: October 21, 2021 at 15:37 hours and 44 seconds = 2021-10-21_15-37-44
<div style="border: 1px solid black; padding: 5px;"> <p> Only default formatting sorts the folders by date in ascending order.</p> <ul style="list-style-type: none"> ■ You need to specify all of the six parts of the format string. ■ In the format string, do not use any characters that are not permitted in folder names (e.g., colon). </div>	
MAX_TILT_ANGLE	Reserved
MAX_RADIUS_MEAS_LENGTH	Maximum measuring range for radius measurement Clearance below the point of focus of camera 1
BLOCK_NEGATIVE_DR2	Prohibit negative DR2 entries to prevent the possibility of contour damage No entry or 0: No 1: Yes
MIN_TOOL_LENGTH	Minimum tool length in mm when measuring tools with a length of 0

Row name	Description
MAX_TOOL_LENGTH	Maximum tool length in mm when measuring tools with a length of 0

1) The PLC data are not addressed by the cycles if no entry is saved there.

5.7 VTC_CALIBRATION.TAB calibration data table

This data should be changed only if the machine kinematics used during tool inspection with the VT 12x is different from the kinematic model used during the calibration of the vision system (e.g., changed spindle kinematics).

For machines with more than one kinematic model (e.g., head exchange), you need to calibrate the vision system separately for each kinematic model. You need to create a copy of the VTC_CALIBRATION.TAB table for each kinematic model. When switching the kinematic model, you overwrite the table in the PLC:\VTC directory with the corresponding calibration data table.

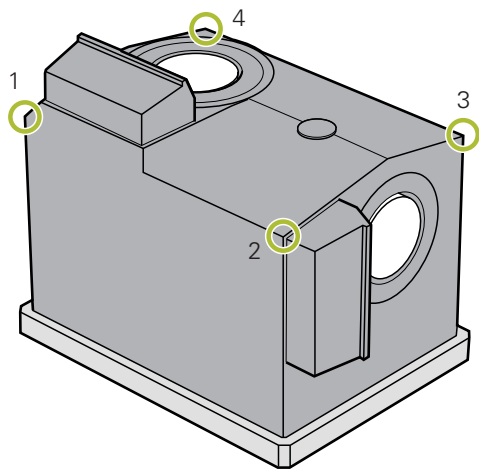
i If the tables differ only slightly from each other, you can also adapt the REF values in the calibration data table as needed.

i Do not use angular head kinematics in conjunction with Visual Tool Control (VTC).

The control saves the calibration data from the setup cycle (Cycle 620) in the calibration data table VTC_CALIBRATION.TAB. The table can be found in the PLC:\VTC directory. The calibration data are used, for example, as REF coordinates for the cameras' points of focus.

Name	Description
VERSION	Version number of the table VTC_CALIBRATION.TAB
TOP_REF_MAIN	Coordinate of the reference point of focus for camera 2 (main axis)
TOP_REF_SECONDARY	Coordinate of the reference point of focus for camera 2 (secondary axis)
TOP_REF_TOOL	Coordinate of the reference point of focus for camera 2 (tool axis)
SIDE_REF_MAIN	Coordinate of the reference point of focus for camera 1 (main axis)
SIDE_REF_SECONDARY	Coordinate of the reference point of focus for camera 1 (secondary axis)
SIDE_REF_TOOL	Coordinate of the reference point of focus for camera 1 (tool axis)
SIDE_DIRECTION	Viewing direction of camera 1 (angle relative to the main axis)
CYC620_CORNER_NR	Corner number for the calibration of the vision system ¹⁾
SIDE_DRC	Radius compensation value for plane of focus on camera 1
SIDE_DLC	Length compensation value for plane of focus on camera 1
TC_OFF_REF_MAIN	Reference offset in the main axis for temperature compensation
TC_OFF_REF_SECONDARY	Reference offset in the secondary axis for temperature compensation
TC_OFF_REF_TOOL	Reference offset in the tool axis for temperature compensation
TC_OFF_V_MAIN	Offset in the main axis for temperature compensation during comparative measurement
TC_OFF_V_SECONDARY	Offset in the secondary axis for temperature compensation during comparative measurement
TC_OFF_V_TOOL	Offset in the tool axis for temperature compensation during comparative measurement

¹⁾ CYC620_CORNER_NR allows you to define the sides of the camera housing to be probed with Cycle 620 (**VT SETUP**). The control probes the sides adjoining the selected corner. The coordinates of the individual measuring points are stored in the software.



5.8 Setting up the VTC application



You will find a detailed description of the VTC application and the possible settings in the VTC User's Manual.

6 VTC on the programming station

For testing purposes, you can also use the VTC PC software in conjunction with a programming station of the control. Install the VTC software and the programming station software on the same PC. Connect the VTC software to the programming station software by using the machine parameter **parameter** (118301); input value: `--opcuaHost=<IP | hostname>`.



The programming station uses the Ethernet adapter 1 as a default in the Virtual Box settings.

- ▶ Enter the IP or host name of the Ethernet adapter 1.

Initial configuration

If you use the VTC software in conjunction with a HEIDENHAIN programming station, please note the following differences in the initial configuration:

- When the programming station software is detected, two additional parameters will automatically be entered in **parameter** (118301):
 - `--vt121data=no`
 - `--simulateVTC=true`

Note about the vision system: Cycle 620 VT SETUP

To be able to use VTC, the vision system has to be calibrated with a touch probe. If this is not done, the control will display the error message **Vision system not calibrated** when a VTC cycle is called.

In order to use the VTC cycles on the programming station, calibrate the vision system as follows:

- ▶ Insert the touch probe with a TOOL CALL
- ▶ Run Cycle 620 VT SETUP

Testing the VTC cycles

In order to test the VTC cycles, it makes sense to integrate the 3D model of the VT 12x vision system into the machine model. The model is included with the cycles as a file (`vt121.m3d` or `vt122.m3d`). This enables you to follow the respective cycle sequence on the model.

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

☎ +49 8669 32-5061

info@heidenhain.de

Technical support ☎ +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101

service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106

service.app@heidenhain.de

www.heidenhain.com

