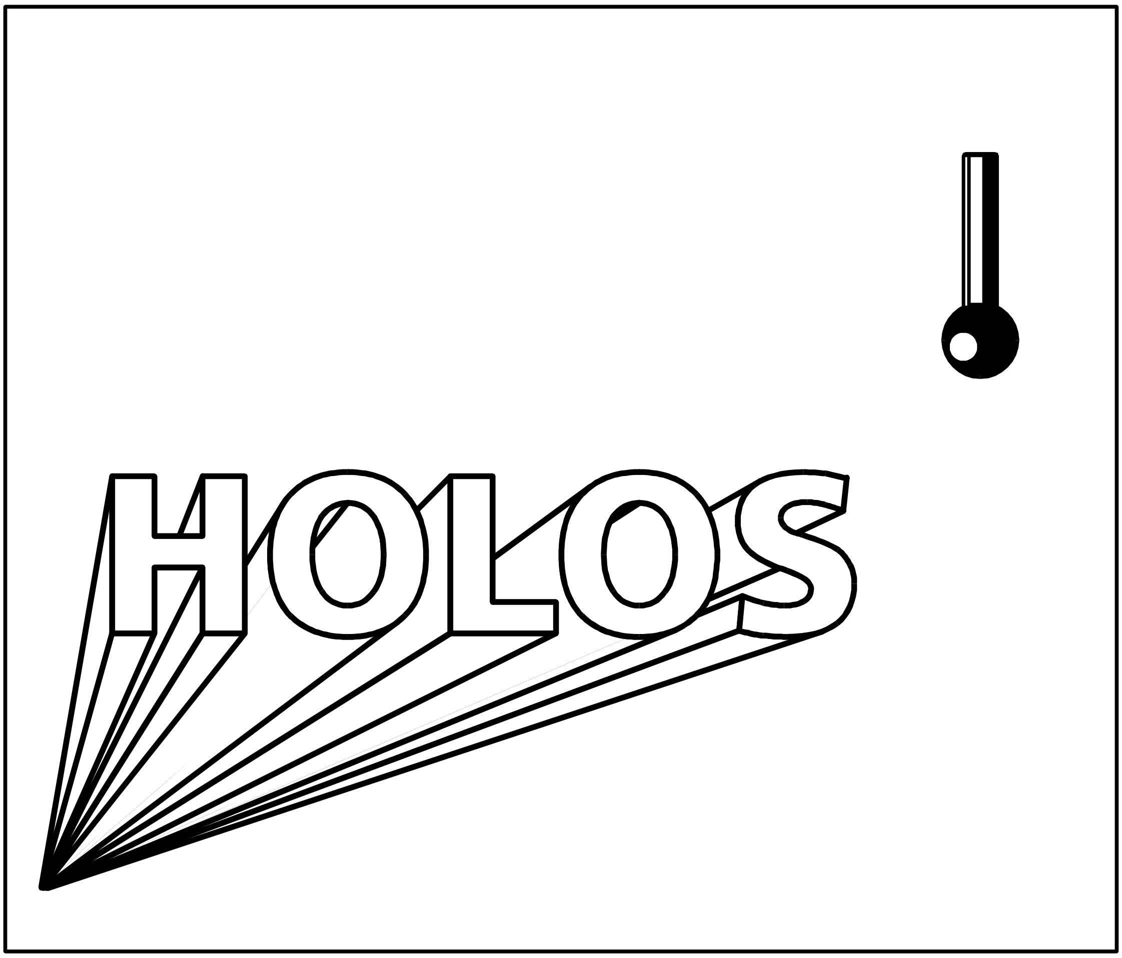
HOLOS-NT/UX



Instruction Manual

Order-No. 001658 02

Date: 07/00 V. 1.75(NT) / 1.65 (UX)

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Contents

1 General Information on HOLOS V. 1.75

1.1 About this operating manual

This operating manual describes HOLOS program version 1.75 (NT) and 1.65 (UNIX).

Screen masks are shown in the NT version as a rule.

Functions which generally or for the described function only apply for one of the two versions are marked as follows:
- (NT) for Windows NT
- (UX) for UNIX

The structure of the Operating Manual is oriented towards the main functions of the program.
The chapters and sections are subdivided into
- general information on each function
- step-by-step operating instructions.

In the appendix you will find both an index and a glossary of technical concepts.

Symbols

The following symbols are used to simplify the description of the operating sequences:

< ... > Function designation

Necessary action

Result of an action

Computer message

Cross reference to chapter, page or figure

Keyboard key, e.g. input key (example: input key)

1.2 System requirements (subject to technical modifications)

NT

On principle, HOLOS will run on any PC with operating system Windows-NT 4.0; the equipment listed below is recommended for optimal use.

- Intel Pentium II at least 266 MHz
- 21" monitor, screen resolution 1280 x 1024
- Open GL-capable graphics card
- Hard disk of at least 5 GB
- 128 MB RAM (more if processing large quantities of data)
- Network card for connecting to Zeiss measuring software
- Windows NT 4.0 service pack 4

UX

- UNIX measuring and evaluation software UMESS Revision 7.9 under HP UNIX V. 9.07 or Revision 8.0 under HP UNIX V. 10.2 for utilization of the full system functionality.
- HP9000 computer model 715/33/64 MHz or 712/60 MHz (as KMG [coordinate measuring machine] computer only with LAN connection) with at least 64 MB RAM and 1 GB hard disk.

1.3 Extent of the program's capabilities

The program has two main objectives:

- Measuring a known surface, which is currently stored in HOLOS internal format.
- Digitizing an unknown surface and subsequent generation of a VDA or IGES surface description.

Performance characteristics in detail

- HOLOS provides simple and convenient functions for measuring free form surfaces and regular geometries.
- A workpiece is aligned either via the relevant functions of the measuring software used, or via the HOLOS alignment functions. To do this, simply scan points on the workpiece and then arrange them graphically on the screen.
- With roughly aligned workpieces, the alignment can be improved by means of a 3D best fit.
- With roughly aligned workpieces, the alignment can be improved by means of a 3D best fit. Whether you are performing a surface or a trimming measurement, HOLOS will do the rest for you.
- For measuring in CNC mode, measuring cycles can be defined on the screen and processed in CNC mode.
- All data obtained are stored automatically.
- User-friendly graphics functions simplify the processing of the surface model.
- VDA and IGES data can be read in and generated using the VDA/IGES processor.
- The measurement results can be evaluated in a number of ways: as a numeric record or as a graphic record with vectorial, numeric or chromatic representation of the deviations.
- Simple tools are available for designing the graphic measuring record. With HOLOS you can design your screen as you wish.
- The digitizing functions allow simple digitization tasks to be easily performed, without much CAD knowledge.
- For digitization, functions are available for generating points, curves or surfaces.
- Surface transitions can be closed and smoothed out by means of simple CAD functions.

1.4 HOLOS-NT Revision 1.75 - innovations in comparison with Revision 1.7 NT

- **Double or multi-column operation** also with the NT-version.
- Measurement of **freeform surfaces:**
  "Contour" and "End point" measuring runs also in the NT-version.
- **Regular geometries:**
  Extended by the elements sphere, cylinder and cone.
  Extended by the links circle and plane.
  "Comment" enables a CNC-run to be interrupted for manual insertions.
  Rotate the element normal by 180°.
  Calculate circle, slot, rectangular hole as a hemisphere.
- **Edit nominal values:**
  Specification of names and tolerances for individual points possible.
  Specify angle position of DSE and RDS.
- **3D-best fit with Calypso:**
  Workpiece systems for a workpiece can be set from HOLOS for Calypso.
  The current workpiece system can be transferred from Calypso to HOLOS and saved.
- "Slice" model with a **clipping plane**, in order to define measuring points on internal sections.
- Display normal direction of curves, areas, patches and faces.
- Display Isolines on faces.
- Define, store and load different tolerance classes.
- Short protocol
- Global variables for the protocol header
- Extended scope of commands for script programming.
- Import / export Catia list.
- Graphic dumps in JPEG- and TIFF-format.

1.5 Hardware and software environment

NT

HOLOS is used on PC's with the Windows-NT operating system. A mouse is essential for operating HOLOS.

HOLOS is designed for CNC-controlled and manual coordinate measuring machines.

The interface between HOLOS and the coordinate measuring machine (CMM) is the CADLINK communication module. CADLINK is a UMESS measuring software option.

Connection is made with the Calypso measuring software via the Calypso OEM interface on CNC-controlled machines, and via the Calypso counter software on manual machines.

UX

HOLOS-UX is used on workstations with the UNIX operating system. A mouse is essential for operating HOLOS.

HOLOS-UX is designed for CNC-controlled and manual coordinate measuring machines.

The interface between HOLOS-UX and the coordinate measuring machine (CMM) is the CADLINK communication module. CADLINK is a UMESS measuring software option.

The UMESS measuring software is a general prerequisite for measurement (reference travel, alignment). In particular, UMESS-UX is a prerequisite for the scanning functions in HOLOS-UX.

1.6 Starting and ending HOLOS

Program start

NT Click on HOLOS in the HOLOS program group, and the program will start.

UX Enter the **start** command in the console window or press the **<Start HOLOS>** function key, if present, in the menu bar of the console window.

When you have started the program, the following steps are executed:

1. The system is initialized and, after a few seconds, the HOLOS user interface appears.
2. The connection to the coordinate measuring machine is made. (in the case of UX via CADLINK)
3. If a surface model already exists on the computer, a query will appear as to whether this model is to be loaded.

Program end

Click on the <End> function in the <File> menu.

1.7 Link-up with the measuring software

HOLOS can be linked up to a number of different measuring software packages. The communication link-up with the various software packages is carried out using a variety of mechanisms.

1.7.1 Link-up with UMESS-UX

NT

HOLOS communicates with the UMESS-UX measuring software via TCP/IP network services (sockets).
For communication with the UMESS-UX measuring software, you are not required to define any parameters, since this task is carried out by the installation tools included in the delivery. You simply need to define the name of the HOLOS computer as a parameter in the CADLINK module on the UMESS side.

UX

The following configurations are possible for link-up with UMESS-UX:

Configuration 1
Both software packages run on the same computer,
output occurs on the same screen.

Configuration 2
Both software packages run on the same computer,
output of the first software occurs on the system console,
output of the second software occurs on the screen of an X-terminal.

Configuration 3
Both software packages run on different computers with different screens, which are connected in a network.
HOLOS communicates with the UMESS-UX measuring software in all configurations via the UNIX network services. This means that even when both software packages are being operated on the same computer, the network services must be activated, even if no "real" network is present.

For communication with the UMESS-UX measuring software you are not required to define any parameters, since this task is carried out by the installation tools included in the delivery. You simply need to define the name of the HOLOS computer as a parameter in the CADLINK module on the UMESS side.

1.7.2 Link-up with UMESS 300 and UMESS 1000, manual Stiefelmayer software

NT

Communication with UMESS 300 and UMESS 1000 and the manual Stiefelmayer software occurs via a serial line (RS232). For this you need to define various parameters which are dependent upon the physical conditions of the conductor network (line length, possible sources of interference).
The parameters for the serial interface are defined with the <Serial interface> function in the <Parameters> menu>. See also Chap. 13.19

UX

For link-up with the UMESS 300 and UMESS 1000 measuring software only one configuration is possible, in which both HOLOS-UX and the measuring software each run on a separate computer. Communication with UMESS 300 and UMESS 1000 occurs via a serial line (RS232). For this you need to define various parameters which are dependent upon the physical conditions of the conductor network (line length, possible sources of interference).
The parameters for the are defined with the <Serial interface> function in the <Parameters> menu

🕮 See also Chap. 13.19

Standard parameter settings

Bits/Character	8
Stopbits	1
Baud-Rate	19200
Timeout	2

Define the parameters for the CADLINK option in the measuring software on the coordinate measuring machine's computer in a similar manner!

1.7.3 Link-up with Calypso (NT)

Link-up with the Calypso measuring software occurs via a TCP/IP network connection with Windows sockets. With manual machines this is via the Calypso counter software, with CNC-controlled machines, via the Calypso-OEM interface.

1.7.4 Link-up with SCANMAX (NT)

Link-up with SCANMAX also occurs via a Windows socket connection. In the SCANMAX software, SCANWARE, the CADLINK module is necessary.

1.8 Double column mode

In double column systems (multiple column systems) you can measure on different measuring columns with HOLOS

Prerequisites

The computers are linked to each other with the measuring software of the respective columns. I.e. in the case of HOLOS-UX with UMESS-UX and in the case of HOLOS-NT with UMESS-UX or Calypso

1.8.1 Set up multiple column mode (UX)

Multiple column mode is set up with the INSTALL installation tool.

- To activate multi-column operation, select <**multicantilever mode**>.
- Then activate all columns at which you wish to measure. HOLOS can only access the columns which you activate at this point.

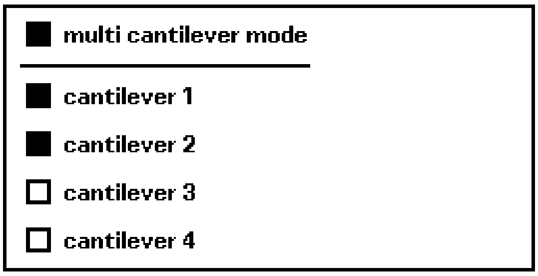


Figure 1-1

After activating multi-column operation for HOLOS, you must initialize the UMESS-UX program package at each column to allow communication with the HOLOS computer.

To do this, start the CADLINK option (DAW 2000). Select the <HOST> function (TAB-F7) and then the <PARAM> function (F5) to define the parameters for the link-up with HOLOS-UX.

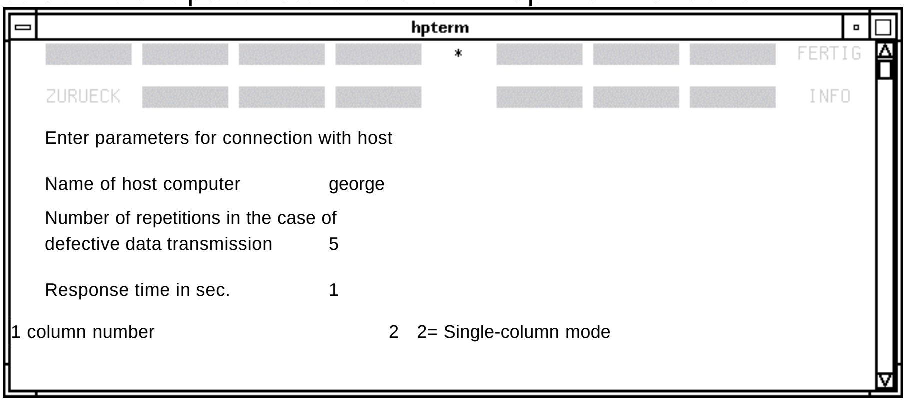


Figure 1-2

Name of host computer

In this field you enter the name of the computer on which you have started HOLOS-UX.
If you do not know the name, enter the command: uname -n on the computer on which you have started HOLOS. uname -n.

The operating system then displays the computer name.

Number of repetitions in the case of defective data transmission

You should not change this entry

Response time in sec

You should not change this entry

Column number

- Enter the number of the respective column (Column 1 Column 4).
- End the entry by confirming with <FINISHED> (F8).
- The UMESS-UX measuring software for the respective columns is now ready for communication with HOLOS-UX.

1.8.2 Measurement in multiple column mode (UX)

In order to start a measurement on a particular column, you must define in HOLOS-UX on which column a measuring cycle is to be started.

- To do this, select the <Receiving station (CADLINK)> function in the <Parameters> menu. Define the number of the column on which the subsequent measuring cycles are to be started.
- Start the measuring operations for the selected column.

> **NOTE**
>
> When programming macros, the respective column number is linked to a measuring cycle. I.e. the measuring cycles are always performed on the column which you have defined during programming for the respective measuring cycle.

Manual measuring cycles selected in HOLOS

- When making manual measurements, you must always define at which column the measuring operation is to be performed, prior to making the measurement.

1.8.3 Setting up multi-column operation (NT)

Prerequisites

The link up of the columns must have been performed in Calypso, and the machine coordinate systems brought into line with each other.
The columns are then disconnected in Calypso and the slave is informed that it must take account of the transformation matrix.

In the <Parameter> menu select the commands <Measuring software> - <Columns (UNIX, Calypso, Counter)>

The selection window for the columns is displayed:

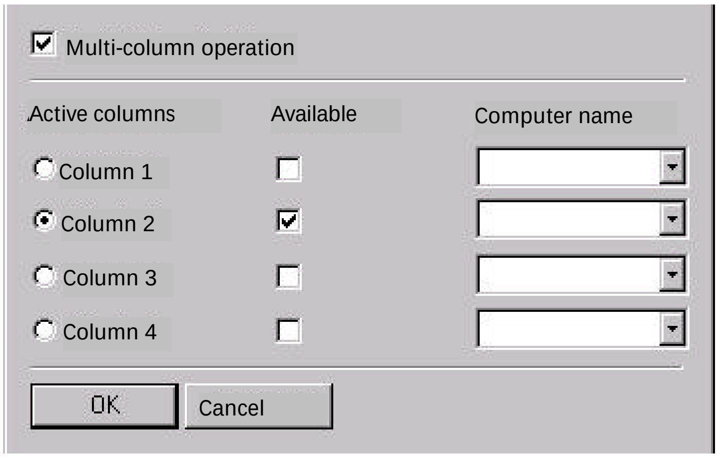


Figure 1-3

Define up to four columns as available, and if you are working with Calypso, specify the name of the computer on which Calypso is running for each available column.

Using an option key, select which column is to be active, i.e. the column to which the measured values are transmitted.
Only one column can be active at a time. You can switch from one column to another either here in the parameters, or select another available column with the indicator in the graphic menu bar.

1.8.4 Measurement in multiple column mode (NT)

In the parameters or using the indicator in the graphic menu bar, select the column to which the measured values are to be transmitted.
In the indicator the active column can always be identified by the yellow boundary.

Then start the measuring runs for the selected column.

NOTE

When programming macros, the respective column number is linked to a measuring run. I.e. the measuring runs always occur at the column which you have defined in the programming for the respective measuring run.

As soon as a transformation (alignment) is transferred (e.g. via the <3D-best fit> function), you will be asked to which column it is to be transferred:

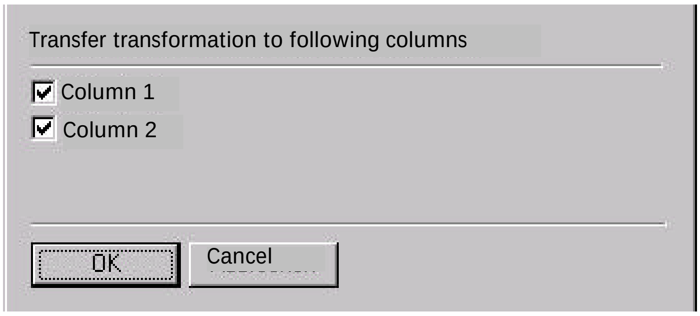


Figure 1-4

All columns that are defined as available are displayed here.

CAUTION

All columns must work with the same workpiece system.

1.9 Screen layout and operation

1.9.1 Screen layout

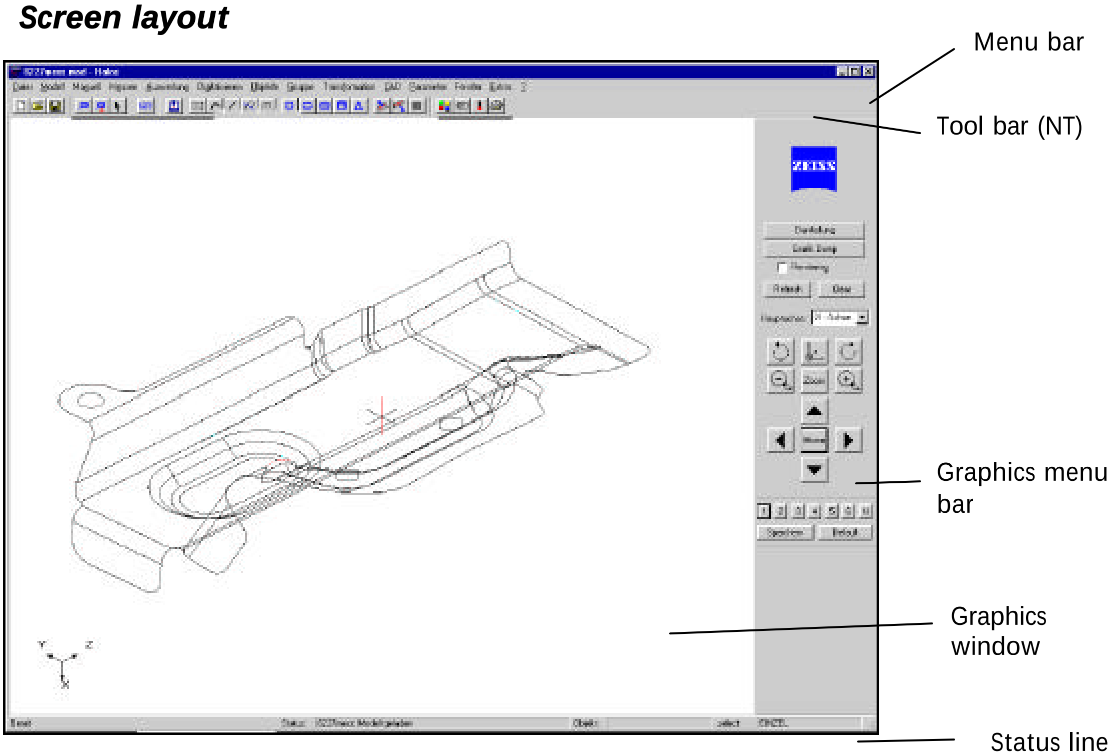


Figure 1-5

Menu bar

The menu bar contains a summary list of all the main program functions. Each main function branches into a menu with subfunctions.

Tool bar

The symbols for quick activation of functions are located in the tool bar. The tool bar consists of various components that can be positioned at random on the screen.

Graphics menu bar

The graphics menu bar contains the complete set of functions that are used to change the graphic display of the model.

Graphics window

The graphics window is where the model is displayed in its current state of processing. The schematic coordinate axis at the bottom left shows in what position the observer is viewing the model. The measuring cycles are defined interactively in the graphics window.

Status line

Important information appears in the status line.

Status:	Last program action, e.g. data have been stored under a defined name.
Model:	Name of the active model.
Object:	Name of the last selected object. Objects are. e.g. curves, surfaces, faces etc.
Select:	Selection mode, e.g. individual or group. The selection mode is modified via the functions <Define group>, <Objects - Analysis> and <Measuring points - select>.

In addition to the screen elements already described, dialog windows are opened to assist in operation of the functions. These windows can be moved and processed as is usual in "Windows" interfaces.

1.9.2 Invoking functions

A function is called up by clicking on the left mouse button.

If you click on a main function in the menu bar, a menu will be opened listing all the subfunctions. Any subfunctions that display an arrow to their right then branch off into further subfunctions.

1.9.3 Activating options

Numerous functions are controlled via dialog windows, in which you need to select or deselect options.

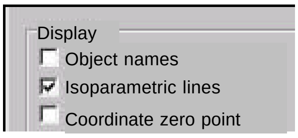


Figure 1-6

An option is activated by clicking on the display field with the left mouse button.

1.9.4 Text entry

Numerous functions are controlled via dialog windows, in which text needs to be entered.

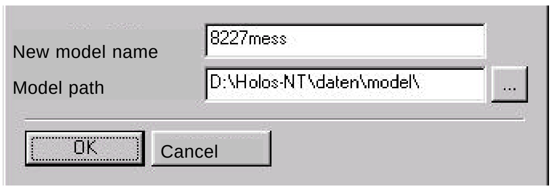


Figure 1-7

Click on the entry field with the left mouse button, so that the cursor flashes in the entry field.

If you want to overwrite an existing entry, keeping the left mouse button pressed down, drag the cursor over the text until it is completely highlighted. Now you can simply enter the new text. Confirm your entry by clicking on <OK> in the dialog window.

The <Cancel> key cancels the function and closes the window.

1.9.5 Selecting elements

Numerous functions are operated via dialog windows in which a selection must be made (files, objects, models etc.).

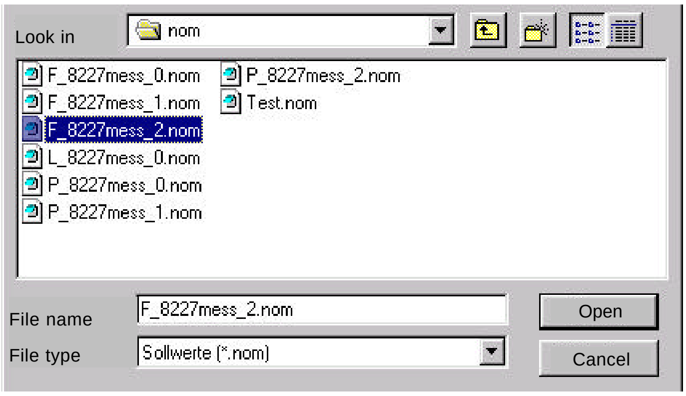


Figure 1-8

You can select the elements as follows:

- Click on the element with the left mouse button and then click on <OK> in the dialog window.
- ·Click on the element twice with the left mouse button, very quickly (double-click). The element will be adopted immediately.
- To select a number of elements, press the SHIFT key and click on each element once, or alternatively drag across the elements whilst keeping the left mouse button pressed down. Then click on <OK>.
- To select several elements which are not positioned directly next to each other, first press CTRL and then click on the relevant files.

Click on <Cancel> to close the dialog window.

1.9.6 Menu editor

In the <Extras> menu (UX: <Admin>) you can use the <Menu editor> function to place frequently required functions into a user defined menu.

The user-defined menu is activated by clicking the right mouse button in the graphics window and then releasing it without moving it (UX: <Ctrl>key and left mouse button).

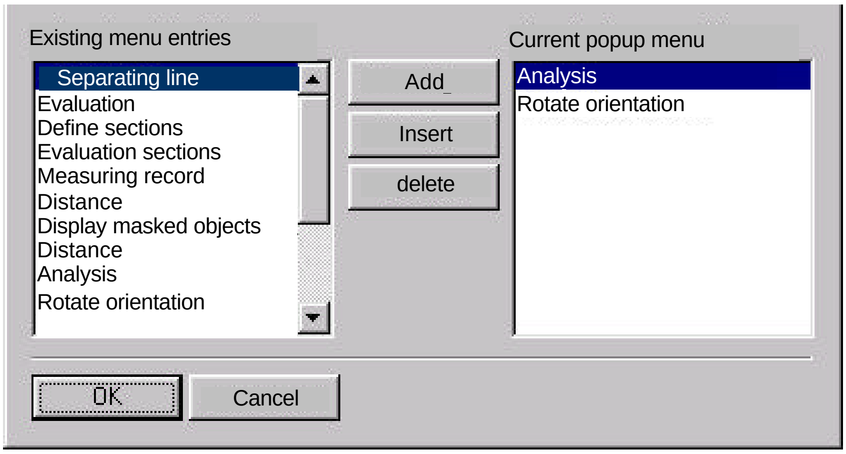


Figure 1-9

Add

To add a function from the "Menu entries available" list to the "Current popup menu" list:

Select a function from the "Menu entries available" list and click on <Add>.

The selected function is transferred to the "Current popup menu" list.

Insert

To insert a function from the "Menu entries available" list into the "Current popup menu" list:

Select a function from the "Menu entries available" list.

Select a function from the "Current popup menu" list and click on <Insert>.

The function is transferred to the "Current popup menu" list above the selected position. If no function is selected in the user menu, then the new function is added at the end.

Delete

To delete a function from the user menu:

Select a function from the "Current popup menu" list and click on <Delete>.

The selected function is removed from the list.

Then click on <OK>, in order to accept and save your defined menu.
Press <Cancel> to exit the menu editor without adopting the adjustments made.

Additional functions in UX

Basis

This key displays all functions for the "Digitize" option in the "Functions available" list, which can be adopted into a user-defined menu.

Digitizing

This key displays all functions for the "Digitize" option in the "Functions available" list, which can be adopted into a user-defined menu.

Autoscan

This key displays all functions for the "Autoscan" option in the "Functions available" list, which can be adopted into a user-defined menu.

Save

Save the defined menu to the hard disk.
Your menu will then be available again when you restart HOLOS. If you do not save a newly defined menu, it will only be kept locally and will not be available when HOLOS is next started.

Restore

Restore a saved menu and reject the changes made.

1.10 Revision level

When making inquiries to the Hotline, you should know which revision level you are using. You can obtain information about this by clicking on ? and <Info about HOLOS> in the menu bar with the left mouse button.

Figure 1-10

2 Working with HOLOS

2.1 Measuring surface models

This chapter gives a concise account of some of the operating steps that you would typically carry out to measure models described as free form surfaces.

As a prerequisite, the appropriate probes must already have been mounted and calibrated.

1. Read the VDA or IGES data record for the model to be processed into your HOLOS workstation and convert it into the HOLOS internal data format.
   Functions: NT: <File>-<Import/Export>-<Import VDA/IGES>
   UX: <VDA>-<VDA postprocessor>
2. After conversion, load the model into the HOLOS main memory.
   Functions: <Model>-<Load model>

   The model is displayed on the screen with the relevant settings. Modify the graphic display if necessary (zoom, poisition on the coordinate measuring machine‘s work plate, position of the coordinate system etc.).
3. Clamp the workpiece to the work plate of the coordinate measuring machine and align it.
   To carry out the alignment, use the alignment function in your respective measuring software (see the UMESS Operating Manual). When using UMESS 1000 or UMESS-UX measuring software, ensure that the workpiece and the workpiece positioning system are identical, as otherwise it is possible for collisions to occur in automatic measuring runs.
   For the alignment, use existing regular geometry elements whose position in space is known. The alignment will then generally be correspondingly accurate.
   If this is not possible because the workpiece does not have any regular geometry elements, align the workpiece as precisely as possible by means of the appropriate UMESS measuring software functions or with the HOLOS alignment functions.

4. A precise alignment can then be carried out by a best fit of manually probed points. However, the first alignment must be known at least sufficiently accurately for probings on the workpiece surface to be detectable by HOLOS.
5. Change to the CADLINK software option in your measuring software
   ADR PROG 369 (UMESS 300)
   Direct selection 2000 (UMESS-1000)
   Direct selection 2000 (UMESS-UX)
6. You can now probe points on the surface of the workpiece. If you wish to perform probings in manual mode (patch identification), you can start immediately in the UMESS-UX and UMESS-1000 measuring software.
   For UMESS 300 measuring software you must define the mode for manual probing (Softkey <PATCH-IDENT>).
7. When the alignment is correct, HOLOS recognizes the probing points and projects them onto the loaded workpiece. Deviations are shown on the screen or in the relevant window.
8. If the alignment was not sufficiently precise, you can achieve an improvement in the alignment by a best fit of the measurement results (3D-best fit).
   Functions: <Manual>-<Patch identification>-<3D-best fit>
   or: <Evaluation>-<3D-best fit>, if you have already saved the probed measuring results.
9. In CNC mode you can define and execute measuring cycles.
   Functions: <Measure>-<Define measuring cycle>-
   <Grid>
   <Line>
   <Meas. points>
   <CAD points>

   Before starting measuring cycles, check the parameters for the measuring cycles (both in the parameter menu of HOLOS-NT and also in the CADLINK measuring software option). Previous definitions for backaway paths or clearance planes can result in collisions with a new workpiece.
   Functions: <Parameters>-<Measuring cycle>
   The measurement results will be displayed on the screen after the measurement together with the defined parameters.

10. Subsequent evaluations can be obtained via the <Evaluation> menu.
    Functions: <Evaluation>-
    <Actual data >
    <Chromatic co-ordinates >
    <Sections>

11. Print the graphic measuring record on the <Graphics dump> graphic printer. A numeric measuring record can be obtained via the <Measuring record> function in the evaluation menu.
    UX: <Plotter> for printout on plotter.

2.2 Generating surface model

This chapter gives a concise account of certain operating steps that you would carry out to digitize models whose mathematical description was previously unknown, and how you obtain the description of the workpiece surface.

A prerequisite is that the appropriate probes must already have been mounted and calibrated.

Essentially, you determine the quality of the digitized surface data even before the actual digitization process, during the preparation and division of the surfaces or surface segments on the workpiece surface.

To carry out this preparatory work, knowledge is required in terms of the theory and the fundamental mathematical background. These topics form an integral part of the training course for the HOLOS software package, and will therefore not be expanded upon at this point.

1. Divide the workpiece surface into surfaces or surface segments. If possible, mark the division with a marker pen. This will make it easier to probe the boundaries of the surface or to obtain reproducible results at a later time.
2. Clamp the workpiece to the work plate of the coordinate measuring machine and align it. The simplest method is to orientate the position of the workpiece to any existing regular geometry elements, since it will then be possible to reproduce the position of the workpiece at any time without difficulty.

To carry out the alignment, use the alignment function in your respective measuring software (see UMESS Operating Manual). When using UMESS 1000 or UMESS-UX measuring software ensure that the workpiece and the workpiece position system are identical, as otherwise it is possible for collisions to occur in automatic measuring cycles.

3. You can now obtain the first approximations of your surfaces or surface segments by manually probing the previously defined boundaries of the surface segments.

   For this, make use of the available functions in the respective CADLINK measuring software:
   DIGIT POINT to digitize individual points
   DIGIT CURVE to digitize curves
   DIGIT GRID to digitize surfaces
   The explanation of these functions forms an integral part of the training course for the HOLOS software package, and will therefore not be expanded upon at this point.

   Since the acquisition of further points to determine the precise mathematical description of the surface segments can, for the most part, take place automatically, in principle the following statement applies when manually probing the surface boundaries:
   As few points as possible, but always as many points as are actually needed!

4. Depending on the digitizing function used in CADLINK you must define the surface segments using the interactive-graphics.

Digitizing function	Action in HOLOS
DIGIT POINT	Definition of the points of a curve Generation of the curves Definition of the curves of a surface Generation of the surface
DIGIT CURVE	Definition of the curves of a surface Generation of the surface
DIGIT GRID	No further actions required

   Each of the possible digitizing functions has advantages and disadvantages in terms of the resulting surface transitions and the manual work before and afterwards. These topics form an integral part of the training course for the HOLOS software package and will therefore not be expanded upon at this point.

5. After defining the first approximations of the surface segments, you can improve the accuracy of the mathematical description by recording further points.
   Functions: <Digitize> - <Digitize surface>

   Define the number of points for the points grid to be recorded in the relevant dialog window. Following confirmation the points will be displayed on the screen and automatically measured. After processing the points grid, the surface will be re-calculated.

   Experience has shown that with correct application of the digitizing function (sensible division of the workpiece surface) a points grid of 6 x 6 points (but a maximum of 7 x 7 points) is quite sufficient to give a satisfactory description of the surface segment. If you find that you cannot achieve sufficient accuracy even with the higher points density, then you must break open the relevant surface segment (further sub-division) and digitize once again.

6. Each time you will receive a message in the status window giving the standard deviation of the probed points for the calculated surface.

   If you need further information concerning the accuracy of the calculated surface segments, then use the functions for measuring free form geometries. The quickest method of checking is via manual probing of the generated surface on the surface of the workpiece (patch identification).

7. Depending on the result of this check, you can obtain further information on the surface geometry by increasing the points density (repetition of steps 5-6).

8. Via the <Patches-> Surface> CAD functions you can close transitions between the individual surface segments and define the degree of continuity for the surface transition.

   During digitization with CADLINK, surfaces that only have one surface segment always occur. Since with the <Patches-> Surface> function surfaces with a number of segments can be defined from single-segment surfaces, this function is also described as re-parameterization.
9. The quality of the generated workpiece surface is graphically displayed as explained in Chapter 2.1. The best statement in the graphic display is obtained via an evaluation with markings or by displaying the chromatic co-ordinates of the probed points grid.

The workpiece descriptions generated with the functions described above can also be transferred to higher level systems as a VDA-FS or IGES file.
NT functions: <File>-<Import/Export>-<Export VDA/IGES preprocessor>
UX functions: <VDA preprocessor>-<Geometry>

For a number of functions copies of surfaces are generated and the original surfaces are masked (e.g. during re-parameterization). If the masked surfaces are not to be transferred to the CAD system then they must be deleted before generation of a VDA-FS or IGES file in HOLOS.

- Select objects (NT)
- Delete selected objects (NT)
- Delete masked objects (UX)
- Define VDA header
- Generate VDA or IGES file

3 Graphic Representation

This chapter describes the functions on the graphics menu bar. These are used to change the graphic representation of the model.

The graphic representation of the model in its respective processing status fulfils important tasks in HOLOS-NT. On the one hand, you are able to judge the quality of the generated surface with the aid of the graphics (e.g. with a chromatic evaluation), and on the other, you are able to define measuring cycles using the interactive graphics on the screen. The projection of the model on the screen is particularly important for the second of the above-mentioned tasks.

As well as the functions on the graphics menu bar, the graphics parameters are also crucial for the graphic representation of the model.

🕮 See Chap. 13.1

Graphic representation

The graphics menu bar contains the following functions:

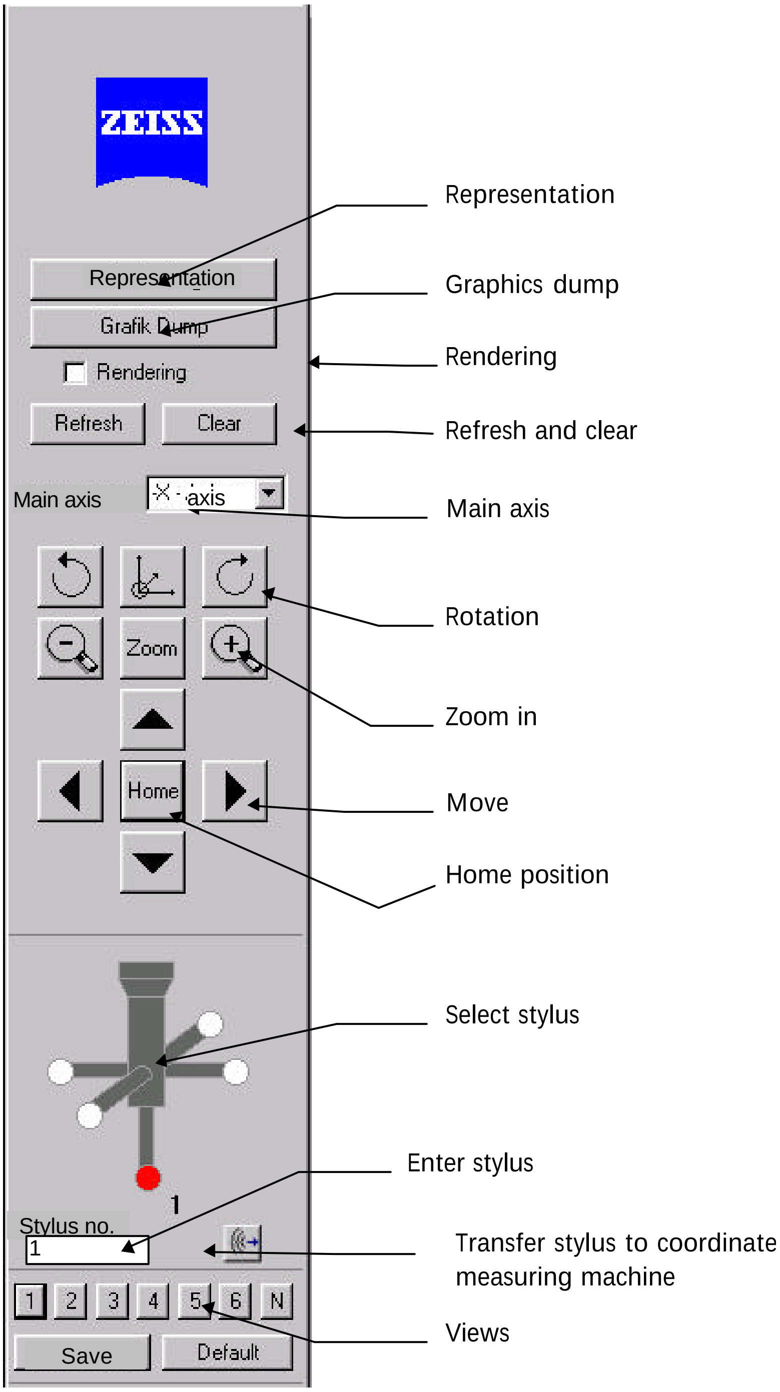


Figure 3-1

The graphics menu bar in UX contains the following functions:

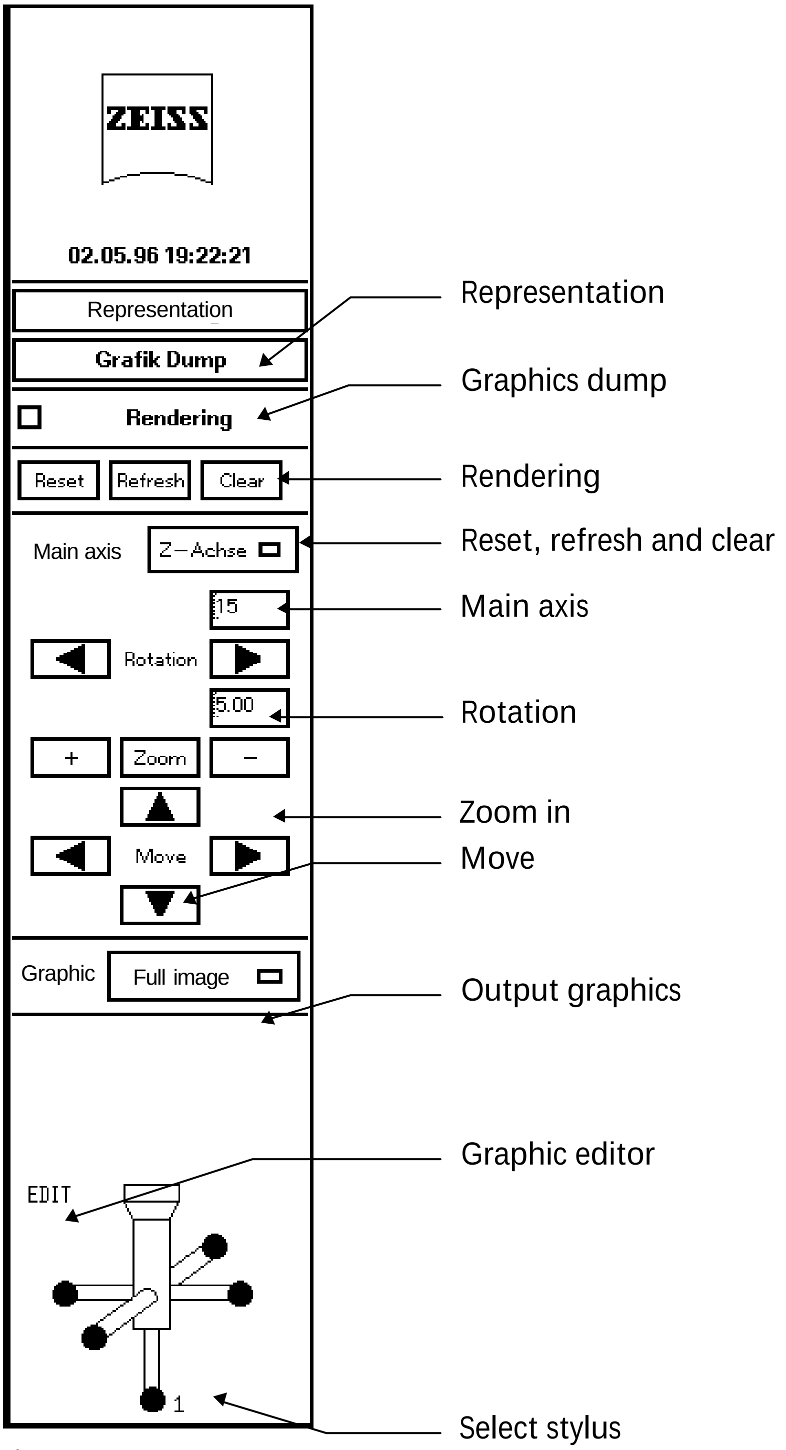


Figure 3-2

3.1 Representation

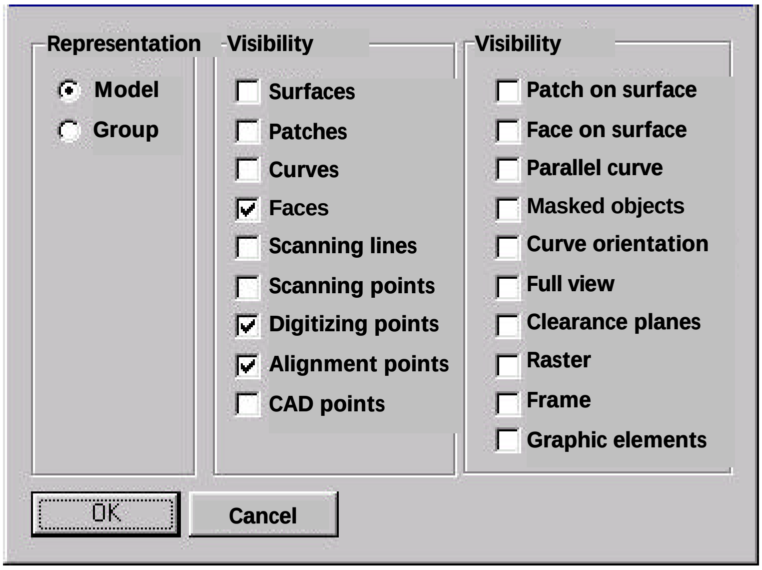


Figure 3-3

The <Representation> function allows you to define which objects are visible on the screen.

Option	**Meaning**
Model	shows complete model
Group	shows only the defined group
Surfaces	shows all surfaces
Patches	shows all patches
Curves	shows all curves
Faces	shows all faces
Scanning lines	shows all scanning lines (generated by laser or stylus)
Scanning points	shows only the points of a scanning line
Digitization points	shows all points that have been generated by the DIGIT POINT function in CADLINK
Alignment points	shows manually scanned alignment points
CAD points	shows points adopted from CAD description (VDA/IGES)

Patch on surface	shows all patches on the surface when this surface is clicked on
Face on surface	shows all faces on the surface when this surface is clicked on
Parallel curves	shows all parallel curves
Masked objects	masked objects are always displayed in brown
Curve orientation	shows the orientation of the curves
Clearance planes	shows the clearance planes that surround the workpiece. You can define the distance of the clearance planes to the workpiece on the <Clearance planes> parameter page. You can directly select clearance planes on the screen and apply them to the measuring cycle parameters.
Raster (NT)	shows a line raster in the graphic representation
Frame	shows the frame defined for the graphic measuring record on the screen
Graphic elements (NT)	all defined graphic elements (lines, circles, texts, bit maps) are displayed.

The definitions for representation are stored for each model and each group of a model and will be in the previously defined status after loading of the model or group.

> ***NOTE***
>
> Only displayed objects can be processed. In other words: if there is no response to a function, check first of all whether the representation for the object is selected.

3.2 Graphics dump

The <Graphics dump> function is used to output the contents of the graphics windowto a graphic printer or into a bitmap file.

NT

The parameters for outputting graphics dumps are set via the <Parameters>-<Print> function. The top section contains the settings for the graphics dump, the middle section contains those for the measuring record, and at the bottom the printer output for the graphic record is set.

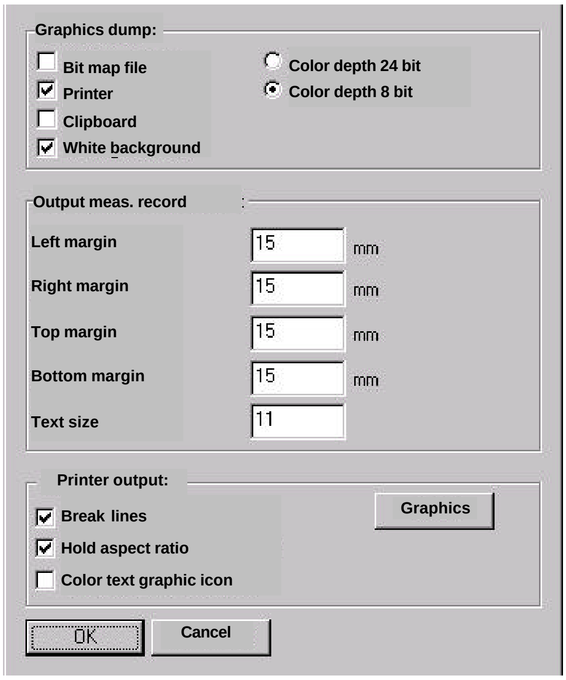


Figure 3-4

UX

Set the following in a dialog window:
- Paper format: A3 or A4
- Inversely display colors: display on white background (an inverse representation saves time and ink).
- Send graphics to the printer: a “GDUMP_0“ file is set up in the directory /users/holos/daten/tmp in PCL format and deleted after printing.
- Number of copies for plotter output.

Print the graphic out or cancel the procedure with "yes" or "no". A query will then appear, which gives you the opportunity to save the graphic as a file. A file name is requested if the corresponding option is selected in the system parameters.

3.3 Rendering

This switch is used to activate the <Rendering> function>. In Rendering mode the workpiece surface is displayed in color, illuminated and without concealed edges.

With very large quantities of data or with complex workpiece geometries, the formation of the graphic representation may be slower.

See Chap. 13.2, 13.3.

3.4 Refresh and clear

These functions are used to re-create the graphics window in a number of different ways.

Refresh	The content of the graphics window is re-created in its latest status.
Clear	Various displays, such as measured values and deviations, are deleted.
Reset (UX)	All values for Move, Zoom and Rotation are reset to the original status.

3.5 Define main axis

This function is used to define the main axis. The definition of the main axis is carried out in a manner similar to that for defining the coordinate system on the coordinate measuring machine, when a different choice of axis is selected.

Example

X-axis upwards, instead of Z-axis = rotation of the coordinate system

After “Home“ (NT) or “Reset“ (UX) the main axis is always upwards.

3.6 Rotate image

There are three ways of rotating the image in the graphics window:

Rotate with the <Rotation> function in the graphics menu bar.

Enter the step width for the rotation in the graphic parameters text field.

Text entry, see Chap. 1.9.4

Click on the arrows.

With each click the image will be rotated by the specified angle about the axis of rotation that was clicked on.

Rotate with the right mouse button (possible at any time).

Position the mouse in the graphics window.

Keep the **right mouse button** pressed down and move the mouse downwards or to the left.

The image rotates clockwise.

Move the mouse upwards or to the right.

The image rotates counter-clockwise.

You can follow the image rotation directly on the screen during this procedure. With large quantities of data, however, constant re-creation of the graphic window requires a lot of time.
In order to avoid this, you can use the <Parameters - Graphics> function to set that only the axis intersection is rotated in conjunction with movement of the mouse.
See Chap. 13.1
The image is only shown in its final position when you release the right mouse button again.

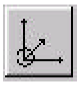
Rotate with the arrow keys on the keyboard (NT)

Press the → or ← key on the keyboard

The image rotates in the respective direction.

Axis of rotation

The axis of rotation for all procedures is the selected coordinate axis. The axis of rotation is color-highlighted in the axis intersection at the bottom left. You can easily change the axis of rotation by clicking on the desired axis in the axis intersection.

Point of rotation (NT) 

The point of rotation for rotating the graphics is the center of gravity of the workpiece. To change the point of rotation, click on the key shown above and then on any point in the graphic representation. In order to revert to the original point of rotation, double-click on this key.

In the case of regular geometry elements, you can place the point of rotation e.g. on the mid-point of a circle or the mid-point of a hemisphere, in order to rotate this element so that the measuring points can be set on all sides.

3.7 Image magnification or reduction

There are four ways to magnify or reduce (using zoom) the size of the image in the graphics window.

Using the <+> and <-> keys in the Zoom function.

Enter the step width for zooming in the graphics parameters text field.

Click on <+> or <->.

The image is magnified or reduced step-by-step.

Using the <Zoom> key in the Zoom function.

Click on <Zoom>.

Keeping the left mouse button pressed down, draw a rectangle over the area that you want to magnify in the graphics window.

As soon as you release the mouse button, the area enclosed by the rectangle will be displayed in magnified form.

Using the middle mouse button (possible at any time).

Keep the **middle mouse button** pressed down and move the mouse to the left or downwards.

The image will be reduced.

Move the mouse to the right or upwards

The image will be magnified.

Using the arrow keys on the keyboard (NT)

Press the “up” or “down” arrow keys on the keyboard.

The image becomes larger or smaller.

3.8 Move image

There are three ways to move the image in the graphics window.

Move using the arrow keys in the <Move> function.

Click on an arrow key.

The image is moved in the direction that was clicked on.

Move using the left mouse button (possible at any time).

Position the mouse in the graphics window.

Keeping the **left mouse button** pressed down, pull up a translation vector that specifies where the image is to be shifted to.

As soon as you release the mouse button the image will be moved.

Move using the arrow keys on the keyboard (NT)

Press an arrow key on the keyboard and the <Ctrl> key simultaneously.

The image is moved in the corresponding direction.

3.9 Home position (NT) / Reset (UX)

All values for Move, Zoom and Rotation are reset to their initial status.

Press on <Home> with the **left mouse button.**

The graphic representation will revert to its initial status.

3.10 Select stylus

This function is used to select the stylus for the subsequent measured values. All measurements will be displayed on the screen in the color of the currently active stylus.

A prerequisite for the selection of a stylus is that the stylus has been activated with the function <Parameter - Stylus>.
See Chap. 13.8

If no specific stylus has been assigned for a measuring run, you should still utilize the Parameter function. The stylus that has been set on the control console will then be assigned.

3.11 Storing and retrieving graphic views

For quick definition of a specific view of the workpiece you can store the graphic display setting and retrieve it when required.

Any number of settings can be stored. The settings are stored with a name in the appropriate workpiece directory.

For quick definition of specific views , you can define six so-called ”Quick positions” Quick positions. In the basic setting these six positions are pre-assigned with the six projection levels of a cube.

Operation

Call up a defined setting for the graphic representation by clicking with the mouse on the buttons in the dialog window below:

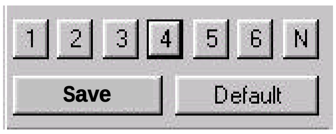


Figure 3-5

Buttons 1... 6

Use buttons 1... 6 to define the settings for the views of the basic setting or the ”Quick positions”.

Button <N>

When you click the **<N>** button, a catalog is displayed with a list of all named settings.

Define the modification to the graphic setting by selecting one of these settings.

Default

This function pre-assigns the basic setting to the "Quick positions".

In the basic setting these six positions are pre-assigned with the six projection levels of a cube.

Save view

To save a specific view select the < Save > (NT) or <Save view> (UX) function.

Click on the <Save> function and then on a button 1...6.

The view is stored under the relevant number.

If you want to save the graphic view setting under a particular name, select the <N> button.

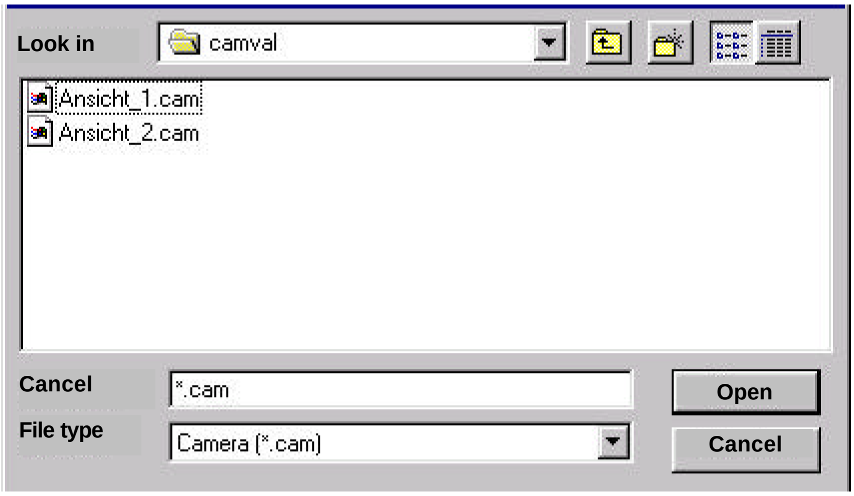


Figure 3-6

A dialog window appears in which you can enter the name under which the temporary setting for the graphic view is to be saved.

Save the view in question with <OK>.

If you close the dialog window using <Cancel> (NT) or <Close> (UX), the setting in question will not be saved.

3.12 Representation of projections (NT)

The image is output to the graphics window in four views (in the three base planes and three-dimensionally) if you activate the <Divide> function via the <Window> menu.

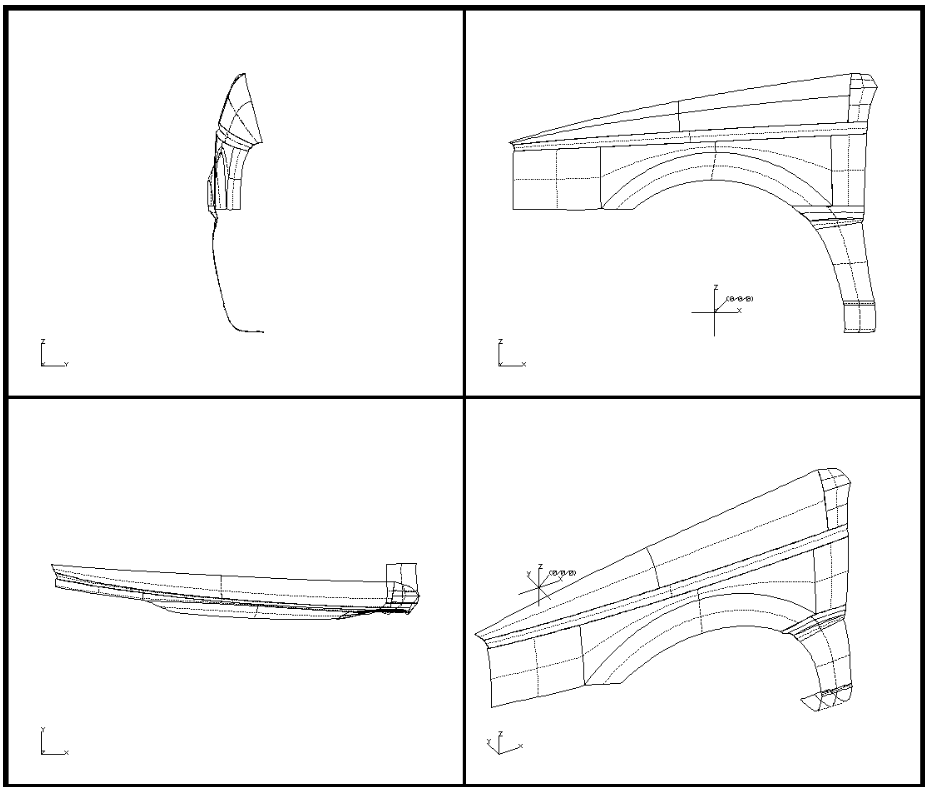


Figure 3-7

You can position the division as desired by moving it with the mouse button.

You can restore any of these four views to full representation by double-clicking on the view with the **left mouse button**. In this "Full" representation, a coordinate counter also runs at the top left for projections showing the position of the mouse.

If you have defined a raster with the <Representation> function, then this will be displayed in the respective views.

📖 See Chap 13.1

3.13 Reverse graphic representation (UX)

With the <Graphics> function you can select between different output options: full image, partial image, screen and plotter.

Full image

The image is output in one view in the graphics window. In full image you can alter the view using the functions described above.

Partial image

The image is output in four views (in the three main planes and three-dimensionally) in the graphics window.

You can restore any of these four views to full representation by clicking on the view with the **right mouse button**. In this "Full" representation, a coordinate counter also runs at the top left for projections, showing the position of the mouse.

If you have defined a raster with the <Parameter - Graphics> function, this will be displayed in the views of the main planes.

See Chap 13.1

Screen

The <Screen> function displays on the screen how the content of the graphics window is to be output to the plotter. The displayed record header is set up with the <Record header> function in the <Parameters> menu.

UX: To define a variable logo in the record header, see Appendix D.

Plotter

With this function you can output the content of the graphics window to the plotter.
If you have defined an inkjet printer for plotter output during installation, the format will be queried.

3.14 Graphic editor (UX)

The <EDIT> function in the graphics menu bar starts the graphic editor for designing the graphic measuring records.
The graphic editor allows you to generate, modify and delete graphic elements on the screen.
Graphic elements include:

- Texts
- Lines
- Rectangles
- Circles

A further function provided by the graphic editor is the ability to move deviations which are represented in deviation icons on the screen.

You can store graphic elements and reload them at a later point in time. Thus, for example, you can generate your own frame layout for the design of the graphic measuring record

3.14.1 Generate graphic elements

Click on <EDIT> in the graphic menu.

A window appears with the functions for designing the graphic measuring record:

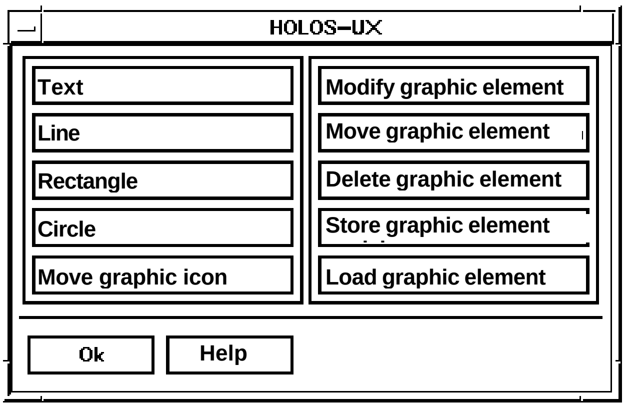


Figure 3-8

Select the desired graphic function

Text Click on the text position on the screen with the left mouse button. An input window is opened for the text. Enter text and confirm with OK.

Line Click on the start of the line on the screen with the left mouse button and, keeping the mouse button pressed down, drag to the end of the line, then release the mouse button.

Rectangle Click on the first corner point on the screen with the left mouse button and, keeping the mouse button pressed down, drag as far as the second corner point, then release the mouse button.

Circle Click on the mid-point of the circle on the screen with the left mouse button and, keeping the mouse button pressed down, drag to the required size, then release the mouse button.

The graphic element is positioned on the screen.

3.14.2 Process graphic elements

Click on <EDIT> in the graphics menu.

The window containing the functions for designing the graphic measuring record appears.

Select the desired processing functions

Move graphic icon Move overlapping deviation icons on the screen:
Pull graphic icon by its arrow tip to the new position, keeping the left mouse button pressed down. Instead of the arrow tip, graphic icons that have been already been moved have a handle on their left side, which is used to pull them.

Clear: removes all graphic icons from the display.
Reset: moves all graphic icons to their original position.

Modify graphic element Click on the desired graphic element with the left mouse button and, keeping the mouse button pressed down, pull it to a new position or change the size. If text has been selected, an input window will open for the modifications

Move graphic element Click on the desired graphic element with the left mouse button and pull to a new position, keeping the mouse button pressed down.

Delete graphic element Click on the desired graphic element with the left mouse button and then release the mouse button.

Store graphic elements An input window is opened for the file name. Select the “Global storage“ option if you wish to load the stored graphic elements from any model at will. Locally stored elements, on the other hand, can only be loaded from the currently active model.

Load graphic elements Specify in a selection window whether local or global data are to be loaded. A selection window is then displayed for the elements found, in which you select the required file by clicking on it and load with OK.

3.15 Scale plotting

You can output the 2D-projection representations of your graphics to scale on a plotter.
The graphics are output to the output device which you have defined as the plotter in the HOLOS installation tool or during printer set up in the <Parameters> menu.

Call up the <Scale> function in the <Graphics> menu of the graphics function bar on the left edge of the screen (UX) or <Model> - Scale plot (NT)

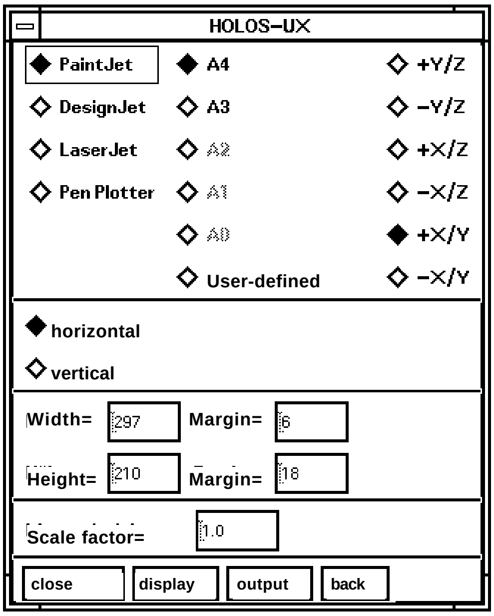


Figure 3-9

Graphic representation

Define in succession the output device, the inserted paper format (see paper format) and the projection plane in which the graphics are to be displayed and the scale factor for outputting the graphics (M1:1 = scale factor 1; M2:1 =

Functions in the bottom bar

close	Closes the open window.
display	Displays the graphic representation with the set parameters on the screen.
output	The graphic representation is output with the set parameters on the plotter / graphic printer (see next figure)
back	This function allows you to change back from the scale representation to the previously active representation.

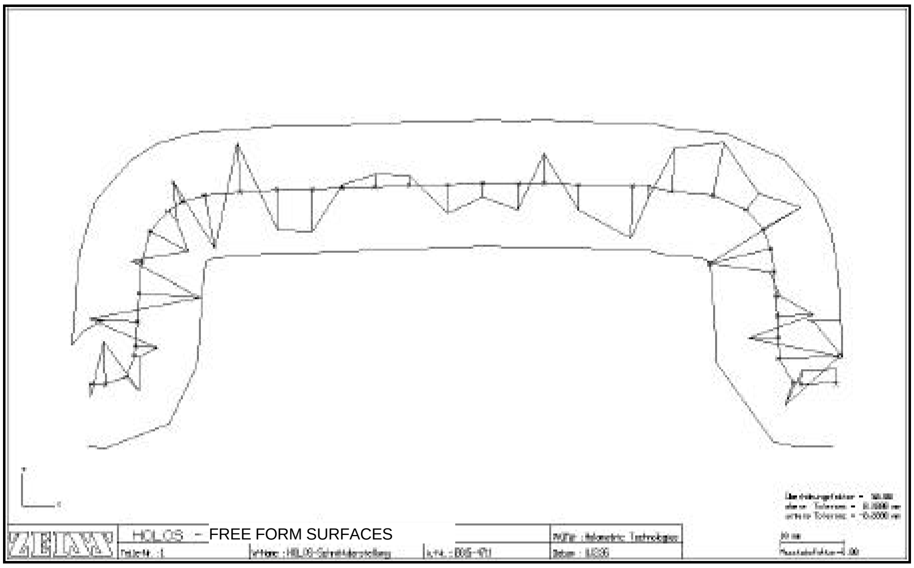


Figure 3-10

Paper formats

Scale output is possible on all paper formats from DIN A4 to DIN A0. The size of the output format is dependent on the output device used.

PaintJet XL300	
LaserJet	DIN A4 ... DIN A3
DesignJet	
Pen Plotter	DIN A4 ... DIN A0

The size of the set paper format is displayed in the width and height fields. When using DIN formats, these values cannot be changed.

If you wish to use another paper format, click on the <user-defined> function. You can then define the values for the height and width of the paper format used and the respective margin setting in the input fields provided.

With pen plotters or with DesignJet, the paper can be inserted horizontally or vertically. As the margin settings must be changed in this case, the position of the inserted sheet must be redefined.

Margin settings

When outputting to a plotter/graphic printer, it is not possible to print on the entire paper area. Therefore, margin settings must be made for the various output devices.

Standard

The preset margin areas are **6 mm for the horizontal margins and 18 mm for the vertical margins**. These values have been determined for various output devices and should therefore not be changed.

If, however, problems arise with the margin setting on your output device (incomplete frame for graphic representation), you can redefine the default values for the margin settings:

⇘ Generate a file called border_values in the directory /users/holos/sys.

⇘ First enter the value for the horizontal margins, then the value for the vertical margins.

↳ When the <Scale> function is next called up, the new set values will be activated.

Output

The following output options are supported by HOLOS-UX for scale representation as 2D output:

Output in the six possible projection planes

1. Representation of
   - Surfaces
   - Curves
   - Trimmed surfaces (faces)
   - Points
   - Scanning lines
2. Representation of deviations
   - With numeric output
   - As graphic icon
   - As marker
3. Representation of sections with
   - Nominal contour
   - Actual contour
   - Max/min values
   - Tolerance band
   - Deviations
4. Design of graphic measuring record
   - Entry of texts, lines, rectangles, circles
   - Modifying, moving, deleting texts, lines, rectangles, circles
   - Moving deviation icons
5. Representation of record header (graphic frame)
6. Moving displayed elements on the screen or output device. When defining the scale representation, all displayed elements are positioned on the screen or output device so that their 2D center of gravity is located in the center of the image.
7. Output of a raster grid

No objects can be selected in scale representation!

4 Data Management

This chapter describes the functions in the <File>menu. They are used principally for the management of files.

The main <File> function subdivides into the following functions:

- Display file
- Print file (NT)
- Delete file
- Rename file (NT)
- Store file (NT)
- Import/Export
- Check VDA file (UX)
- Exit (NT) / End (UX)

4.1 Basic information about data storage

Almost all data that are generated during a working session in HOLOS are saved automatically by the program. Typical of such data are, for example, measuring runs and individual objects.

> ***NOTE***
>
> There are a number of ways in which measured points can be generated manually. These will not be saved automatically, but you can save them afterwards by using the relevant function.

Automatic storage of objects

All objects generated, i.e. curves, patches, surfaces, are saved automatically.

The objects are given fixed names with consecutive numbering:

Curves	HCUR0001
Patches and surfaces	HSRF0001

These names are also entered into the generated VDA file.

> **NOTE**
>
> The numbering does not always reflect the sequence of generation, as the program looks for free spaces when numbering.

The names of the objects will be displayed in the graphic representation if you set the relevant parameters with the <Parameter-Graphics> function.

🕮 See Chap. 13.1

Automatic storage of measuring runs

Nominal and actual data are generated during the definition and execution of measuring runs. On principle, nominal data are called “name.nom“ in the NT version, and “name.mess“ in the UX version; actual data are called “name.act“(NT) and “name.ist“ (UX). The name of these files depends upon whether the data refers to one object or to a number of objects.

Reference to one object:	File bears the object name, e.g. HSRF0012_1.nom (or .mess)
Reference to several objects:	File bears the model name, e.g. holosmodell_0.act. (or.ist)

In addition to the name, the files continue to be numbered consecutively. The storage of data is notified in the status line.

If file names are not to be generated automatically, you can change the entry for the generation of file names accordingly in the <Parameters>-<System> menu.

You will then be requested to enter a file name each time data are saved.

4.2 Display files

The <Display file> function allows you to display the various files on the screen with the help of an editor.

Info file Display the model info file. The info file is generated if the model is converted from a VDA or IGES file into internal HOLOS format. It contains information about the header of the VDA or IGES file, comment lines, information about the number of converted elements contained in the file, as well as details of any errors that have occurred.

HOLOS warning (NT) All warnings output by HOLOS during the run time are saved in this file.

HOLOS error All errors output by HOLOS during the run time are saved in this file.

HOLOS info (NT) All information output by HOLOS during the run time is saved in this file.

Record file (NT) You can use this function to display the content of stored record files in an editor.

Graphics dump Graphics dumps can be saved as a bitmap file. This function allows you to display stored bitmap files.

Nominal data (NT) The <Display nominal data> function allows you to display information about stored nominal values. You can obtain information about the type of measuring run, the number of points, as well as a points list.

4.3 Print files

The <Print file> function allows you to output the content of different files to a connected printer.

Record file This function allows you to output the content of stored record files to a printer.

Graphics dump Graphics dumps can be saved as a bitmap file. You can use this function to print out stored bitmap files.

4.4 Delete files

You can delete various types of data using the <Delete file> function. After selecting the data type, the existing files will appear.

Record file You can delete the stored record files for a model using this function.

Actual data Measured values are saved as actual data after each measuring run. The actual data can be deleted using this function.
NT: The actual data of regular geometries can also be deleted.

Nominal data The “Delete nominal data” function is used to delete the stored nominal values for a model.

If you are deleting nominal data, the program also checks to see if actual data (= "measuring data") also exists. The program informs you about any files that are found, and queries whether these are also to be deleted.

NOTE

Because measuring runs will automatically be saved as nominal and actual data, you should update the data stock from time to time by making deletions.

View (NT) Views are stored files via which you have saved defined views of the graphic representation. You can delete these data using this function.

Unused objects In many cases you will obtain an “impure” CAD model for further processing in HOLOS, i.e. the surface model contains geometric objects that are not processed in HOLOS or cannot be processed. Because these objects, in part, have references to other objects, it is potentially difficult to remove such objects using the existing delete functions.
Unused objects are identified and removed from the system using the <Delete unused objects> function.

Model comparison (NT) When comparing models, various information is saved about the elements of a model. This function is used to delete the stored information. Information from a model comparison cannot subsequently be displayed.

Graphic elements (NT) This function is used to delete stored graphic elements (lines, rectangles, texts, bitmaps).

Graphics dump Graphics dumps that are saved as a bitmap file can be deleted from the hard disk using this function.

Sections (NT) You can delete the files for representing sectional deviations using the <Delete sections> function.

Scanning zone (NT) You can delete the stored files for digitization of scanning zones using this function.

CAD points (NT) Stored CAD points can be deleted from the hard disk using this function.
For UX, you will find the corresponding function in the "Data" menu under "Data > Eliminate CAD points"

Symbol positions (NT) The stored positions of the symbols for representing deviations are deleted.

Regular geometries (NT) Delete regular geometry elements.

4.5 Rename file

You can change the names of different files using these functions.

Record file You can rename stored record files using this function.

View Views are files in which defined graphic views are saved. Select this function in order to change the name of these data.

Graphic elements Here you can change the name of files which contain stored graphic elements (lines, rectangles, texts, bitmaps).

Graphics dump Select this function in order to change the names of stored bitmap files.

Sections Use this function to change the names of files which contain data for the representation of sectional deviations.

Scanning zone Scanning zones are files with points of areas for scanning. You change the name of these data here.

4.6 Savefiles (NT)

You can save data on the hard disk using the <Save file> function.

CAD points

CAD points are point data which are transferred from CAD files (VDAFS or IGES). These files can be imported into a model.

As normally only CAD points are managed within a model that originate from a CAD file from which the model was generated, you can save subsequently imported CAD points on the hard disk using this function.

4.7 Transfer model data (NT)

Using this function you can transfer data from another model into the current model.

Select the “Transfer“ function in the “File“ menu.

The model selection is displayed:

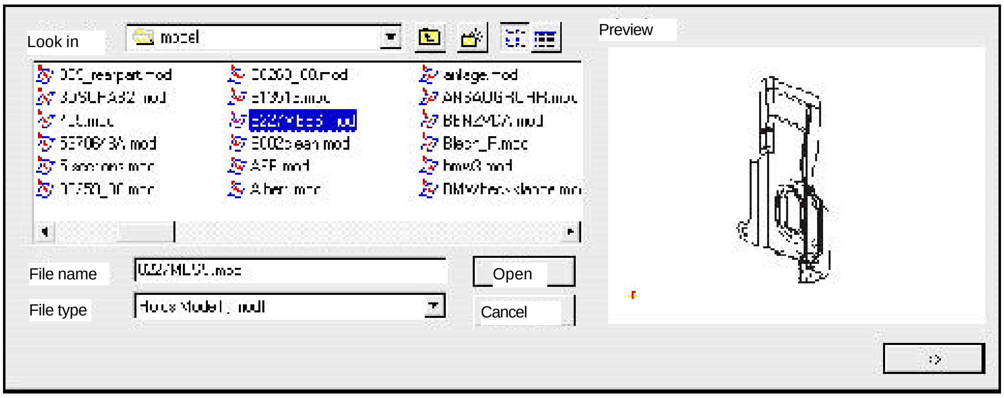


Figure 4-1

Select the model from which you wish to transfer data.

The selection window for data transfer is displayed:

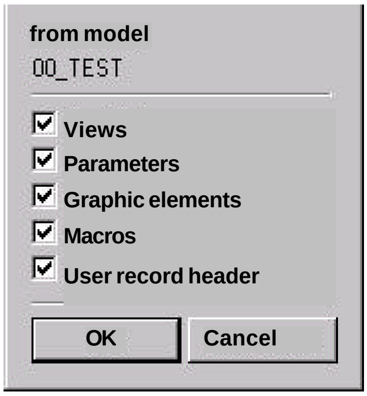


Figure 4-2

Select the data for transfer.

Views	Transfer all stored views of the model.
Parameters	Transfer all stored parameters for graphic representation, for measuring runs etc..
Graphic elements	Transfer all stored graphic elements.
Macros	Transfer all macros of the model. Please note that measuring runs which are executed in macros must have the same name as in the original model.
User record header	Transfer the settings of the user-defined record header.

4.8 Import / Export

Data can be imported or exported in HOLOS using the <Import/Export> function (NT). In the UX version the <import> and <export> functions are separate.

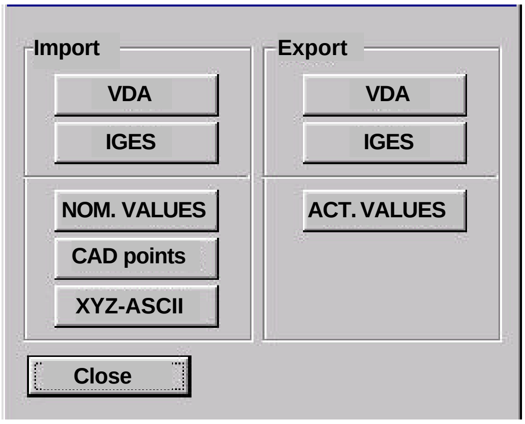


Figure 4-3

4.8.1 Import

VDA (NT) This function is used to start the VDA postprocessor, in order to import data from a VDA file into HOLOS.

IGES (NT) This function is used to start the IGES postprocessor, in order to import data from an IGES file into HOLOS.

Nominal values This function is used to import nominal values which have previously been exported from another model using the Export function.

CAD points You can use this function to import CAD points into HOLOS. CAD points are points that are contained in a VDA or IGES file. Points that originate from a workpiece file are read directly into the respective HOLOS model, but are also saved in a points file.

You can consequently import points from any file into an opened HOLOS model.

Click on <Import/export file>-<CAD points>

A window containing all available CAD point files opens.

Select a file and click on <Open>.

The points contained in the file are imported into the opened model.

UX peculiarities

For importing, the “Import > CAD points (local)“ function is used to import CAD points which have been transferred with the workpiece geometry into a VDA or IGES file.
The “Import > CAD points (global)“ function imports points which have been transferred with the VDA or IGES file of another workpiece.
In the dialog window individual points rather than point files are displayed for selection.

XYZ-ASCII (NT) This function allows you to import points from a point list in HOLOS. A point list is an ASCII file which contains values for the point coordinates x, y, z. The values can be separated by any information separator. The only prerequisite is that all three coordinates are in a line.

Click on <Import/export file>-<XYZ-ASCII>.

A window opens for selecting point lists.

Position the selection list on the directory in which the point lists are located, select a file and click on <Open>.

The points contained in the file are imported into the open model.

4.8.2 Export

VDA (NT) This function starts the VDA preprocessor in order to export data from HOLOS into a VDAFS file.

IGES (NT) This function starts the IGES preprocessor in order to export data from HOLOS into an IGES file.

> ***NOTE***
>
> Models that have been generated from a VDAFS file cannot be exported into an IGES file, and models that have been generated from an IGES file cannot be exported into a VDA file. To do this, different surface elements would require conversions that cannot currently be executed.

Nominal values This function allows you to export the nominal values of a model. You can then read the exported nominal values into another model using the <Import> function.

4.9 Check VDA file (UX)

The <check VDA file> function allows you to check VDA files for conformity with the VDAFS 2.0 interface format. However, in the present program version, only the line length of 80 characters is checked. The result is given in the status line.

A dialog window opens for selecting the VDA file.

📖 The function of the selection window is described in detail in Chap. 1.9.5.

4.10 Exit

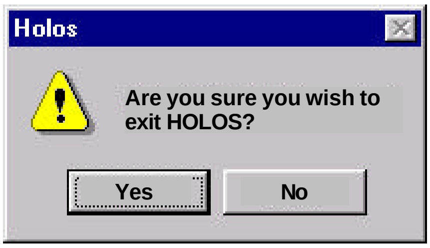


Figure 4-4

You can exit a current working session using this function.

If not all internally stored data have been saved to the hard disk during a working session (manually probed measuring points), you must answer the query as to whether you want to save the data or not, in a window which appears on the screen.

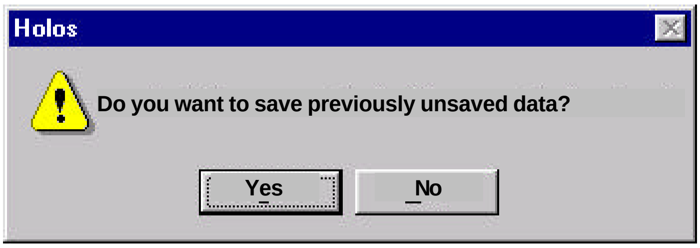


Figure 4-5

5 Model Management

This chapter describes the functions in the <Model> menu>. The functions are concerned with the management of models. A model is understood to mean the description of a workpiece. A model can consist of curves, surfaces, faces etc.

The main <Model> function is sub-divided into the following functions:

- New model (NT) / Generate model (UX)
- Open model (NT) / Load model (UX)
- Delete model
- Save model (NT)
- Close model
- Copy model
- Rename model
- Add model
- Compare model
- Information on model

The Delete model function is only available if no model is located in the HOLOS main memory.

The **HT-Navigator** is available in the NT version for simple model management. It is automatically installed and registered with the programs during installation.

5.1 New model (NT) / Generate model (UX)

The <New model> (NT) or <Generate model> (UX) function allows you to create a new model. Through this function, the system is initialized to accept a new model.

After calling up the function, an input window appears.

In order to set up the model, enter a model name and click on <New> (NT) or <OK> (UX).

HOLOS is now ready to accept data for the new model.

5.2 Open model (NT) / Load model (UX)

The <Open model> (NT) or <Load model> (UX) function allows you to load a model saved on the hard disk into the main memory and hence into the graphic window. All of the currently existing models will be displayed for selection of the model.

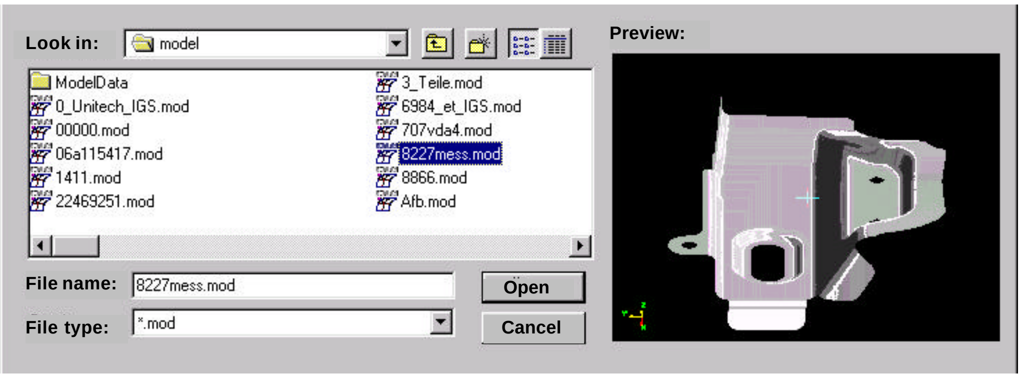


Figure 5-1

Select the relevant model by clicking on the model name with the left mouse button. Then click on <Open> (NT) or <OK> (UX).
If the model has already been loaded once and a preview has been generated, you will see a graphic representation of the model in the preview window.

The model will be loaded with the same graphic settings that it had when it was saved.

5.3 Delete model

With the <Delete model> function you delete a model from the hard disk. For selection of the model, all existing models are displayed in a window.

Select the relevant model by clicking on the model name with the left mouse button. Then click on <Delete>.

The selected model will not actually be deleted until after you have positively confirmed your action in a safety-check query put to you by the program.

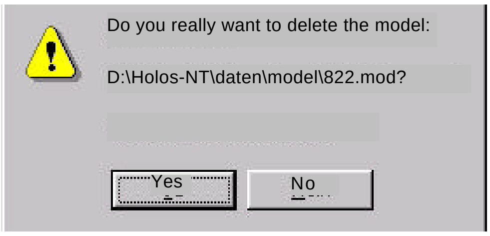


Figure 5-2

Click on <Yes> (NT) or <OK> (UX), if you wish to delete the model.

5.4 Save model and save model under (NT)

Using the <Save model> function you can save a loaded model when changes have been made to the structure of the model. Normally changes are saved automatically by HOLOS. However, the point in time at which changes are saved depends upon the setting in the system parameters.

If you wish to save data independently of this setting, you can do this by clicking on the <Save model> function.

With “Save as” you can save the model information as well as parameters and groups under a new name and/or another directory. Measuring jobs and regular geometries etc. will not be saved! – To do this use the “Copy” function!

5.5 *Close model*

With the <Close model> function, the currently active model will be removed from the main memory and thus also from the graphic window.

5.6 *Copy model*

With the <Copy model> function you copy the model that currently exists in the main memory and file it away on the hard disk under another name. To file away the copied model, you need to enter a name for the model.

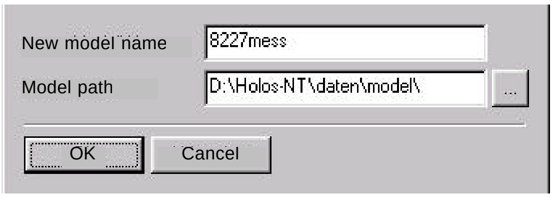


Figure 5-3

Enter the name of the model under which the copy of the model is to be saved.

Enter a directory name under which the copy of the model is to be saved (this is only necessary if the copy of the model is to be saved under another directory).

Click on <OK> in order to copy the model.

A copy of the model is saved under the entered name.

5.7 *Rename model*

With the <Rename model> function you assign a new name to an opened model.

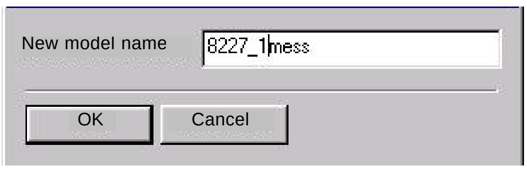


Figure 5-4

Enter the new name for the model.

Click on <OK> in order to accept the new name.

5.8 Add model

Using this function you can add a further model to an opened model. This allows you to combine different parts of a model into a whole model.

After selecting the function you will obtain the model selection list:

Select the relevant model by clicking on the model name with the left mouse button. Then click on <Open> (NT) or <OK> (UX).

If the model has already been loaded once and a preview has been generated, you will see a graphic representation of the model in the preview window.

The model is loaded and added to the already opened model.

5.9 Compare model

The <Compare model> function allows you to compare the modification status of different models that are saved for HOLOS.

If HOLOS is able to establish changes to the surface description, the relevant areas are marked in color.

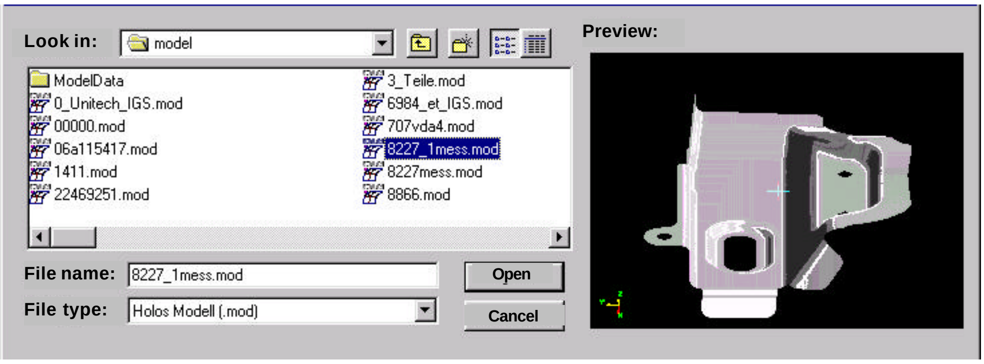


Figure 5-5

Select the relevant model by clicking on the model name with the left mouse button. Then click on <Open>(NT) or <OK> (UX).

The following query appears:

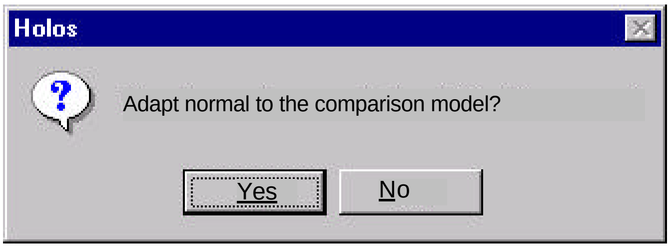


Figure 5-6

Click on <Yes>, if the orientation of the surfaces (the normal vectors of the surfaces) is to be adapted to the orientation of the surfaces of the comparison model.

or

click on <No>, if no adaptation is to occur.

HOLOS now performs the calculations required to compare the two models. The result is displayed on the screen with the surface descriptions in different colors.

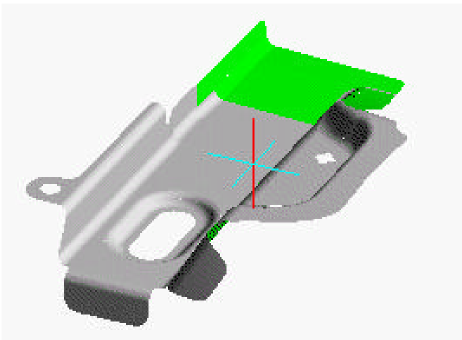


Figure 5-7, Modified surface descriptions

5.10 Archive model (UX)

With the <Archive model> function you can archive and save HOLOS models.

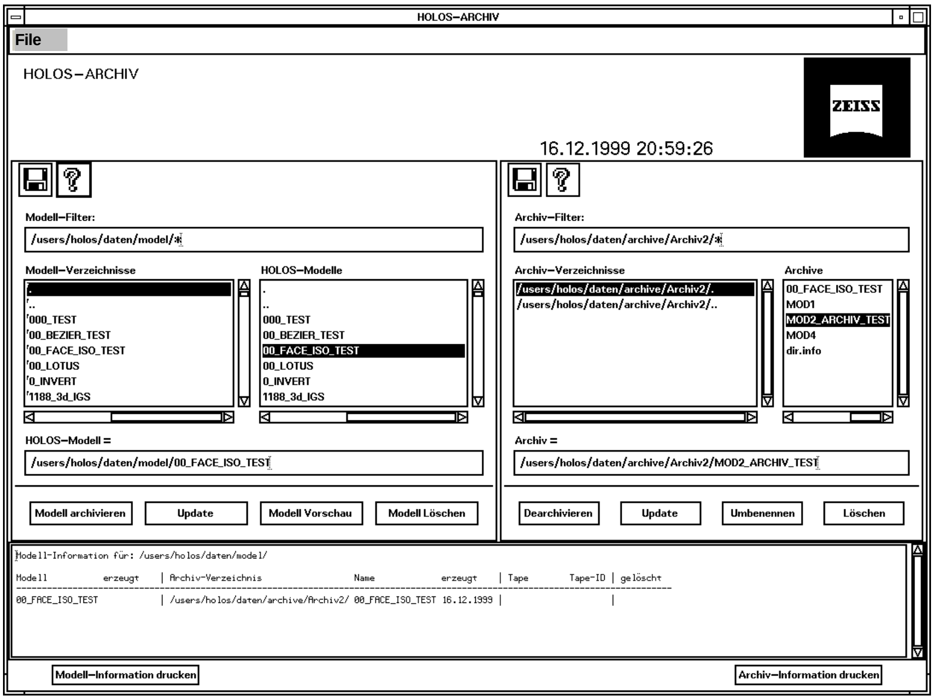


Figure 5-9, HOLOS ARCHIVE

The left column shows the currently set model directory. You can switch to other directories by clicking on the directory name.
The right window shows the currently set archive directory.

Archive directories can be located on any computer in the network, but the hard disk of the network computer must be connected to the HOLOS computer (mount command). Archive directories must be previously set up at the corresponding location. Contact your system administrator for further details.

5.10.1 Archive model

- Select a model and click on <Archive model>.

  With "Preview" you can view the model beforehand, if it has been generated with HOLOS via the "Admin" menu. An input window for an archive name is displayed.

- Enter a name or maintain the suggested name, which always corresponds with the selected model.

  Archive the model with <OK>. If an archive with the same name already exists, an error message will appear. Otherwise, a window to recheck all names and directories is displayed:

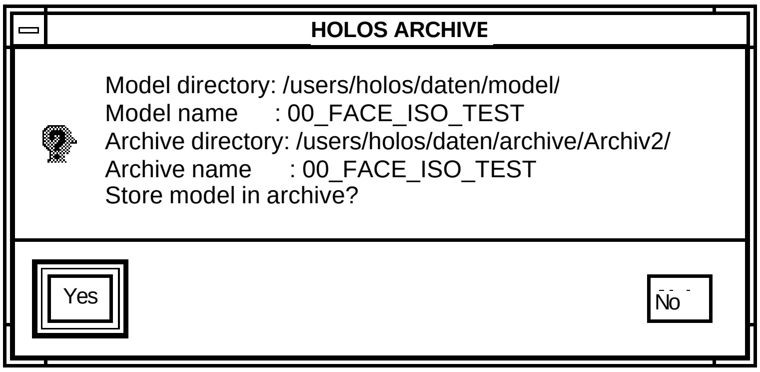


Figure 5-10, Store model

- If you click <Yes> the model will be stored in the set archive directory. Information about the transfer is stored in a directory-dependent file.

- Models that are no longer required are deleted with "Delete model".

  With "Update" you can update the list of models.

5.10.2 Retrieve model from archive

Select an archive and click on <Dearchive>.

An input window for a model name is displayed.

Enter a name or maintain the suggested name, which always corresponds to the selected archive.

Dearchive the model with <OK>.

If a model with the same name already exists, an error message appears. Otherwise, a window to recheck all names and directories is displayed:

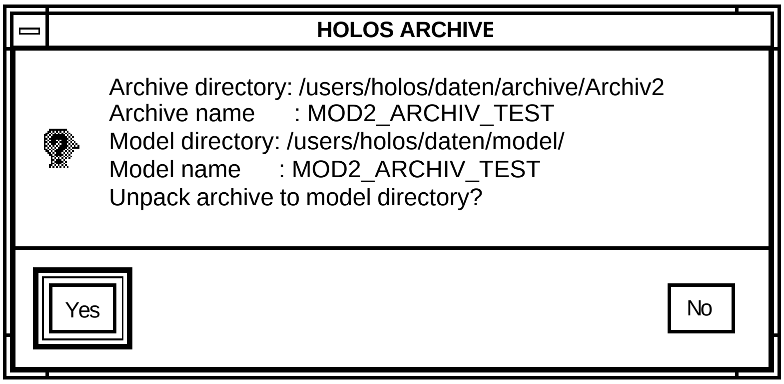


Figure 5-11, Unpack archive

If you click <Yes> the model is retrieved from the archive directory into the specified model directory and can be processed with HOLOS again. Information about the dearchiving is stored in a directory-dependent file.
Archives that are no longer required are deleted with "Delete".

"Rename" opens an input window, in which you can enter a new name for the currently selected archive.

"Update" updates the archive list.

5.10.3 Store models and archives on DAT tape

Select a model or archive and then click on the floppy disk symbol on the corresponding page of the archive window. A query will be made as to whether the model/archive is to be transferred.

Click on <Yes> to store the model/archive on DAT tape.

An input window for the DAT tape parameters is displayed:

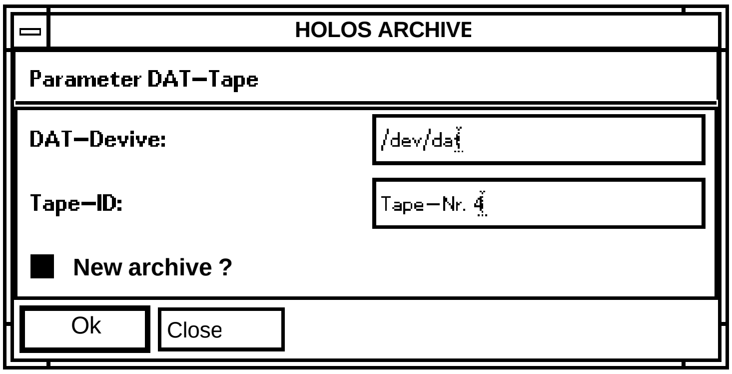


Figure 5-12, Unpack archive

DAT device — Enter the name of the device file under which the DAT drive is installed on your computer – generally, /dev/dat.

Tape ID — Enter an identification for the tape, e.g. a simple character string with which the tape is tagged.

New archive — This option creates a new archive on the DAT tape.

Caution:
All existing data on the DAT tape will be overwritten and will no longer be able to be restored.

If you wish to append your data to existing data on the DAT tape, this option must not be selected!

5.10.4 Information on model and archive directories

During the various actions, information is stored in a directory-dependent file.
Information about a model or an archive directory is obtained by clicking on the question mark on the corresponding page of the window.

The files contain the following information:

Model directory:

```
Model information for:   /users/holos/daten/model/

Model            created  | Archive directory                  Name            created    | Tape       Tape-ID | deleted
------------------------------------------------------------------------------------------------------------------------
00_FACE_ISO_TEST          | /users/holos/daten/archive/Archiv2/ 00_FACE_ISO_TEST 16.12.1999 |                    |
000_TEST                  | /users/holos/daten/archive/Archiv2/ 000_TEST         16.12.1999 |                    |
```

Model	Model name
Created	Date on which the model was created, if it has been generated from an archive
Archive directory	Name of the archive directory in which the model has been stored
Name	Archive name, under which the model has been stored
Created	Date on which the archive was created
Tape	Date of storage on DAT tape.
Tape ID	For storing registered ID of the DAT tape.
Deleted	Date on which the deleted model was deleted

Archive directory:

```
Archive information for: /users/holos/daten/archive/Archiv2/

Archive          created    | Model directory                 Name              created    | Tape       Tape-ID                     | deleted
-------------------------------------------------------------------------------------------------------------------------------------------
dir.info                    |                                                              | 03.11.1999  123                        |
MOD4             03.11.1999 | /users/holos/daten/ARCHIV_TEST/ MOD4              03.11.1999 | 15.11.1999  Archivdatei /tmp/TT_ARCHIV |
MOD2_ARCHIV_TEST 03.11.1999 | /users/holos/daten/ARCHIV_TEST/ MOD2                         | 03.11.1999  Auslagerungsdatei 2        |
MOD1             15.11.1999 | /users/holos/daten/ARCHIV_TEST/ MOD2                         |                                        |
00_FACE_ISO_TEST 16.12.1999 | /users/holos/daten/model/       00_FACE_ISO_TEST             |                                        |
```

Archive	Archive name
Created	Date on which the archive was created
Model directory	Name of the model directory into which a model has been dearchived
Name	Model name, under which the model has been dearchived
Created	Date on which the model was dearchived
Tape	Date of storage on DAT tape.
Tape ID	For storing registered ID of DAT tape.
Deleted	Date on which deleted archive was deleted

5.11 Model information

With the <Model information> function you can obtain various information about the model loaded in the main memory, e.g. the number of different objects and whether or not they are masked.

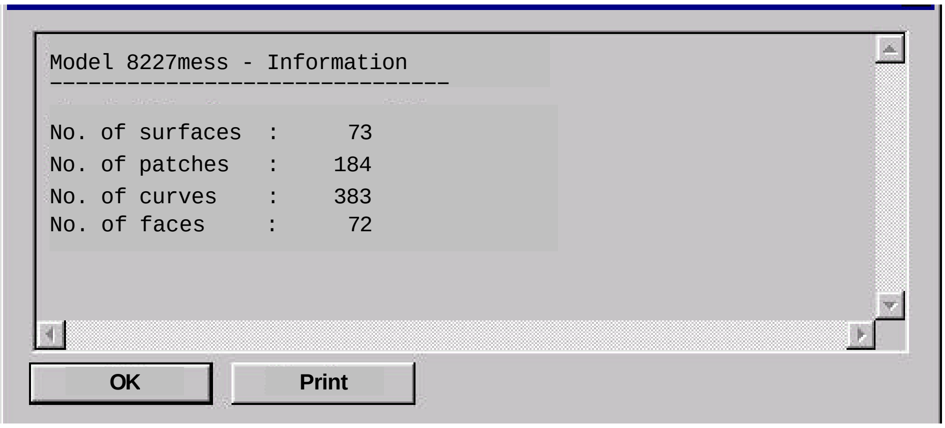


Figure 5-8

6 Manual Measurement

This chapter describes the manual measurement functions that are available with HOLOS.

HOLOS contains functions for various manual measuring procedures:

- Measurement of shape deviations on surfaces (patch identification)
- Edge measurement (trimming measurement)
- Alignment
- Digitization (the functions for digitization are only contained in the Digitization option)
- By means of the <Manual> function you can manage and further process measured points obtained by manual probing.

The respective menu entries can only be activated if the relevant point information is available after manual probing.

6.1 Patch identification

The main <Patch ident> function is sub-divided into the following functions:

- Save probing points
- Delete probing points
- Display probing points
- Display deviation window
- 3D best fit
- Measurement record
- Chromatic coordinates
- Display values large

6.1.1 Manual probing

For a known model, it is possible to manually probe the workpiece at any time, and without the use of any special function.

A manual probing triggers a number of actions within the program:

- A search is made in a defined search area around the probing point for a patch of the model on which the probing point is located ("Patch identification"). The search area is set using the <Patch ident> parameter function.
- If a patch is found, the probing point will be graphically represented in the form of a probe. Its deviation from the nominal point will be displayed immediately. Adjustments are made to how the deviation is represented via the <Graphics> parameter function.

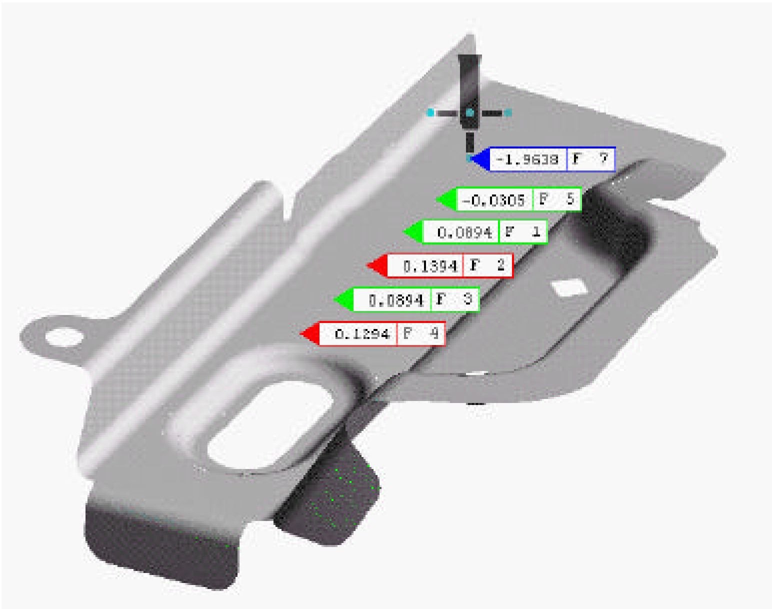


Figure 6-1

- In addition to the graphic representation of the deviation, a window will appear, displaying the coordinates of the point and its deviation relative to the currently defined tolerance range (see Fig. 6-2).

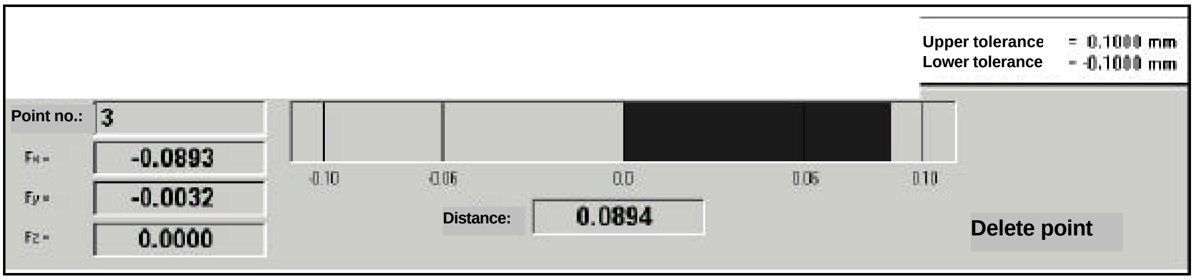


Figure 6-2

6.1.2 Save probing points

The <Save points> (NT) or <Save probing points> (UX) function saves on the hard disk all the points in the buffer which have been manually probed up until now. Two files are created in this process:

- Actual values file: contains the probing points
- Nominal values file: contains the surface points found as a result of a perpendicular projection of the probing points on the surface.

The file names will be assigned automatically or can be defined by you, if you have activated the relevant file storage option in the system parameters.
The selected points stored can be called up for analysis or for a CNC measuring run.
If after saving you probe further points, these will begin again with point no. 1 and will, when saved, be put into a new file.

6.1.3 Delete probing points

The <Delete points> (NT) or <Delete probing points> (UX) function deletes all probing points that are currently held in the buffer. Probing points that are saved in files will not be deleted.

6.1.4 Display points (NT)

The <Display points> function allows you to re-activate the display of deviations for manually probed points if you have switched this off, e.g. by pressing <Clear> in the graphic menu bar.

6.1.5 Display deviation window

To display the deviation distribution, you can open a window in which you can manage the manually probed points. You can view the values of individual points from the point list and delete individual points from the list if required.

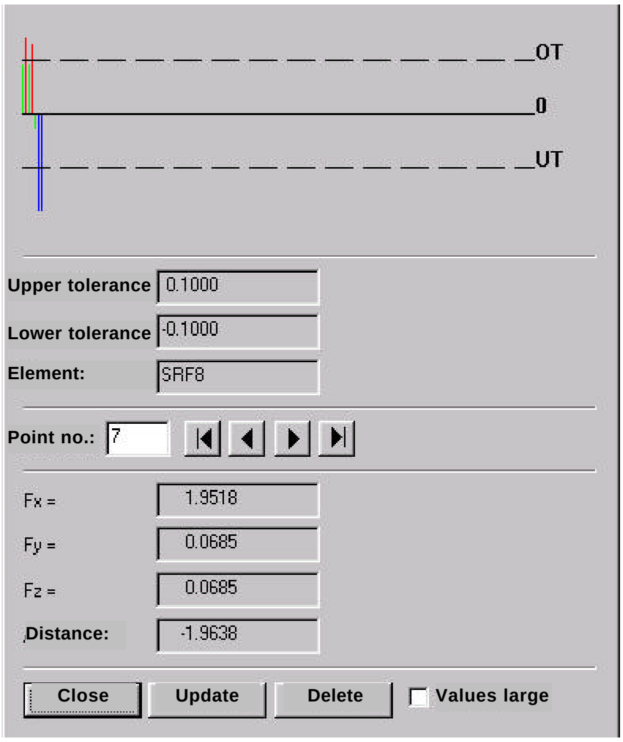


Figure 6-3

Function	Meaning
Point no.	Jump to a point by entering the point number
|< < and > >|	Go through the points step-by-step
Update	The graphic display is refreshed
Delete point	The current point is deleted

6.1.6 3D best fit

The <3D best fit> function carries out a best fit of the workpiece using the manual probing points that are currently held in the buffer.

📖 A precise description of this function can be found in Chap 9.4.

6.1.7 Display measuring record

The <Measuring record> function allows you to create a measuring record that contains all manually probed points and their deviations. The output device for the measuring record depends on the setting in the parameter function <Measuring record>.

🕮 See Chap. 13.10

The structure of the measuring record is identical to that in the <Evaluation> menu and dependent on the currently set measuring record type.

🕮 See Chap. 9.5

6.1.8 Chromatic coordinates

This function allows you to activate the display of deviations in chromatic coordinates, whereby all manually probed points that are currently located in the HOLOS main memory are taken into account in the deviation display.

Further information on displaying deviations with chromatic coordinates can be found in the <Evaluation> chapter.

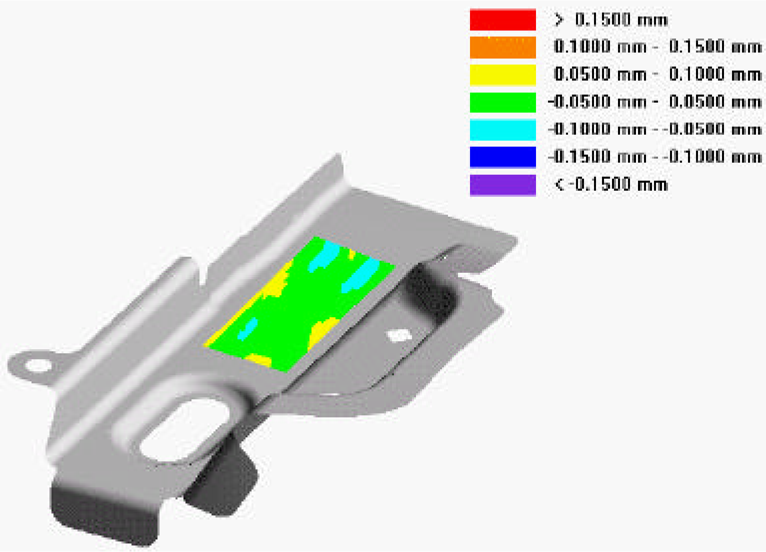


Figure 6-4, Display of deviations with chromatic coordinates

6.1.9 Display values large

If you have to work further away from your screen, especially in the case of larger coordinate measuring machines, it is relatively difficult to identify the measuring results on the screen.

By using the <Values large> switch, you can enlarge the function display of the measurement results output.
The <Values large> switch in the deviation distribution window has the same effect.

You can select whether actual or nominal values or the deviations are to be displayed in the individual axes. The absolute difference “D“ is always output at the same time.

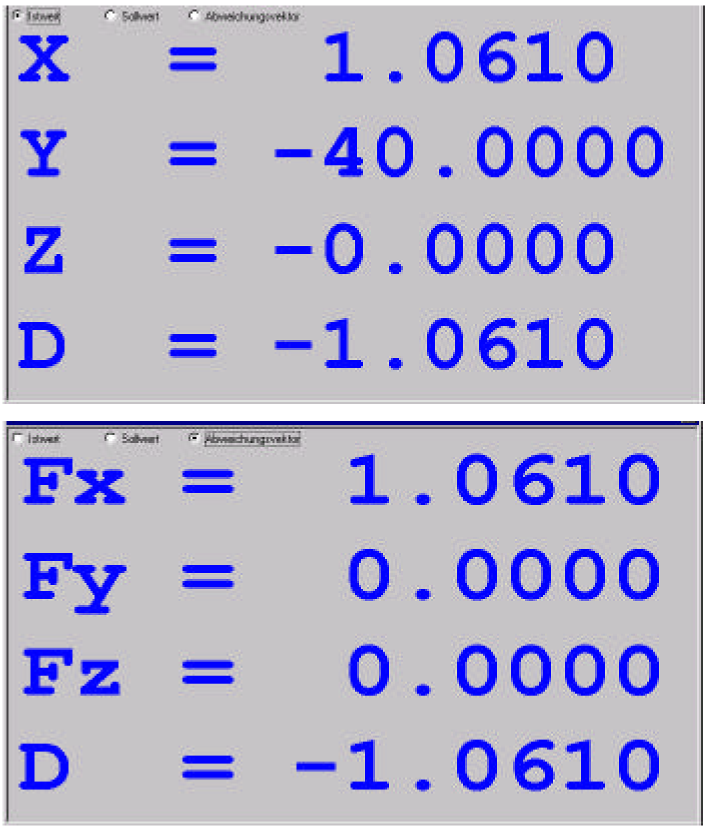


Figure 6-5

6.2 Edge measurement

The edge measurement function is used for the measurement of deviations in the edges of a workpiece.

> ***NOTE***
>
> It is advisable to select the vectorial form of representation since in this form, the directions will also be displayed.

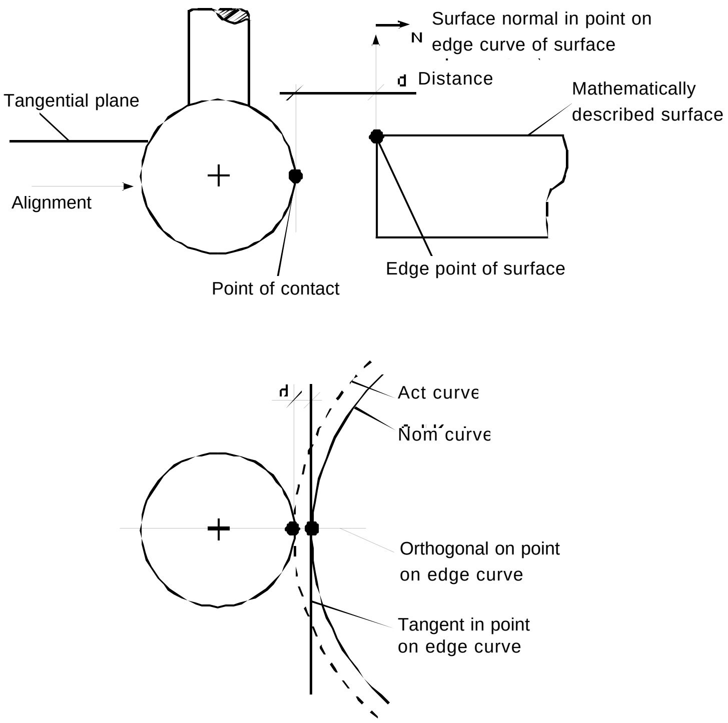


Figure 6-6

Measuring sequence

Prerequisite

Before manually probing edge points, you must activate the edge measurement mode in the UMESS option, CADLINK (MAN-MES button).

If you use HOLOS functions for aligning the workpiece (assignment of probing points), the corresponding axis selection must be defined in UMESS if the coordinate axes of the workpiece coordinate system do not agree with the axes of the machine coordinate system.
If the position of the coordinate axes is not correctly defined, then HOLOS will have an incorrect probing direction during manual probing and will therefore not be able to correctly define the workpiece edge.

1. Probing on the edge.
2. Determination of the point on the surface's edge curve.
3. Projection of the probe midpoint in the tangential plane of the point on the edge curve.
4. Probe radius correction in the direction of the orthogonal of the point on the edge curve.
5. Determination of the distance between the point of contact projected in the tangential plane and the point on the edge curve.

The deviations are displayed in a similar manner to the description of the <Patch identification>function.

Functions and operation

All functions and forms of representation are identical to the <Patch ident> function.

If you wish to directly allocate a measuring point to a geometric element (SURF or FACE), you must expressly select this element. No other elements are taken into consideration in the evaluation.

If you have not selected an element, HOLOS itself identifies the nearest element and calculates the deviation for it.

6.3 Alignment

With the <Alignment> function you can use manually probed alignment points to align your workpiece.

A precondition for this is that the manual measurement has been reset to <Alignment points> in the UMESS option CADLINK. You set this mode via the <MAN MES>function in the CADLINK menu bar.

With other measuring software packages, you must define the type of the manually probed points in HOLOS. Information regarding this is contained in the next chapter.

When you probe alignment points on the workpiece, these are graphically displayed on screen in HOLOS with numbering in the order of their probing. You can switch the graphical display of the alignment points on or off in the display parameter page in the left-hand graphics menu bar.

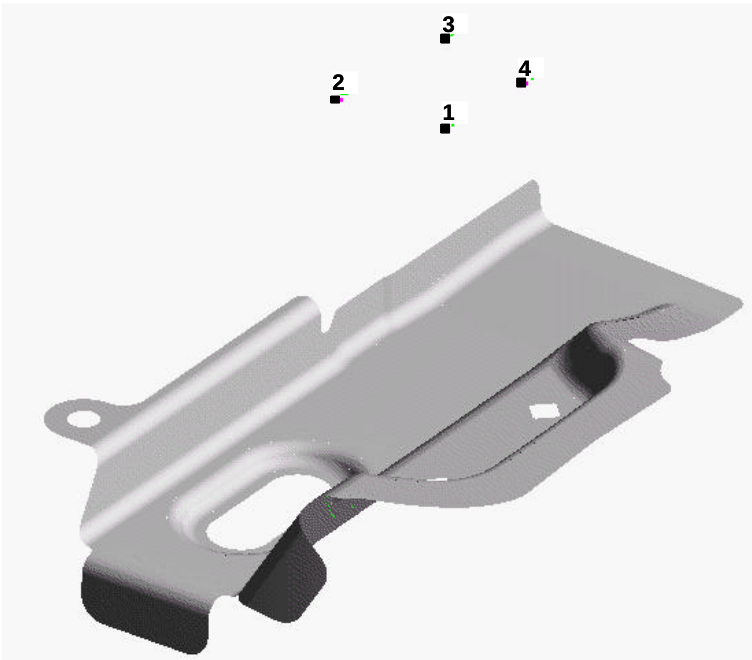


Figure 6-7

Operation

⇘ Clamp the workpiece to the coordinate measuring device base plate so that the direction of its axes roughly corresponds to the direction of the axes of the coordinate measuring device.

If this is not possible, you must define an appropriate axis selection in UMESS-UX.

⇘ Probe points on the workpiece whose positions you can identify on the workpiece.

Using the functions described below, you can assign these points to defined positions on the workpiece, and determine the alignment of the workpiece on the coordinate measuring machine.
After calculating the alignment, the transformation specification can be transferred to the relevant measuring software. The manually probed points are transformed into this new coordinate system.
The newly defined coordinate system is now defined in the measuring software. Subsequent probings produce points in the new coordinate system.
In order to permanently preserve the new workpiece coordinate system, you must save it using the relevant functions available in the measuring software.

The main <Alignment> function is divided into the following functions:

- Assign alignment points
- Correction
- Delete assignment
- Delete alignment points
- Calculate alignment

6.3.1 Assign alignment points

The <Assign Alignment points> function allows you to assign manually probed alignment points to a position on the workpiece. Therefore, probe the positions on the workpiece surface that you can easily identify in the graphic display in HOLOS.

Click on the <Assign alignment points> function.

The message ”Select point” appears in the status line. Go into the mode for selecting an alignment point.

Select an alignment point.

The selected point is color-marked. The message “Assign surface” appears in the status line, and you go into the mode for defining the position on a surface of the workpiece.

Click with the left mouse button on the appropriate position on the workpiece surface.

The assignment to a position on the workpiece is graphically displayed by a connecting line between the selected alignment point and a point on the workpiece.

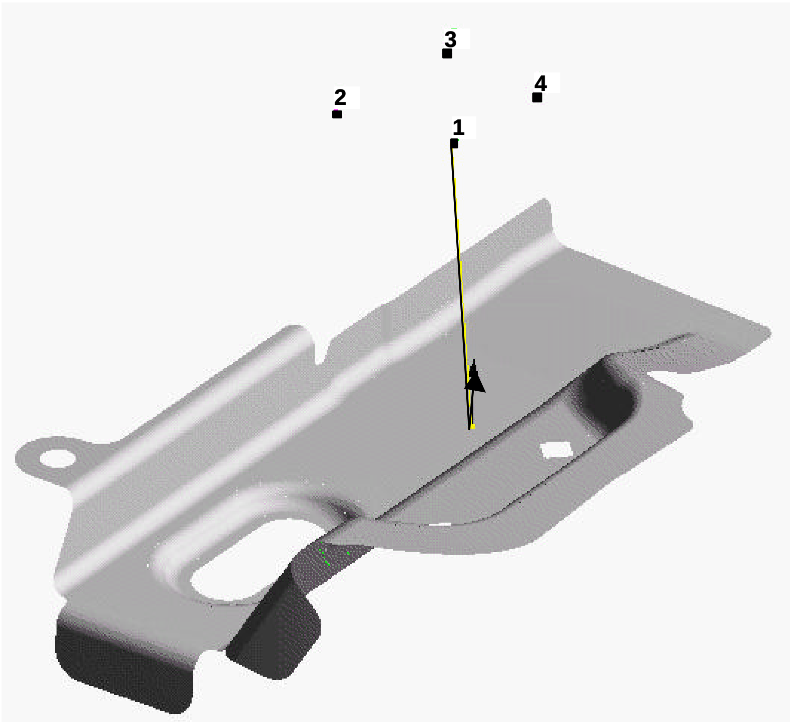


Figure 6-8

The display of the normal direction on the selected surface is also shown.

If the surface orientation is incorrectly defined, you must rotate the orientation so that the probe radius correction can be correctly calculated.

After assignment to a position on the workpiece surface, return to the mode for selecting an alignment point and proceed with the next assignment.

6.3.2 Correction

The <Correction> function deletes the most recently defined assignment of an alignment point to a position on the workpiece surface.

6.3.3 Delete assignments

The <Delete assignments> function deletes all assignments of alignment points.

6.3.4 Delete alignment points

The <Delete alignment points> function deletes all manually probed alignment points.

> **NOTE**
>
> With this function the alignment points are removed from the system and cannot be restored.

If you only want to remove alignment points from the graphic display, you can do this via the graphic display parameter page.

See Chap. 13.1, "Graphic display parameters"

6.3.5 Calculate alignment

After you have defined the necessary assignments, you can calculate the transformation specification for a new workpiece coordinate system.

The definition of the degrees of freedom for the best fit of the coordinate system is carried out via the 3D best fit parameters.

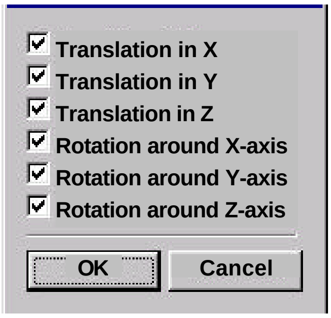


Figure 6-9

For a complete best fit, in which all degrees of freedom (i.e. translational and rotational) are to be taken into account, you must define at least three alignment points.

However, a minimum of four points, not located on a plane or in a straight line, is recommended.

If you carry out the calculation with less than three points, only the translational portions are taken into account for the calculation.

A message is displayed on screen, that the rotational portions have not been taken into account.

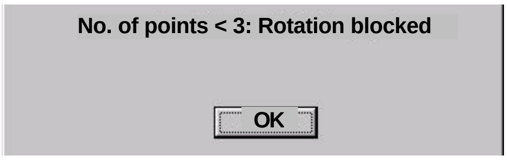


Figure 6-10

After the calculation, a window appears in which the results of the calculation are displayed.

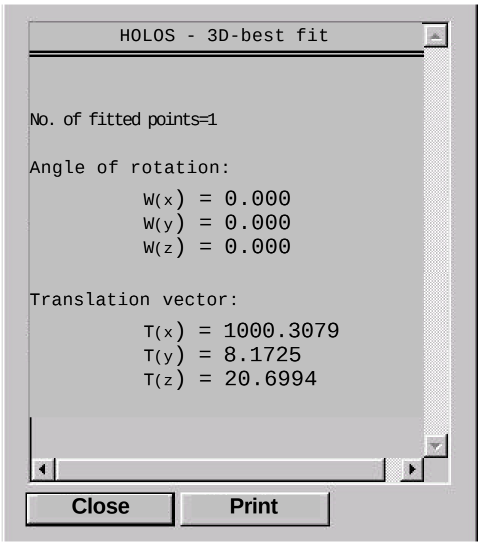


Figure 6-11

To transfer the transformation specification and the definition of the new workpiece coordinate system into the measuring software, you must answer <YES> to the query about the transfer of the transformation matrix.

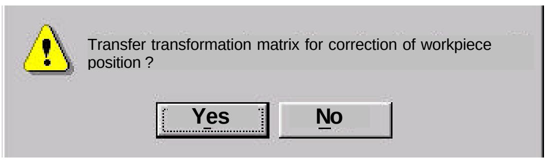


Figure 6-12

> ***NOTE***
>
> Ensure that the connection to the measuring software is activated, as the alignment points are transformed into the new workpiece coordinate system immediately after transfer of the transformation matrix.

6.4 Manual point input (NT)

When working with UMESS-UX measuring software, you can directly set which type of measurement is to be performed, in the CADLINK option. In this case, no further actions are necessary.

When working with other measuring software packages (Calypso, SCANMAX or UMESS 300), you must define in which mode HOLOS is to process manually probed points.

Patch identification In this mode, manually probed points are treated as surface points and evaluated accordingly.

Edge measurement In this mode, points for trimming measurement are processed.

Alignment Manually probed points are adopted as alignment points and can be assigned to the surface description in order to align the workpiece.

Digitization point In this mode, HOLOS treats the relevant points as digitization points which can be further processed for the generation of curves and surfaces.
The digitization functions are only available if the Digitization option is installed on your system.

Intermediate pos. If you are using Calypso or U-Mess-Unix, the current probe position can be displayed during travel.

6.5 Continue manual run (NT)

Manual runs for surface or edge measurement can be continued in a later HOLOS session.

Normally, manual runs are re-defined if the points of a manual measurement have been saved as a measuring run via the <Save points> function.

If, however, further points are added to a measuring run, a saved manual run can be reloaded into the main memory.

6.5.1 Patch identification

Click on the <Patch identification> function, if you wish to continue a manual run for surface measurement.

You will obtain a list of all manually probed measuring runs.

Click on the relevant run and then on <Open>.

The points are loaded into the main memory and can be further processed as described in the manual measurement function.

6.5.2 Edge measurement

Click on the <Edge measurement> function, if you wish to continue a manual run for edge measurement.

You will obtain a list of all manually probed measuring runs.

Click on the relevant run and then on <Open>.

The points are loaded into the main memory and can be further processed as described in the manual measurement function.

7 Measurement of Free Form Surfaces

This chapter describes the functions of the <Measurement> menu. The functions are used in CNC operation for the measurement and inspection of known freeform surfaces (i.e. surfaces for which a description is already available).

The measurement of freeform surfaces comprises the following operating steps:

- interactive definition of measuring runs on the screen
- execution of measuring run in CNC operation

The main <Measurement> function is sub-divided into the following functions:

- Define measuring run
- Start measuring run
- Display measuring run (NT) / Display nominal values (UX)
- Cancel measuring run
- Simulate measuring run
- Edit nominal values (NT)

Prerequisite

The functions of the <Measurement> menu are **only** suitable for CNC operation and only available if the CNC version of HOLOS has been installed on your system!

A surface description must already exist, i.e.
- either an imported VDA file
- or surfaces generated in HOLOS-NT

7.1 Define measuring run

Using the subfunctions of the <Define measuring run> function, you can generate various types of measuring runs interactively on the screen.

- Grid
- Curve (NT)
- Line
- Plane/workpiece section
- Raster
- 0.5 points
- Parallel curve (UX)
- Regular geometry
- Measuring points
- Edge points
- CAD point (UX)
- CAD points → Measuring points (NT)
- CAD points → Edge points
- Contour
- Corner point
- Net point/Net section
- Scan line (in UX only with UMESS-UX)
- Scan zone (in UX only with UMESS-UX)
- Start last measuring run
- Scanning according to nominal value (in UX only with UMESS-UX)

The nominal points of a measuring run are displayed as arrows in the normal direction to the surface. Arrows are shown in the color of the selected probe.

NOTE

With imported VDA data it can happen that the surface normals point into the workpiece. If you define measuring runs on these surfaces, the probing direction will lead to a collision.
Therefore, check the orientation of the surfaces beforehand and if necessary invert them (with the <Objects - Analysis> and <Objects - Rotate orientation> functions).

7.1.1 Grid

You can generate a points grid either on a single object (surface, face, patch) or on a group of objects. Before the generation of the points grid you must, in any case, select the objects either individually or combined together in a group.

The points grid is then generated for each surface and saved as a separate measuring run. The active probe with which the grid is to be measured is also saved at the same time.

The points grid is regularly distributed in the U and V directions over each surface (boundary included).

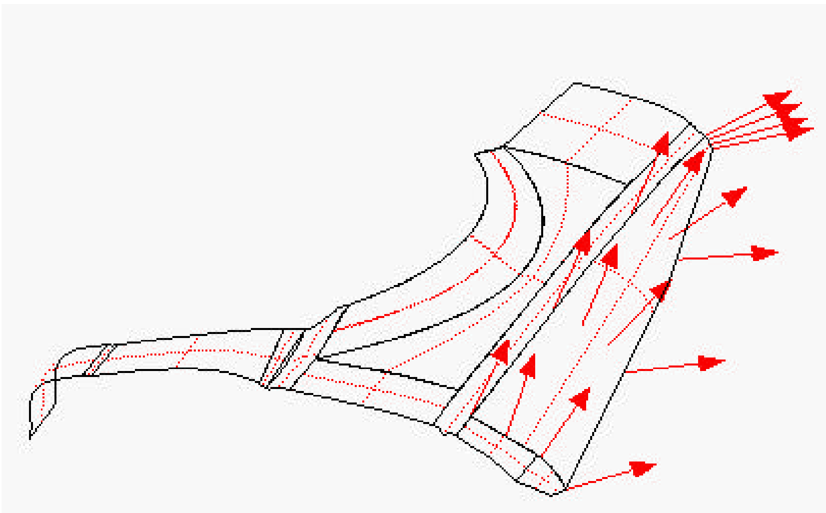


Figure 7-1

You can obtain a display of the U and V direction with the <Objects - Analysis> function.

NOTE

If a grid is generated on a surface that is bounded by a FACE element, only those measured points that lie within the FACE will be considered.

Measurement of free form surfaces

Purpose of the function

With this function you can check a surface up to 100%.

A regular grid is an ideal basis, although not a precondition, for a chromatic evaluation of the surface.

Using the actual values of a grid you can calculate a new surface using the <Grid -> Surface> function.
(Digitization option).

Operation

First of all, combine into a group all the objects that are to be measured -> the objects are marked in color.

Click on the <Grid> function.

A window opens for entering grid points.

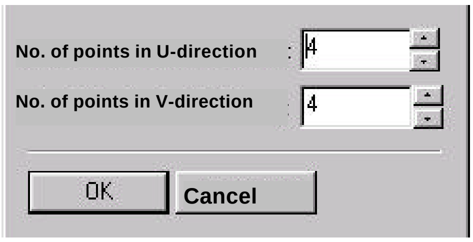


Figure 7-2

Click on the arrows (up or down) next to the input field (NT). Or click on one of the numbers in the number field (UX).

The number will be transferred to the input field.

Or:

Enter the number directly into the input field.

Confirm the entry with <OK>.

The defined grids will be displayed on the screen in the color of the active probe. The status line gives the names of the measuring runs.

7.1.2 Curve (NT)

Using this function, you can define measuring points on curves. However, the prerequisite for this is that the points on the curve can be projected onto a surface.

Select a curve on which measuring points are to be defined.

A window opens for entering the curve points.

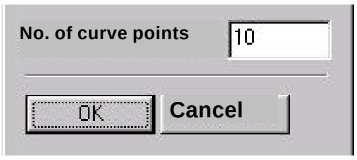


Figure 7-3

Enter the number of points in the input field.

Confirm the entry with <OK>.

Provided the points could be projected onto a surface, the defined curve points are displayed on the screen in the color of the active probe.

The measuring run is saved.

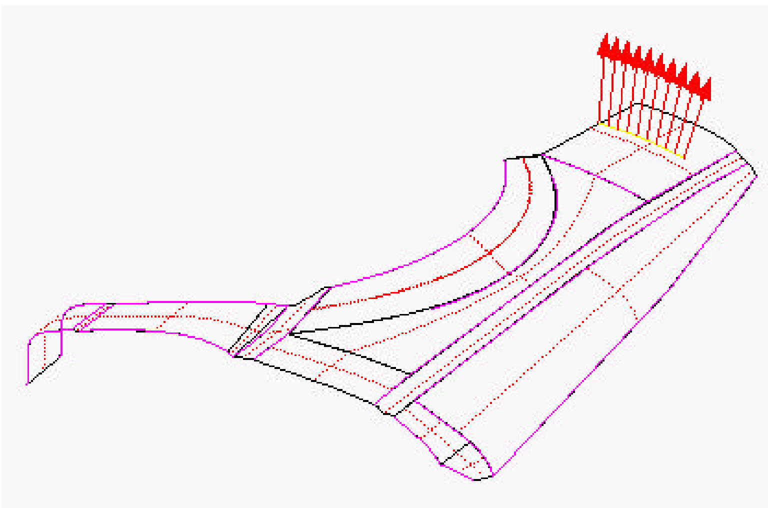


Figure 7-4

7.1.3 Line

In the definition of a line, the workpiece is intersected by a plane that results from the projection of the line on the workpiece. Measured points are only placed on those objects that the observer sees as lying “in front”. For highly-curved surfaces there is, under certain circumstances, no clearly-defined point of intersection with the surface. This results in measured points on surfaces that are located further “behind”.

If measured points are required to lie only on specific objects, combine these objects together into a group. The line will then be projected solely onto this group. Measured points that lie outside the group will not be considered. Gaps will develop on the screen.

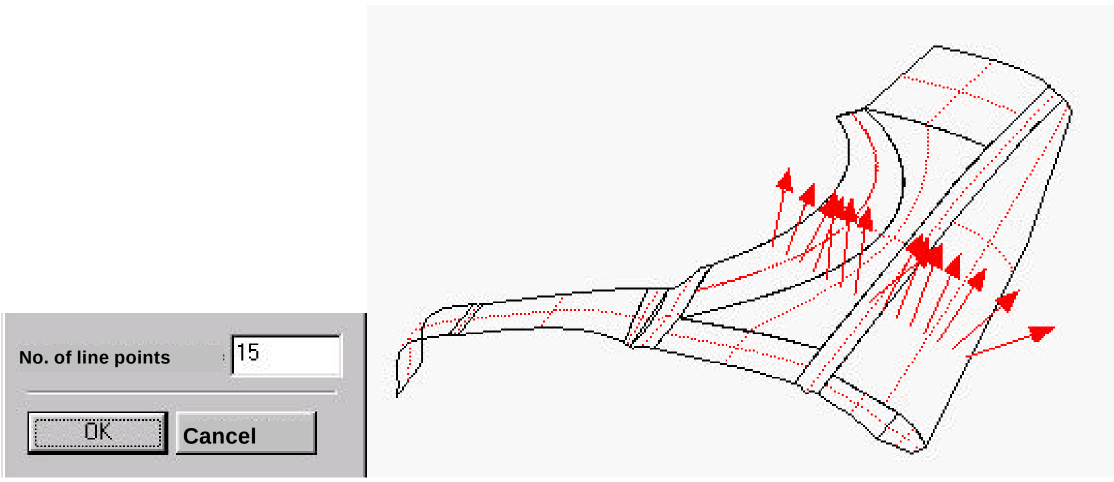


Figure 7-5, Points generation with a constant distance on a line

One measuring run will be generated per line. It will be displayed on the screen with arrows in the colors of the currently active probe. The measuring run will be saved automatically.

Operation

Click on the <Line> - <Specify number of points> function.

A window opens for entering the line points.

Enter the number directly into the entry field and confirm your entry with <OK>.

Position the mouse on the starting point.

Keeping the left mouse button pressed down, move the mouse onto the target point.

Release the mouse button.

The line is drawn. The measured points will be displayed as arrows in the color of the currently active probe.

The measuring run is saved.

Curvature-dependent points generation on a line

Operation

Click on the <Line> - <Curvature dependent> function.

A window opens for entering further parameters.

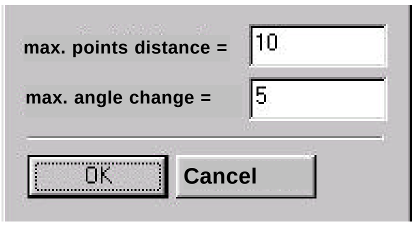


Figure 7-6

max. points distance

This parameter controls the maximum distance between the generated measured points, for cases where the criterion for curvature- dependent generation is not satisfied.

max. angle change

This parameter defines the value for the maximum angle change after which a measured point is to be generated.

Accept the parameters by clicking on <OK>.

Position the mouse on the starting point.

Keeping the left mouse button pressed down, move the mouse onto the target point.

Release the mouse button.

The line is drawn. The measured points will be displayed as arrows in the color of the currently active probe.

The measuring run is saved.

7.1.4 Plane / workpiece intersection

With the <Plane/workpiece intersection> function you can generate measured points on a plane which cuts the workpiece in a defined area.

The intersections can be generated with a defined number or with a curvature-dependent number of measured points.

Definition of a constant number of points:

Click on the <Plane/workpiece intersection> - <Specify number of points>.

A window opens for entering further parameters.

Enter the required number of points and confirm with <OK>.

Definition of a curvature-dependent number of points:

Click on the <Plane/workpiece intersection > - <curvature-dependent > function.

A window opens for entering the maximum points distance and the maximum angle change.

max. points distance: This parameter controls the maximum distance between the generated measured points, for cases where the criterion for curvature-dependent generation is not satisfied.

max. angle change: This parameter defines the value for the maximum angle change after which a measured point is to be generated.

UX: Nominal values as scanning line: With this option you can save nominal values as a scanning line.

Accept the parameters by clicking on <OK>.

Define intersection plane

When the parameters for generating the measured points have been defined, the intersection plane is established.

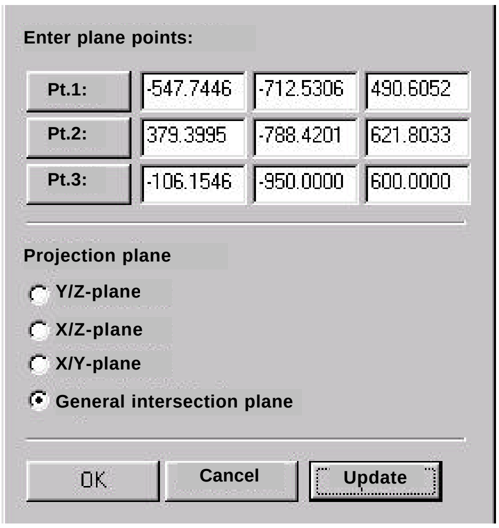


Figure 7-7

The intersection plane is determined by two plane vectors:

The first plane vector results from the direction of a preselected start point to a preselected end point on the plane intersection.

The second plane vector results either from the direction vector on the projection plane, in which both points lie, or from a specified auxiliary point, if a general plane is defined.

If the intersection is defined in one of the three possible projection displays, the direction vector on the respectively set projection plane is accepted, if you are not working with a general plane.

If the intersection is defined in the perspective display, then the plane into which the two points are to be projected must be preselected.

First of all, define the projection plane for calculating the intersection or press the < General intersection plane > switch , if you wish to define the second plane vector via an auxiliary point.

Graphically interactive definition of plane points

In order to define the plane points in a graphically interactive manner, you can either click with the left mouse button on points on the workpiece surface or existing points which are graphically displayed (CAD points, DIGIT points or points on scanning lines).

Click on <Pt.1:>, in order to determine the start point for the intersection plane.

Then click on an existing point or a point on the workpiece surface.

Click on <Pt. 2:>, in order to determine the end point for the intersection plane.

Then click on an existing point or a point on the workpiece surface.

Click on <Pt. 3:>, in order to determine an auxiliary point for the plane definition , if you are defining a general plane.

Then click on an existing point or a point on the workpiece surface.

The start and end point, as well as the auxiliary point used to define the plane, are displayed on the screen. The coordinates of the selected points are displayed in the respective fields and can subsequently be changed manually (see next section).

Click on <OK> to start calculation of the plane points.

The points on the intersection plane are calculated.

Manual entry

The start, end and auxiliary points can also be specified by manual entry, if the relevant coordinates are known.

The points to be entered (start and end point) must lie on the workpiece surface. You can define the maximum permissible distance of the points from the workpiece surface in the parameter page <System parameters> as the “maximum distance during data import”.

Enter the coordinates of the points in the fields provided:
Pt. 1: Start point
Pt. 2: End point
Pt. 3: Auxiliary point

When you have entered the coordinates, click on <Update>.

The points are adopted and displayed on the screen.

Click on <OK> to start calculation of the plane points

The points on the intersection plane are calculated.

The calculated measured points are graphically displayed on the screen and saved in a file as a measuring run.

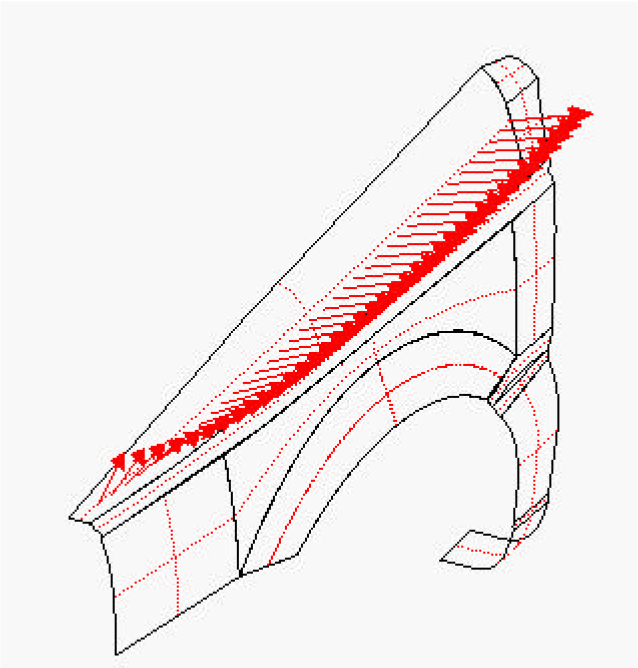


Figure 7-8

Correct surface orientation

The orientation of measured points on incorrectly oriented surfaces can be corrected in the same manner as when generating measured points on a line (system parameters, correct surface orientation).

The prerequisite for proper correction of the surface orientation is that the orientation of the surface on which the start point of the plane is defined (first measured point), is correct.

Plane/workpiece intersection with segmentation

The function for generating an intersection with segmentation generates an intersection for a special type of evaluation.

In this case, segments are formed on the calculated intersection and are divided into straight and curved pieces.

In the input window enter a value for the maximum curvature change after which a new segment is to be generated.

Then click on <OK> to define an intersection.

Define the intersection as described above.

The intersection is calculated.

After measuring, segmented intersections are evaluated explicitly as intersections. (See Chapter 9.2 Evaluation of intersections)

Only the points at the center of the respective segment are evaluated and displayed.

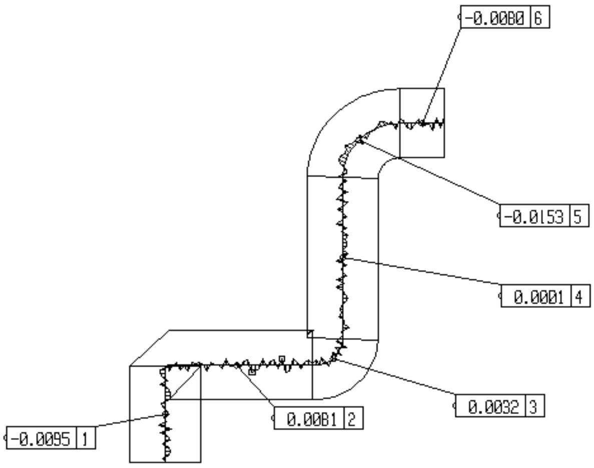


Figure 7-9

7.1.5 Raster

During the definition of a points raster the raster points are projected from the image plane onto the workpiece. As a rule, measured points will only be placed on those objects that the observer sees as lying “in front”. For highly-curved surfaces there is, under certain circumstances, no clearly-defined point of intersection with the surface, which results in measured points on more distant surfaces.

If measured points are required to lie only on specific objects, these objects are collected together in a group. The raster points are then projected solely onto these objects. Measured points that lie outside are not considered.

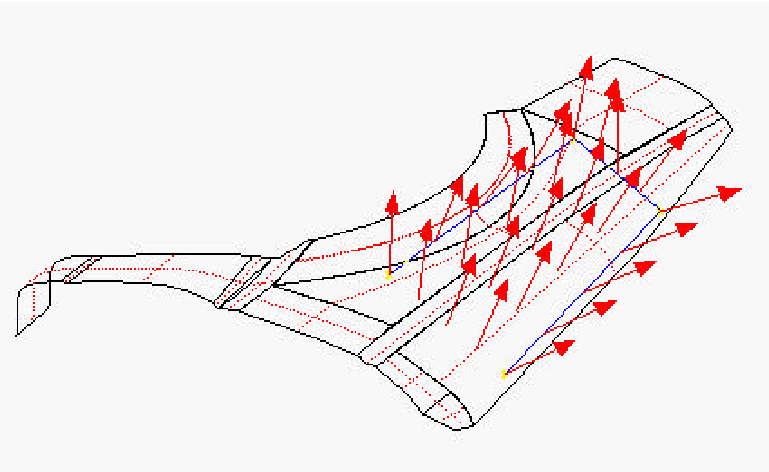


Figure 7-10

The area that is to be rasterized is established by defining a boundary rectangle. One measuring run is generated per raster and displayed on the screen with arrows in the colors of the currently active probe. The measuring run is saved automatically.

The number of raster points must be greater than 2 in each direction!

Purpose of the function

Since you determine the area for the raster yourself, it is therefore possible for you to measure very specific and critical areas. In comparison with the grid you can leave the border area of a surface blank.

A regular raster is an ideal basis, although not a precondition, for a chromatic evaluation of the surface.

Using the actual values of a raster you can calculate a new surface using the <Grid -> Surface> function.

Operation

Click on the <Raster> function.

A window opens for entering raster pointsin the U and V directions.

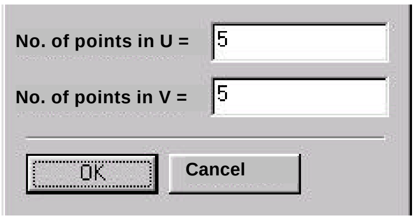


Figure 7-11

Enter the numbers directly into the input fields and confirm your entries with <OK>.

Now you need to draw the border for the area that is to be rasterized. Proceed as follows:

Click on the four corner points of the area with the mouse.

The fourth and first corners are joined up automatically. The area is rasterized immediately. The raster points are displayed as arrows in the color of the currently active probe.

The measuring run is saved.

7.1.6 0.5 points

0.5 points are the mid-points of individual patches or surfaces (in the parameters u = 0.5 and v = 0.5).

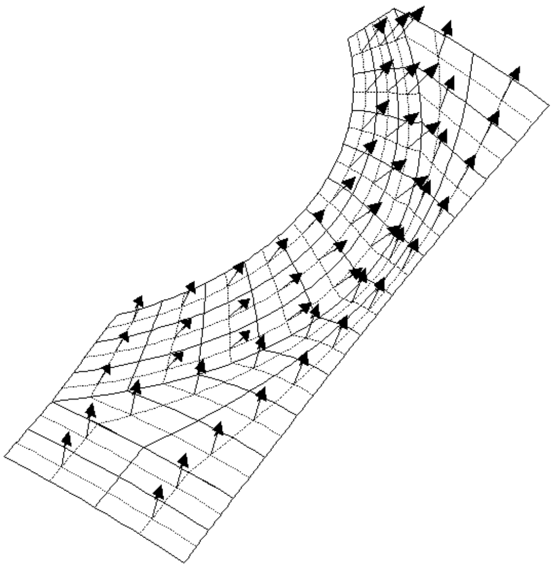


Figure 7-12

The function for generating 0.5 points branches into three subfunctions.

Function	**Meaning**
<Group>	Measured points are only generated on the selected group.
<Surfaces>	Measured points are generated on all surfaces of the model.
<Patches>	Measured points are generated on all patches of the model.

In the case of surfaces which are limited by FACE elements, measured points are only accepted if they lie within the FACE.

> ***NOTE***
>
> When departing 0.5-points, the coordinate measuring machine does not move in a meandering fashion, as e.g. with a grid, but dependent on the sequence of the defined points.
> In highly-curved areas there is therefore a risk of collision!
> Select the areas accordingly, or move the coordinate measuring machine into the clearance plane after each point (set in the measuring run parameters, Chap. 13.5).

Operation

Click on the <0.5 points> function and then <Group>, <Surfaces> or <Patches>.

The measured points are displayed as arrows in the color of the active probe.

The measuring run is saved automatically.

7.1.7 Parallel curve (UX)

This function is used to measure parallel to surface boundaries .

Due to the potential risk of collision, it is not always possible to measure directly on the surface boundaries. Therefore, you must generate measuring points parallel to the surface boundaries (offset curve).

Surface boundaries are the edges of surfaces and patches or the boundary curves of trimmed surfaces (FACES).

The generation of measuring points can either occur with a preset number of measured points or dependent on curvature.

Operation

Select the function <Define measuring run>-<Parallel curve> and then either <Specify number of points> or <curvature-dependent>.

When you click on the function with the left mouse button, an input window opens for defining the following parameters:

Measurement of free form surfaces

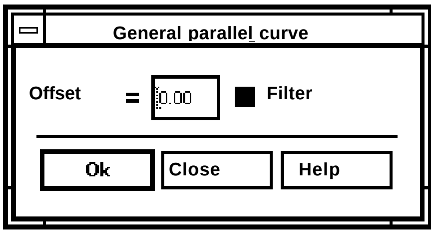


Figure 7-13

Offset The offset denotes the distance of the parallel curve from the surface boundary.

Filter The entire parallel curve is always generated on a selected surface. When the filter is active, the “inner“ points are eliminated.

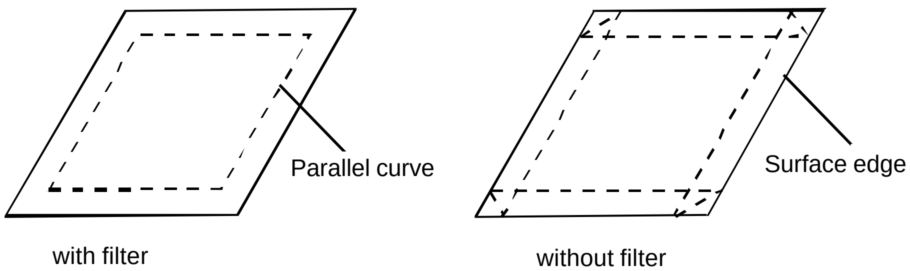


Figure 7-14

Accept the parameters by clicking on <OK>.

The parallel curve is generated on the previously selected surface.

Define the area of the graphically represented parallel curve, on which measuring points are to be generated, by clicking on a start and an end point on the parallel curve. If the start and end point are identical, measuring points will be generated for the entire parallel curve.

The defined area is marked in contrasting color.

> ***NOTE***
>
> If you have inadvertently defined the “wrong” area, switch the order of the start and end point.

As soon as the area is defined by its limit points, a dialog window appears for point generation.

Point generation with constant distance on a line

Select the function <Parallel curve> - <Specify number of points>.

A window opens for entering the number of line points.

Enter the number of points and confirm with <OK>.

Curvature-dependent point generation on a line

Select the function <Parallel curve>-<curvature-dependent>.

A window opens for entering the parameters

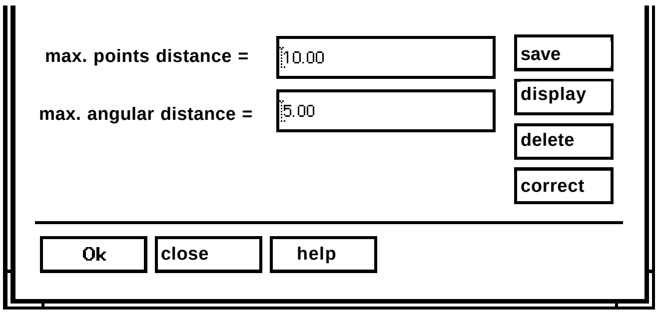


Figure 7-15

max. points distance:	This parameter controls the maximum distance between the generated measuring points, for cases where the criterion for curvature-dependent generation is not satisfied.
max. angle change:	This parameter defines the value for the maximum angle change after which a measuring point is to be generated.
save	The generated measuring points are stored internally in a table. These measuring points are saved as a measuring run in a file using the <Save> function. When points are next generateda new measuring run is defined.
display	If the content of the graphic window has been deleted (clear), you can use this function to display all points in the internal memory for parallel curves.

Measurement of free form surfaces

delete — All measuring points on parallel curves present in the internal memory are eliminated.

correct — You can delete individual measuring points using the Correct key. Each press of the key removes the last measuring point respectively.

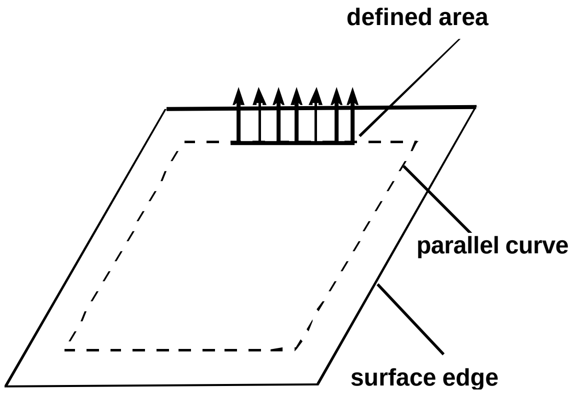


Figure 7-16

Enter the appropriate values or select a function and confirm your selection with <OK>.

The function for generating measuring points on the defined area boundary is activated.

7.1.8 Regular geometry

Measuring runs for regular geometry elements are generated via the function for measuring regular geometries.

> ***Prerequisite***
>
> It is a prerequisite for automatic measurement of regular geometries that the elements are described as freeform geometries in HOLOS. Regular geometries can either be adopted via the IGES interface of CAD systems or generated via the functions of the Digitization option/CAD functions.

Measuring procedures for automatic measurement are available for the following regular geometry elements:

- Circle
- Slot
- Rectangular hole
- Cylinder
- Cone

After measurement and in subsequent evaluations, the measuring points are automatically evaluated as regular geometry elements.

Operation

⇘ Select the <Define measuring run>-<Regular geometries> functions in the <Measurement> menu.

⇘ First of all, select a surface element that describes the required geometry element.
Then define the parameters.

> 🕮 The parameters for the various regular geometry elements are described below.

⇘ Confirm your entries with <OK>.

↳ The measuring run is saved.

Measurement of free form surfaces

Measuring a circle

First select a surface element that describes a circle. Then define the parameters for circle measurement.

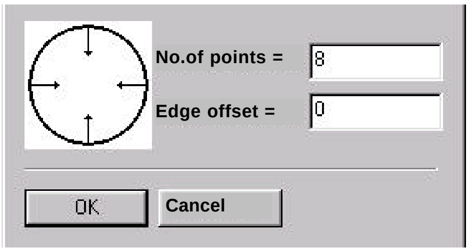


Figure 7-17

No. of points	Enter the number of points that are to be measured on the circle.
Edge offset	Specify a value for the edge offset if the points on the circle are not to be probed directly on a workpiece edge. The probe points are then moved downwards by this amount.

Measuring a slot

First select a surface element that describes a slot. Then define the parameters for measuring the slot.

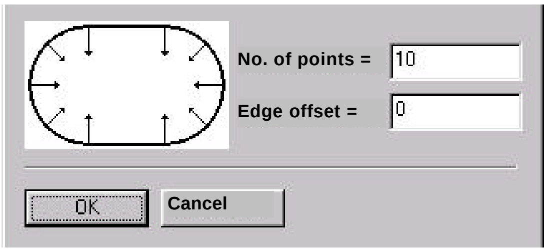


Figure 7-18

No. of points	Enter the number of points that are to be measured on the slot.
Edge offset	Specify a value for the edge offset here, if the points on the slot are not to be probed directly on the workpiece edge. The probe points are then moved downwards by this amount.

Measuring a rectangular hole

⇘ First select a surface element that describes a rectangular hole. Then define the parameters for measuring the rectangular hole.

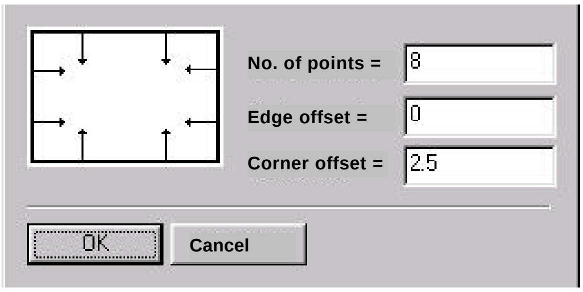


Figure 7-19

No. of points	Enter the number of points that are to be measured on the rectangular hole.
Edge offset	Specify a value for the edge offset, if the points on the rectangular hole are not to be probed directly on a workpiece edge. The probe points are then moved downwards by this amount.
Corner offset	Points in the corner of a rectangular hole cannot be probed. Enter a value for the corner offset. The input value must be greater than the radius of the probe used.

Measurement of free form surfaces

Measuring a cylinder

First select a surface element that describes a cylinder. Then define the parameters for measuring the cylinder.

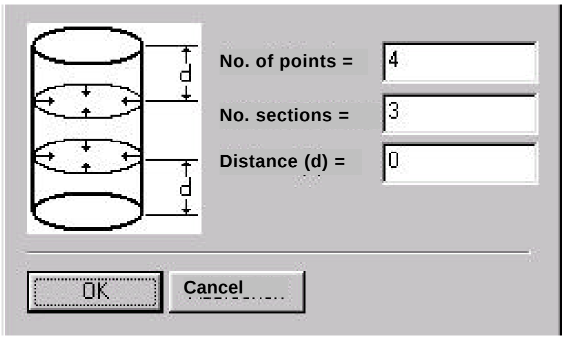


Figure 7-20

No. of points	Enter the number of points that are to be measured on the cylinder.
No. of sections	Enter the number of sections that are to be measured on the cylinder. If only one section is defined, the cylinder is measured and evaluated as a circle.
Distance	Enter a value for the distance which the probe points are to have from the upper and lower cylinder edge.

Measuring a cone

First select a surface element that describes a cone. Then define the parameters for cone measurement.

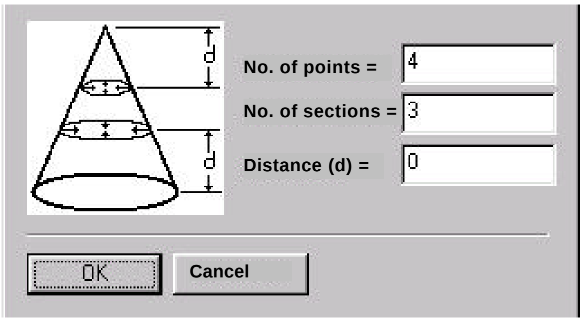


Figure 7-21

No. of points — Enter the number of points that are to be measured on the cone.

No. of sections — Here enter the number of sections that are to be measured on the cone. If only one section is defined, the cone is measured and evaluated as a circle.

Distance — Enter a value for the distance which the probe points are to have from the upper and lower edge of the cone.

7.1.9 Measuring points

Measuring points are defined by clicking on a point on the screen. The point that has been clicked on is then projected onto the workpiece.

When defining measuring points, the points are projected from the image plane onto the workpiece. As a rule, measuring points are only projected onto objects that lie “in front“ of the observer. With highly curved surfaces, in some cases it may not be possible to define a clear point of intersection with the surface, which results in measuring points being located on surfaces that are further away.

If measuring points are required to lie only on specific objects, collect these objects together in a group. The measuring points will then be projected solely onto these objects. Measuring points that lie outside will not be considered.

Measurement of free form surfaces

The mode for defining measuring points is activated with the <Measuring points> function. When active, you can define measuring points at any time by clicking on them (visible as "Select: point" in the status line).

Operation

Click on the <Select measuring points> function (NT) or <Measuring points> (UX).

A window containing further functions opens.

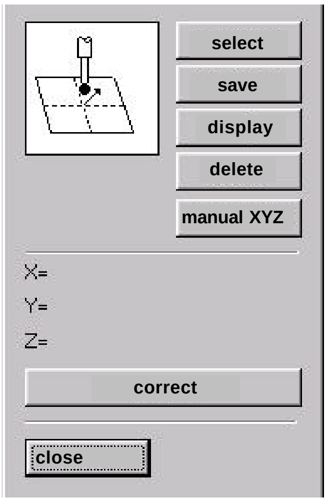


Figure 7-22

Function	**Meaning**
select	Switches on the mode for defining measuring points. This saves the path via the menu system, if for example the mode was exited to define a group.
save	Writes all selected measured points into a file as a measuring run. New selections after saving define a new measuring run.
display	All selected points are re-displayed on the screen if you have previously switched off their display.
delete	Deletes all measuring points from the buffer. The nominal values will not be deleted!
manual XYZ (NT)	This function allows you to define a measuring point by entering its

coordinates x, y, z. The prerequisite for this is that the point can be projected onto a surface (see below).

Click on the <Select> function.

Position the measuring points on the screen.

The coordinates will be displayed for each point that is positioned.

The coordinate window's <Correction> function allows you to delete the last point from the buffer.

If you locate an incorrect surface orientation when generating measured points (normal vectors point towards the interior of the workpiece):

Click on the surface with the mouse.

Activate the <Rotate orientation> function in the <Object> menu.

This rotates the orientation of the surface. All measured points located on this surface are also automatically rotated.

Manual entry of points

To manually enter the coordinates of a measuring point, click on <manual XYZ>.

NT: You can define several net points via the “Net points” key.

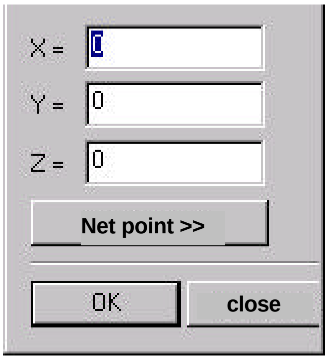


Figure 7-23

Enter the point coordinates into the input fields.

Click on <OK>

The point is projected onto a surface, transferred into the data structure and displayed on the screen.

Click on <Close> in order to close the window.

UX

You will find the window for manual coordinate entry in the “Measurement“ menu - <Define measuring run> - <manual entry XYZ>.

7.1.10 Edge points

Measuring points for edge measurement (Trim measurement) are defined by clicking on a point on the screen. The selected point is then projected onto the next edge of the workpiece.

If measuring points only lie on particular objects, bring these objects together in a group. The measured points are then only projected onto the edges of these objects. Measured points which are selected on other objects are not taken into account.

The mode for defining edge points is activated in the <Measurement>-<Define measuring run> menu with the <Edge points> function. When this is active, you can define measured points at any time by clicking (visible as “Select: point” in the status line).

Operation

Click on the <Edge points> function.

A window containing further functions opens.

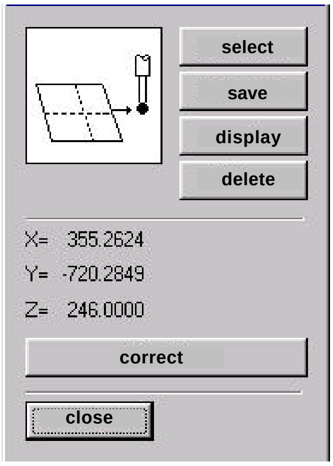


Figure 7-24

Function	**Meaning**
select	Switches on the mode for defining measuring points. This saves the path via the menu system, if the mode has been exited, for example, to define a group.
save	Writes all selected measuring points into a file as a measuring run. Selecting again after saving defines a new measuring run.
display	Re-displays all selected points if the screen content has been deleted by one of the graphic functions.
delete	Deletes all measuring points from the buffer.

Click on the <Select> function.

Set the measuring points by clicking on an edge on the screen.

The coordinates are displayed for each point selected, and the edge points are displayed on the screen with their scanning and normal direction in the color of the currently active probe.

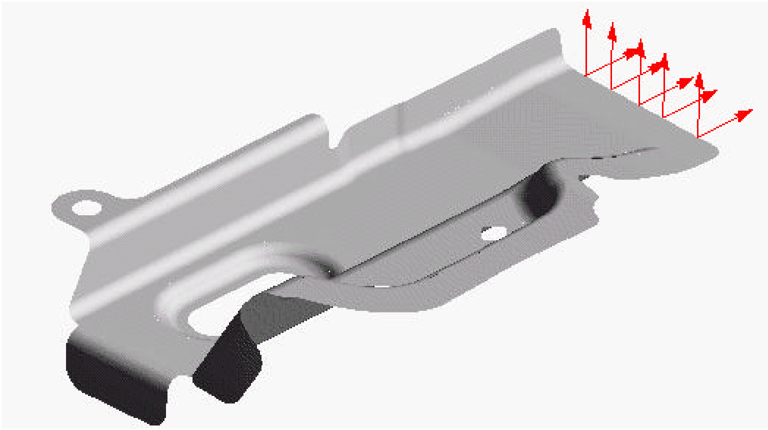


Figure 7-25

The <Correction> function in the co-ordinate window deletes the last point from the buffer.

Before scanning, select the <Save> function.

A dialog window appears which allows you to define in which mode the edge points are scanned.

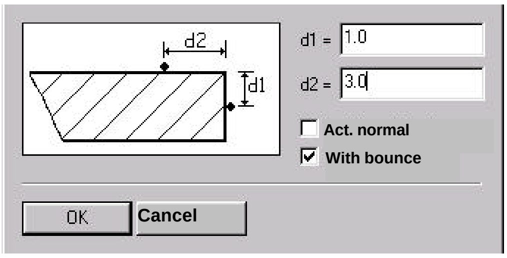


Figure 7-26

With bounce

Activate this switch if the edge is to be measured taking bounce into account in the case of sprung metal parts. When measuring, first of all a point on the surface is scanned. When scanning the edge point, the distance between the preselected nominal value and the determined actual position of the surface point is taken into account.

Define actual normal

Activate this switch if a modified normal direction is to be taken into account in the case of bent metal parts. During measurement, the actual normal on the surface is determined with the help of the surface point. When scanning the edge point, the distance between the preselected nominal value and the determined actual position of the surface point is taken into account. Modified scanning directions are also taken into account.

d1

Enter a value by which the edge point is to be moved downwards.

d2

Enter a value by which the surface point is to be moved inwards.

> **NOTE**
>
> Measuring runs that have been saved cannot subsequently be changed. Therefore, before saving you must decide in which mode the edge is to be measured. The relevant information is saved with the measuring run.

7.1.11 CAD points -> measuring points

Points defined in CAD (CAD points) can be used to define measuring runs once they have been imported.

🕮 See Chap 4.7.1 "Importing CAD points “

Measurement of free form surfaces

Operation

Select the <Measurement>-<CAD points -> Measuring points> function (NT) or <Measurement>-<CAD points> (UX).

A dialog window opens.

Select the required function.

select

Activates the mode for selecting individual CAD points. This is indicated by the display: **Select: POINT** at the bottom right edge of the screen.

> ***NOTE***
>
> CAD points will be adopted as measuring points if a surface can be identified for a given point with a distance from it not exceeding the set value for data import.

The adopted measured points are shown on the screen as arrows in their normal direction to the surface and in the color of the selected probe, along with numbering.

select from...to (NT)

With this function you can select a series of CAD points that lie in a defined order.

Click on the first CAD point

Click on the second CAD point

All points that lie between the first and second selected points are adopted as measuring points.

deselect

With this function you can deselect all currently selected CAD points. You can now define a new measuring run.

save

All selected measured points are written to a file as a measuring run. New selections after saving will define a new measuring run.

display (NT)

All selected points are re-displayed on the screen if you have previously switched off their display.

correct

The correction key deletes the measuring point that was last adopted.

all CAD points

This function adopts all CAD points that satisfy the distance criterion, as measured points in a measuring run. The sequence will be determined by the position of the current point in the VDA file.

NOTE

To avoid collisions during measurement you should define an appropriate clearance plane, to which the probe can travel after each probing.

The measuring runs defined using the above-mentioned methods will be saved as a file which can then be further processed by using the management functions available for measuring runs.

7.1.12 CAD points → edge points

The <CAD point → edge points> function operates in a similar manner to the <CAD point → measuring points> function described above.

When selecting CAD points, the adopted nominal values are displayed together with their scanning direction in the tangential plane. The color of the arrow corresponds to the defined probe.

7.1.13 Contour

If two surface pieces are joined together by a rounded edge, the theoretical line of intersection can be calculated by extrapolation of the two surface pieces (calculated extension).
This geometric element is used particularly in car body construction and in this context is called a contour. The virtual, i.e. physically non-scannable points of a contour are called contour points.

As the contour cannot be directly measured, the relevant contours must be calculated.

Prerequisite

To measure contour points with HOLOS, nominal contour points from the CAD description (CAD points) must be available.
In addition, both edges must be available as an independent surface description, whose intersection gives the contour line.

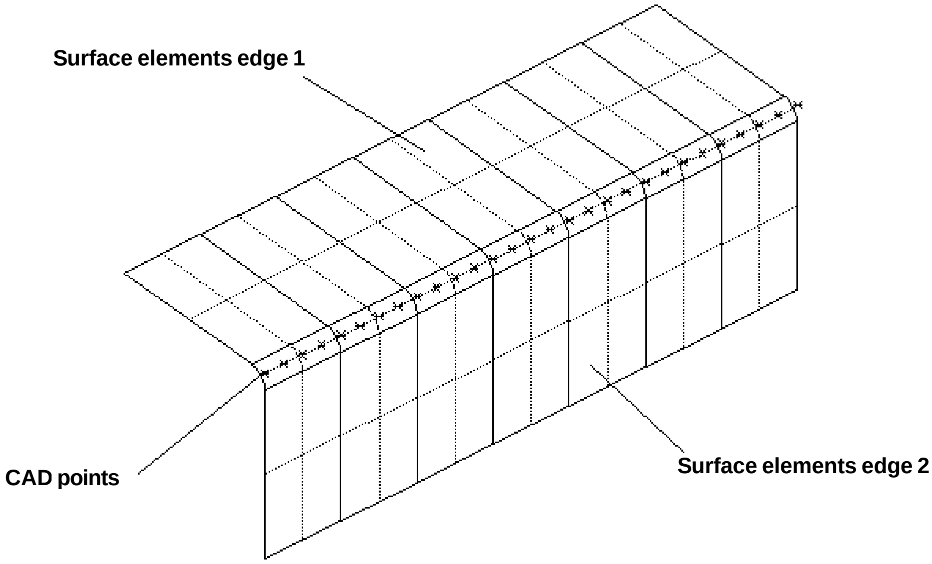


Figure 7-27

Definition of a measuring run

Measuring runs for measuring contour points can be defined in a graphically interactive manner and stored in HOLOS and are thus CNC-capable.

The measuring run is defined in several steps:

1. First define which contour points you wish to measure.
2. Next define the surface elements which form the first edge.
3. Then define the surface elements which form the second edge.
4. Finally, define the parameters for the measuring run.
5. From the entered definitions, the program calculates the probing points in which the actual normal can be defined during probing on the coordinate measuring machine.

To define a measuring run, activate the <Contour line> function in the <Define measuring run> menu.

A dialog window appears for defining the various parameters for the contour measurement:

Measurement of free form surfaces

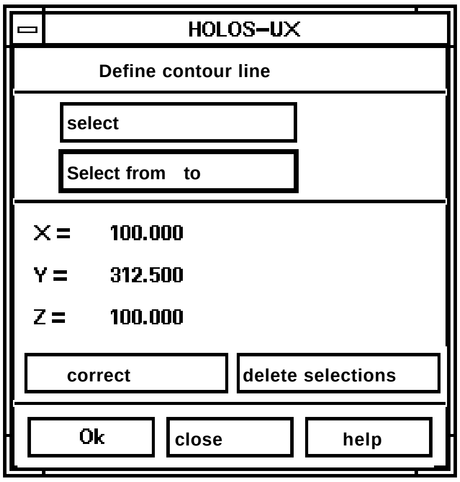


Figure 7-28

Function	Meaning
select	Mode for selecting contour points: In this mode, each contour point must be individually clicked on with the left mouse button.
select from... to...	Mode for selecting a series of contour points: First click on the start point, then on the end point of the contour line. All points in between are taken into account in the measurement.
correct	Deletes the last selected contour point.
delete selections	Deletes all selected elements (contour points, edge elements).
OK	Confirm the selected contour points. You then reach the next level for edge definition.
close	Cancels the definition of a measuring run.

When you have defined the parameters for the outline measurement, click on “OK“.

The dialog window appears for defining both edges whose theoretical line of intersection forms the contour line:

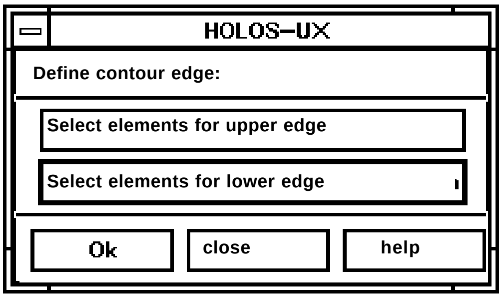


Figure 7-29

Function	Meaning
Select elements for upper edge	Mode for selecting the elements for the first edge: The selected elements are displayed on the screen in contrasting color.
Select elements for upper edge	Mode for selecting the elements for the second edge: The selected elements are displayed in contrasting color on the screen.
OK	Confirm the definition of both edges. You then reach the next level for defining the parameters for calculating the probing points.
Close	Cancels the definition of a measuring run.

When you have defined the edges, click on “OK“.

A dialog window appears for defining the parameters for calculating the probing points:

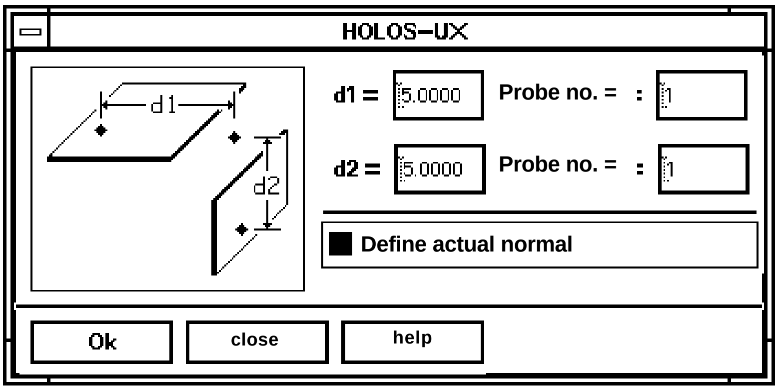


Figure 7-30

Function	**Meaning**
d1:	The value d1 defines the distance of the probing points on the first edge from the selected contour points.
d2:	The value d2 defines the distance of the probing points on the second edge from the selected contour points.
Probe no.:	The number of the probe which is used to probe the points.
Define actual normal:	Switched on: During probing on the coordinate measuring machine, the actual normal on the surface is determined in the preset probing points. Switched off: The normals in the preset nominal contour points are used, in order to determine the deviation of the actual contour point.
Ok:	Starts the function for calculating the probing points.
Close:	Cancels the definition of a measuring run.

When you have defined the parameters for calculating the probing points, click on “OK“.

The probing points are calculated and graphically displayed on the screen.

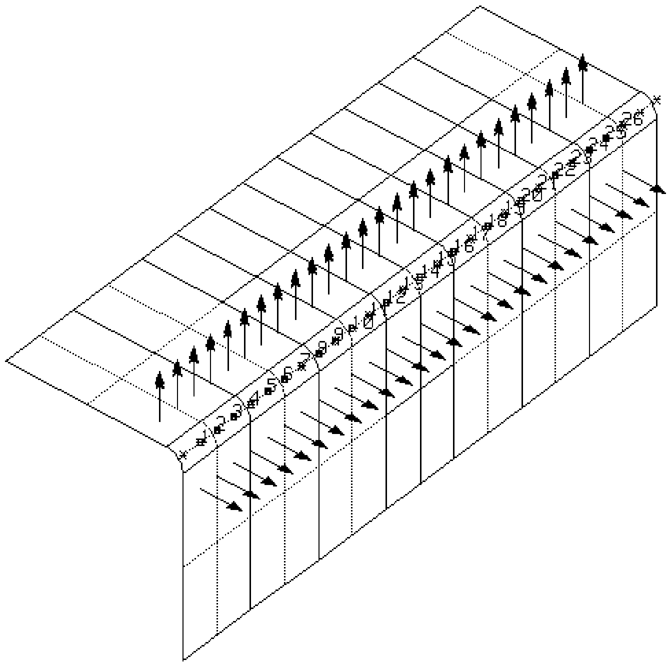


Figure 7-31

Special case with just one contour point

If you have selected just one contour point for defining a measuring run, the projection plane in which the contour point measurement is defined must be preset.

To define the projection plane, a dialog window appears, in which you select one of three basic planes.

If, on the other hand, several contour points are selected for calculating the probing points, then the probing points are always calculated in the plane which lies vertically on the connecting vector of two consecutive contour points.

Start measuring run

Start the defined measuring run in the usual manner.

When measuring the contour points, first all points on the first edge are probed, then those on the second edge.

Evaluation

In order to determine an actual contour point, both planes are brought to the intersection, which result from measuring two probing points.

If the switch for defining the actual normals during the definition of a measuring run was switched off, those planes are used which are defined by the normal vectors of the calculated probing points.

The intersection straight line of both planes is intersected with the plane in which a selected contour point lies. The resulting intersection point defines the actual contour point.

To calculate the deviations, the difference between the nominal and the actual contour point is determined and graphically displayed on the screen.

7.1.14 Corner points

Corner points are theoretical points which are defined by the intersection of three surface sections, forming a corner:

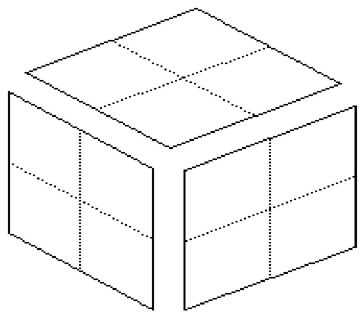


Figure 7-32

In order to define a corner point, select three points on the respective surfaces segments. The **nominal corner point** then results from the point of intersection of the three planes, which contain the selected measuring points.

During the measurement, the actual normal is determined on the probed surface in the respective probing points. The intersection of these three determined planes defines the **actual corner point**.

For the deviation, the difference is calculated between the preset nominal point and the calculated actual point and displayed on the screen.

Defining a measuring run

Select the <Corner point> function in the <Define measuring run> menu.

A dialog window appears for defining the points on the three surface segments, which form a corner:

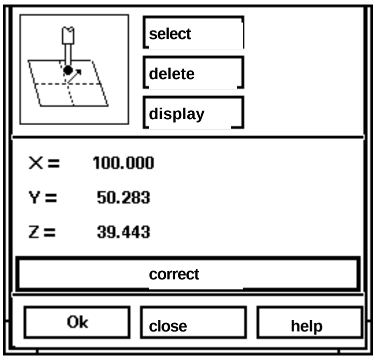


Figure 7-33

Function	**Meaning**
select	Mode for selecting the three surface points via which a nominal corner point is calculated.
delete	Deletes all selected surface points.
display	Displays the previously selected surface points, if the content of the graphic screen has been deleted by another function.
correct	Deletes the last selected surface point.
Ok	Starts calculation of the nominal corner point. The corner point is saved as a measuring run and can be measured via the <Start measuring run> function.

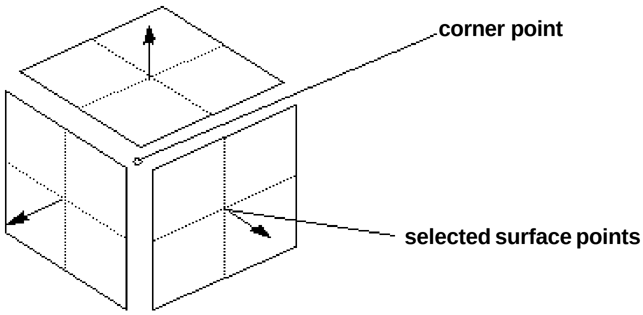


Figure 7-34

7.1.15 Net point/Net section

Net points or net sections are measuring points which are scanned on a defined grid on the workpiece.

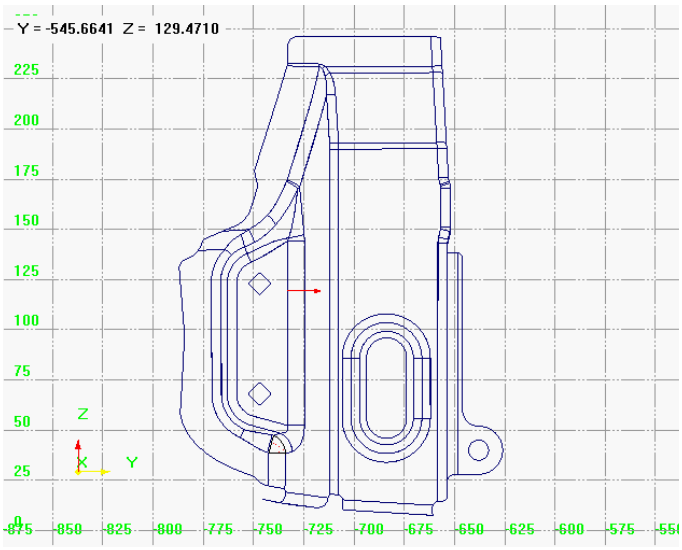


Figure 7-35

With HOLOS-NT you can define individual net points, net lines (net sections) or net rasters.

⇘ Start the <Net point/net section> function in the <Measurement> - <Define measuring run> menu and define the parameters for generating a measuring run:

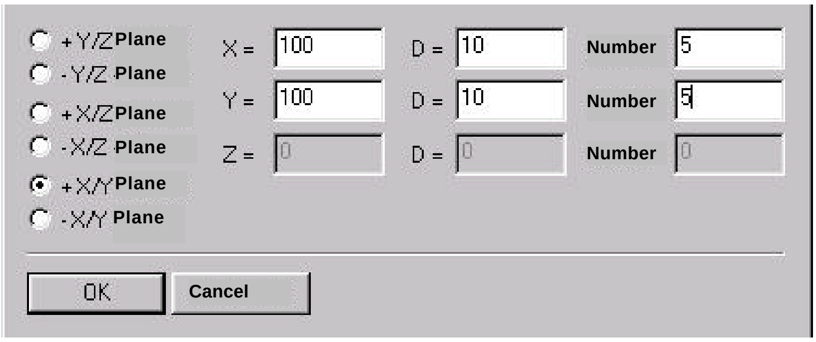


Figure 7-36

Net plane

The net plane defines the projection plane of the grid as well as the projection direction on the workpiece.

Examples:

Net plane +X/Y defines the X/Y plane as the projection plane of the grid (X/Y-net). The measured points are projected onto the workpiece as seen from the positive Z-axis (from above).

Net plane -X/Y similarly defines the X/Y plane as the projection plane of the grid. However, the measured points are projected onto the workpiece as seen from the negative Z-axis (from below).

Net coordinates

When the projection plane has been defined, the net coordinates are entered.
Net coordinates can only be defined for the respective projection plane (X/Y coordinates for the X/Y plane); the input mask for the third coordinate is blocked.

Definition of a net point

A net point is defined by the following parameters:

- Enter the net co-ordinates for the point.
- Enter the value 1 in the input mask for the number of points.

- The content of input mask D= is ignored.
- Click on <OK> with the left mouse button.
- The measured point in the input net co-ordinate is projected onto the workpiece and stored.

Definition of a net line (net section)

A net line is defined by the following parameters:

- Enter the net coordinate of the start point for the net line.
- Enter the distance of the net co-ordinates for the required section direction (D =).
- Enter the number of points for the required section direction (Distance =).
  Enter the value 1 in the input mask for the number of points of the second section direction.
- Click on <OK> with the left mouse button.
- The measuring points for the net line are projected in the defined net co-ordinates onto the workpiece and saved.

Definition of a net raster

A net raster is defined by the following parameters:

- Enter the net coordinate of the start point for the net raster.
- Enter the distance of the net coordinates for both section directions of the net raster (D =).
- Enter the number of points for both section directions of the net raster.
- The measured points for the net raster are projected in the defined net coordinates onto the workpiece and saved.

7.1.16 Scan line

With this function you can scan points on a line that you have defined.

Operation

Select the <Scan line> function.

Define a line with points A and B, by drawing the line from start point A using the mouse with the left button pressed down.

The line is displayed on the screen whilst it is being drawn.

Release the mouse button.

A window opens for defining the scanning parameters.

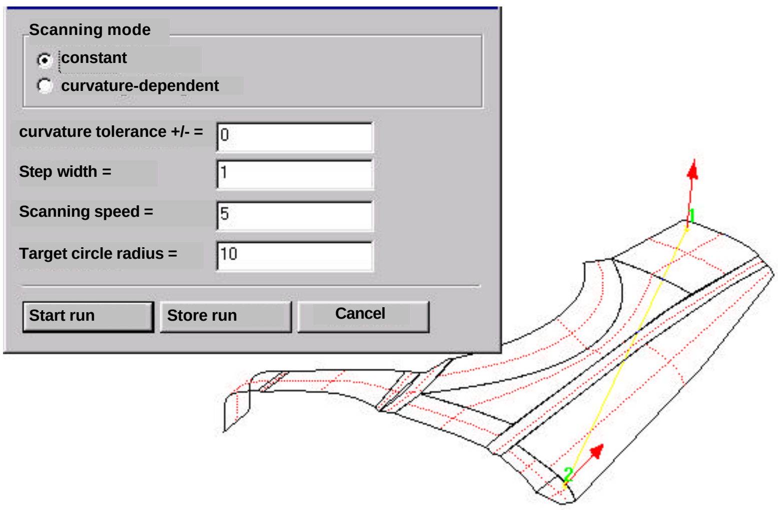


Figure 7-37

constant

Points are recorded during the scanning run with a constant point distance.

Measurement of free form surfaces

curvature-dependent

Points are recorded during the scanning run dependent upon the curvature change of the contour to be scanned.

curvature tolerance

This parameter defines the curvature tolerance for the scanned contour during curvature dependent points recording. If the curvature tolerance is exceeded, a new measuring point will be adopted.

The curvature tolerance describes the maximum chord error between three recorded measured points in mm.

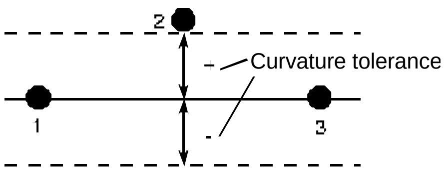


Figure 7-38

step width

This parameter defines the distance between the recorded points for point recording with constant distance.

For curvature-dependent point recording, the maximum point distance will be defined as a result of this. If within the specified area no curvature change in the contour takes place, a new measured point will be adopted when the specified distance is reached.

scanning speed

This parameter defines the traverse rate of the coordinate measuring device during the scanning run.

target circle radius

The target circle radius defines the radius of a circle around the target point of a scanning line. After entering this target circle, the coordinate measuring device reduces the traverse rate in order to terminate the scanning run on reaching the target point.

start run

With the <Start run> function you start the scanning run with the start and end points calculated beforehand, and the parameters that have been defined.

save run (NT)

With the <Save run> function you can save the run as a measuring run and execute it later.

cancel

Use this function to cancel the entire process.

Enter the relevant values and select one of the functions.

The corresponding function will be executed.

7.1.17 Scan area

With this function you can scan any defined area.

Operation

Select the <Scanning area> function.

A window opens for defining the required area.

Measurement of free form surfaces

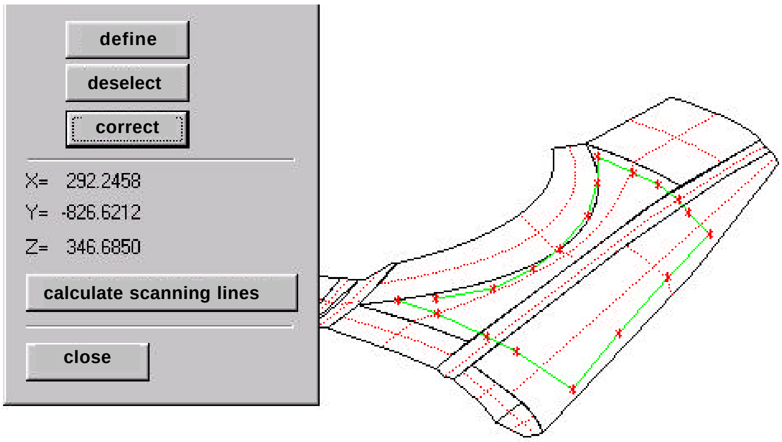


Figure 7-39

define

The <Define> function activates the mode for defining the area to be scanned. This is done by establishing the area's boundary points. You can create any boundary points you wish for the area to be defined.

To define boundary points, click on the specific point in the graphic representation of the workpiece surface.

The boundary points are immediately accepted and joined together as a polygon. The boundary points polygon is displayed graphically on the screen, and the current coordinates of the last point to be selected are shown in the center of the displayed window as X, Y and Z values.

Using this window's correction key you can delete the displayed point (i.e. the last one to be selected) from the boundary polygon.

The boundary points polygon can - but does not have to - be closed. If the first and last points are not identical, the area is automatically closed.

deselect

With the <Deselect> function you delete the last boundary area defined, and you can redefine an area.

calculate scanning lines

To calculate scanning lines you must define further parameters.

Select the <Calculate scanning lines> function.

A window opens for selecting the projection plane and entering an angle as well as the line distance:

projection plane

The intersection plane for the area to be scanned is defined by a projection plane and an angle in the projection plane. The probing or backaway direction in the current start/end points is determined by the normal vector of the corresponding point on the surface.

angle

Use this parameter to define the location of the respective intersection planes in the projection plane.

distance between lines

The distance between lines defines the distance between the individual scanning lines .

Enter the required parameters and confirm with <OK>.

The start/end points for the subsequent scanning run are calculated and graphically displayed.

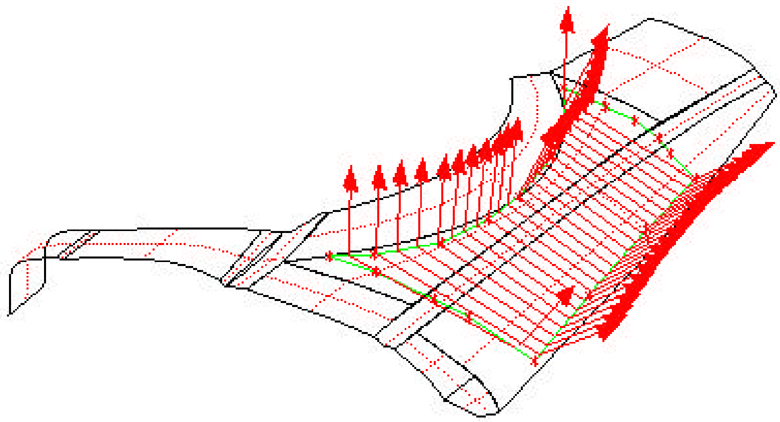


Figure 7-40

NOTE

Observe the direction of the normal vectors in the respective start/end points. If the orientation of a surface is not correctly defined, the probe will be pointing in the wrong direction, which leads to collisions during probing or backaway during the subsequent run.

The dialog window for defining the parameters for the scanning run then appears:

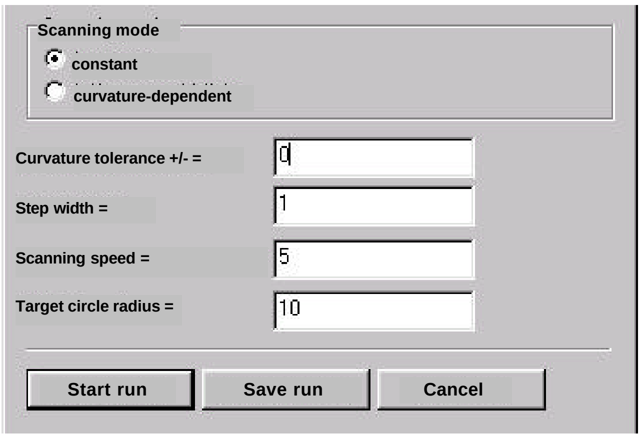


Figure 7-41

constant

Points are recorded during the scanning run with a constant point distance.

curvature-dependent

Points are recorded during the scanning run dependent on the curvature change of the contour to be scanned.

curvature tolerance

This parameter defines the curvature tolerance for the scanned contour during curvature dependent points recording. If the curvature tolerance is exceeded a new measured point will be adopted.

The curvature tolerance describes the maximum chord error between three recorded measuring points in mm.

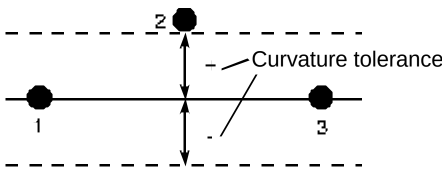


Figure 7-42

step width

This parameter defines the distance between the recorded points during points recording with a constant distance.

With curvature-dependent points recording, the maximum points distance will be defined as a result of this. If no curvature change takes place in the contour within the specified area, a new measured point will be adopted after reaching the specified distance.

scanning speed

This parameter defines the traverse rate of the coordinate measuring device during the scanning run.

target circle radius

The target circle radius defines the radius of a circle around the target point of a scanning line. After entering this target circle the coordinate measuring device reduces the traverse rate in order to terminate the scanning run on reaching the end point.

start run

With the <Start run> function you start the scanning run with the start and end points calculated beforehand, and the parameters that have been defined.

save run (NT)

With the <Save run> function you can save the run as a measuring run and execute it later.

cancel

This function cancels the entire process.

Enter the relevant values and select a function.

The corresponding function is executed.

7.1.18 Start last measuring run

With this function you start the last defined measuring run immediately without any user queries beforehand.

Operation

Select the <Define measuring run>-<Start measuring run> function.

The last defined measuring run will be started immediately without any further queries.

7.1.19 Scanning in accordance with nominal values

With this function you start the last defined measuring run in the “Scanning in accordance with nominal values“ mode.
Measuring runs in the “Scanning in accordance with nominal values“ mode can only be executed on coordinate measuring devices with a measuring probe head or with laser probes.

Operation

Select the <Define measuring run>-<Scanning in accordance with nominal value> function.

A dialog window opens for defining the scanning parameters.

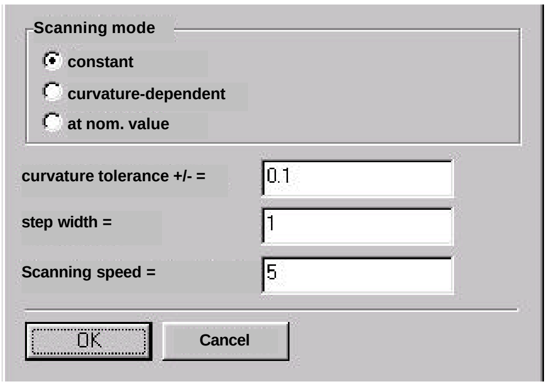


Figure 7-43

constant The points are recorded during the scanning run with a constant points distance.

curvature dependent The points are recorded during the scanning run dependent upon the curvature change of the contour to be scanned.

at nominal value The points are recorded during the scanning run at the nominal value, i.e. measuring points are only recorded where nominal values are defined.

step width This parameter defines the distance between the recorded points during points recording with a constant distance.
During curvature-dependent points recording, the maximum points distance is defined as a result of this. If no curvature change occurs in the contour within the specified area, a new measured point is adopted upon reaching the specified distance.

scanning speed This parameter defines the traverse rate of the coordinate measuring device during the scanning run.

Enter the relevant parameters and click on <OK>

The last defined measuring run is started immediately in the “Scanning according to nominal values” mode, without any further queries.

7.2 Start measuring run

This function selects an already existing measuring run for the active model.

Operation

Select the <Measurement>-<Start measuring run> function.

A window opens for selecting existing measuring runs.

Select the required measuring runs.

The measuring runs are started as soon as you have selected them and confirmed them with <open> (NT) or <OK> (UX).

NOTE

The points or surfaces of a measuring run are processed in the order in which they have been selected!

In principle, you can start a number of measuring runs, since they will be buffered in a queue and processed in the background.

If you are no longer certain of the name of the desired measuring run, a graphic display of the measuring run can be obtained with the <Display nominal values> function.

NOTE

Move the coordinate measuring device to a suitable starting position before the start of a measuring run!

7.3 Start measuring run (scanning in accordance with nominal values)

With this function you select an already existing measuring run for the active model.

The measuring run is executed in the "Scanning in accordance with nominal values" mode.

Measuring runs in the "Scanning in accordance with nominal values" mode can only be executed on coordinate measuring devices with a measuring probe head or with laser probes.

Operation

Select the <Measurement>-<Start measuring run> function.

A window opens for selecting the available measuring runs.

Select the required measuring runs and click on <Open>.

A window opens for defining the scanning parameters.

Enter the relevant parameters.

The measuring runs are started as soon as you have selected them and confirmed them with <OK>.

7.4 Display measuring run (NT) / Display nominal values (UX)

NT

With the <Display measuring run> function, you can display measuring runs which have already been defined and are stored on the hard disk, on the screen.

UX

The <Display nominal values> function displays the defined measuring runs graphically on the screen (each point as an arrow in the normal direction). The function displays for selection all existing measuring runs for the active model.

Operation

Select the <Measurement>-<Display measuring run> function (NT) or <Display nominal values> (UX).

A window opens for selecting existing measuring runs.

Select the required measuring runs and click on <Open> (NT) or <OK> (UX).

The selected measuring runs are displayed on the screen.

7.5 Cancel measuring run

The <Cancel measuring run> function cancels the queue in which the measuring runs are buffered.

NOTE

A measuring run which has already been processed by the measuring software will **not** be cancelled with this function! It can only be cancelled using the cancel functions in the measuring software or CADLINK.

Procedure when a collision occurs

Before you release the probe, first cancel all measuring runs with the <Cancel measuring run> function!

7.6 Simulate measuring run

With the <Simulate measuring run> function you can simulate a measuring run on the screen.

When programming macros without a coordinate measuring device, you cannot perform measurements in order to obtain an evaluation. However, measured values are essential to define the positions of the deviation icons for evaluations within a macro.
If you simulate a measuring run on the screen for this purpose, you can freely define the positions of the measured values which occur in the subsequent evaluation, and store them for later evaluation.

Operation

Select the <Measurement>-<Simulate measuring run> function.

A window opens for selecting the available measuring runs.

Select the required measuring runs and click on <Open> (NT) or <OK> (UX)

A dialog opens for defining the simulation parameters.

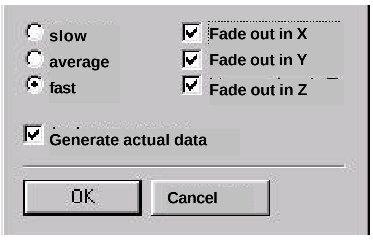


Figure 7-44

Slow, average, fast These parameters are used to control the speed with which the probe is displayed on the screen.

Generate actual data Press this key if actual values are to be generated during the simulation.
An evaluation of the simulated measuring runs is only possible with this option. Measured values from a simulation are marked accordingly in the measuring record.

Measurement of free form surfaces

Fading out in X, Y, Z

When generating actual data, you can fade out the data. You define in which direction a deviation for the actual values is to be simulated.

Define the relevant parameters and click on <OK>

The selected measuring runs are simulated on the screen with the set measuring run parameters.

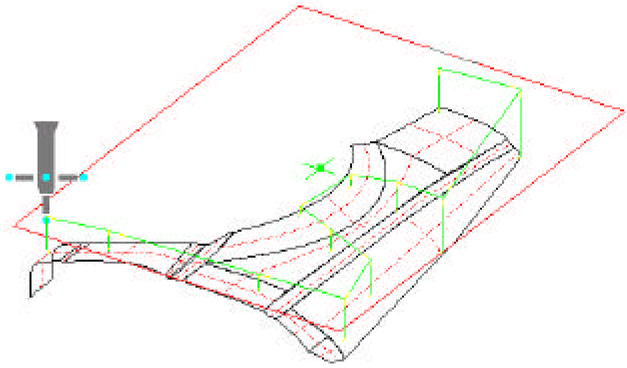


Figure 7-45

7.7 Edit nominal values

With the <Edit nominal values> function you can modify measuring runs that have already been saved. Modified measuring runs are saved under a new name.

If you wish to save a measuring run under the same name, activate <File name>->define> in the system parameters.

see Chapter 13.16 System parameters

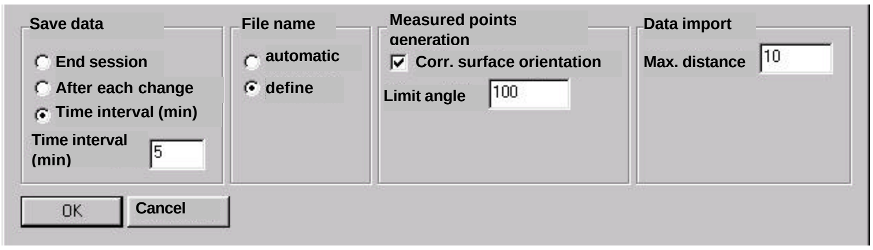


Figure 7-46

You can then enter a name when saving the modified measuring run and thus overwrite the old measuring run.

If measured values already exist for this measuring run, it may no longer be possible to evaluate these correctly.

Operation

Click on the <Edit nominal values> function.

A dialog window for evaluating measuring runs appears on the screen.

Select one or more measuring runs and click on <Open>.

The dialog window for the nominal value editor opens.

Measurement of free form surfaces

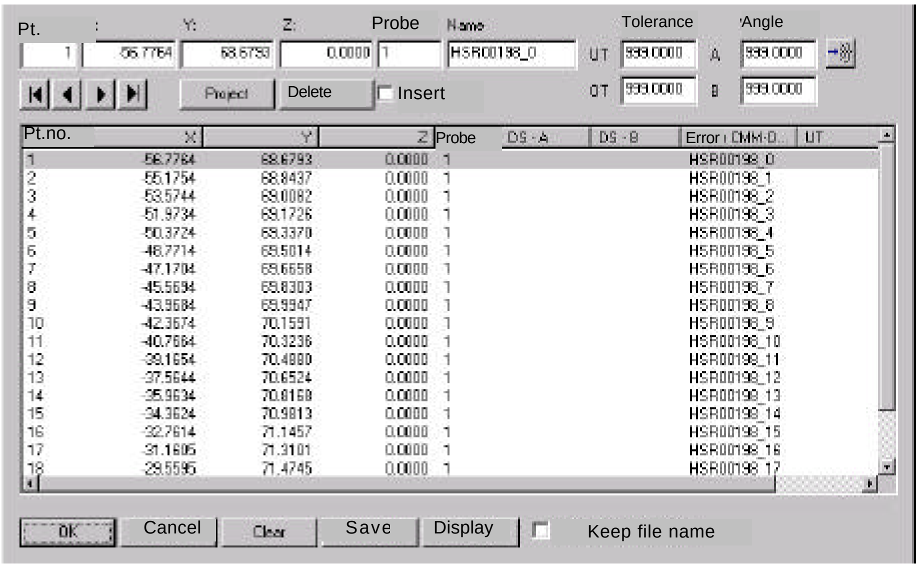


Figure 7-47

You can combine several measuring runs if you select more than one run. You can achieve the same result by selecting another measuring run while point data are still located in the editor window.

Only with HOLOS-NT:

Name	A **name** is automatically generated for each measuring point. You can change it.
Tolerance	You can preset **tolerances for individual points**. If a point does not have a preset tolerance, then a search is made for the tolerance for the area. If this is not present either, then the global tolerance applies.
Angle	The **angle position** can be specified for DSE and RDS.

You can **sort** the nominal values by clicking on the keys, e.g. by point no., X, Y, Z etc.

In order to select several points, use the Shift and Control keys in accordance with Windows convention.

With the Send key, you fetch the current **angle** of DSE or RDS, which is then entered automatically.

Changing measured values

To change a nominal value, you must move the relevant nominal value into the top line of the editor.

Use the control keys (arrow keys) in order to move the nominal value to be changed into the top line

or

click on the nominal value to be changed with the left mouse button.

Enter the new coordinates for the nominal value into the input fields X, Y, Z and click on <Projection>.

The entered value is projected onto the workpiece and adopted when a new point is found. If no projection is possible for the entered value, no change occurs.

Change probe number

In order to change the probe number of an individual measuring point, enter the number into the "Probe" input field and confirm the entry with the return key.
In order to allocate all measuring points to the same probe, click on the right mouse button in the editor window. "Set probe" appears, with which you can accept the current probe for all measuring points in the list.

Deleting measured values

To delete a nominal value, you must move the relevant nominal value into the top line of the editor.

Use the control keys (arrow keys) in order to move the nominal value to be deleted into the top line

or

click on the nominal value to be deleted with the left mouse button.

Click on <Delete>.

The measured value is deleted.

Inserting measured values

You can insert measured values into an existing measuring run.

To insert a nominal value, you must move the nominal value into the top line of the editor, in front of which the new value is to be inserted.

Click on <Insert>.

Enter the new coordinates for the nominal value into the input fields X, Y, Z and click on <Projection>.

The entered value is projected onto the workpiece and adopted when a point is found. If no projection is possible for the entered value, no adoption takes place

or

click on <Insert>.

Activate the function for defining measuring points on the workpiece surface <Measurement>-<Define measuring run> – <Measuring point>.

Click with the left mouse button on the workpiece surface in order to define new measuring points.

The measuring points are adopted and inserted into the measuring run in front of the point in the input line.

The inserted measuring point is always given the position in the point list for which the point number appears in the top input line.

Keys

The keys beneath the editor window have the following meaning:

Keep file name
With this option, the measuring run is saved under the same name as previously. If the option is not selected, then you will be asked to enter a new file name.
The option has no meaning if several measuring runs are loaded into the nominal value editor.

OK
You can exit the editor with <OK> if you have not made any changes to a measuring run. If you have made changes, you cannot exit the editor via this function whilst there are data in the editor window.

Cancel
You can exit the editor with <Cancel> if you have made changes to a measuring run which you do not wish to accept.

Clear
<Clear> deletes the content of the editor window. You can then load new measuring runs into the editor window.

Save
You can save a modified measuring run via the <Save>function. If you have not made any changes, the measuring run will not be saved.
If file names are allocated automatically, a new measuring run is always saved. If you wish to overwrite a measuring run, you must define the file name yourself (see above).

Display
<Display> allows you to graphically display the measuring run on the screen if you have previously switched off the nominal value display via <Clear> in the graphics menu bar.

8 Regular geometry (NT)

This chapter describes the functions of the <Regular geometry> menu.

The main function < Regular geometry >-< Regular geometry> starts the module for management, measurement and evaluation of regular geometries.

8.1 Screen layout

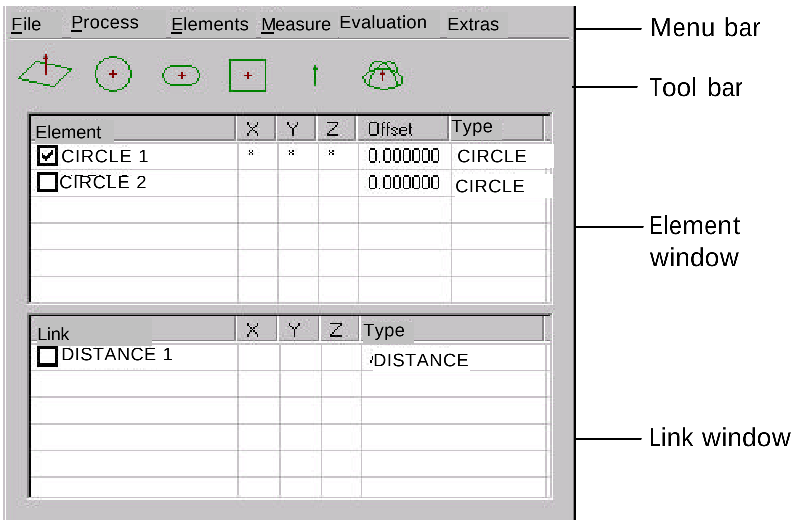


Figure 8-1, Screen layout

Menu bar

The menu bar contains all main functions of the regular geometry module. Each main function branches into a menu with subfunctions.

Tool bar

The tool bar contains the symbols for quick creation of regular geometry elements.

Element window

The regular geometry elements are listed in the element window.

Element:	Description of the element with status:
	Element is selected and not marked
	Element is not selected and marked
	Element is selected and marked.
X, Y, Z:	Selection for RPS alignment
Offset +Z:	Correction value for offset elements, e.g. in order to take account of material thicknesses
Type:	Name of the element type

Link window

The link window lists the regular geometry links.

Link:	Description of link with status
X, Y, Z:	Selection for RPS alignment
Type:	Name of link type

NOTE

If a link is selected, then the necessary elements are automatically also selected. If the selection of an element is cancelled, then the selection of all referenced links is also cancelled.

8.2 Data management

The functions of the <File> menu are mainly for the management of regular geometry files.

- New
- Open
- Save
- Save as
- Import Catia list
- Export Catia list
- Close

8.2.1 New

Create a new regular geometry list.

If the active list has not yet been saved, you will be asked to save it first. This also applies for the <Open> and <Close> functions.

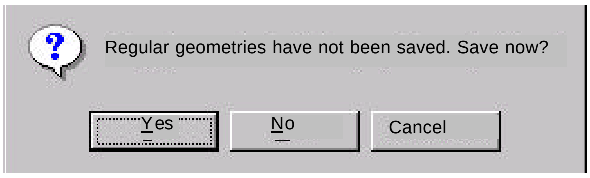


Figure 8-2

Yes:	Save list and execute function.
No:	Do not save list but execute function.
Cancel:	Do not save list or execute function

8.2.2 Open and close

An existing regular geometry list is selected and opened.

Figure 8-3, File name for regular geometries

"Close" exits the regular geometry. It must be closed before another model can be loaded.

8.2.3 Save and save as

Save the current list. If the list has not yet been assigned a name, then the <Save as> function is automatically called up.

With <Save as>, the current list is saved under a new name.

8.2.4 Import / Export Catia list

The Daimler-Chrysler company has created an interface in collaboration with the developers of Catia, which enables simple exchange of regular geometries between HOLOS and Catia.

To import, click on <File>-<Import Catia list>.

A window opens for selecting a list.

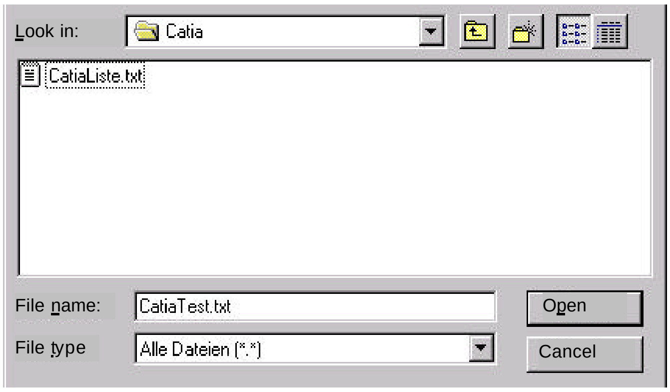


Figure 8-4, Catia list

Click on the file to be imported and then on <Open>. The Catia list can be read from any directory.

The Catia list is imported, and the corresponding elements are set.

The record for the imported elements is displayed:

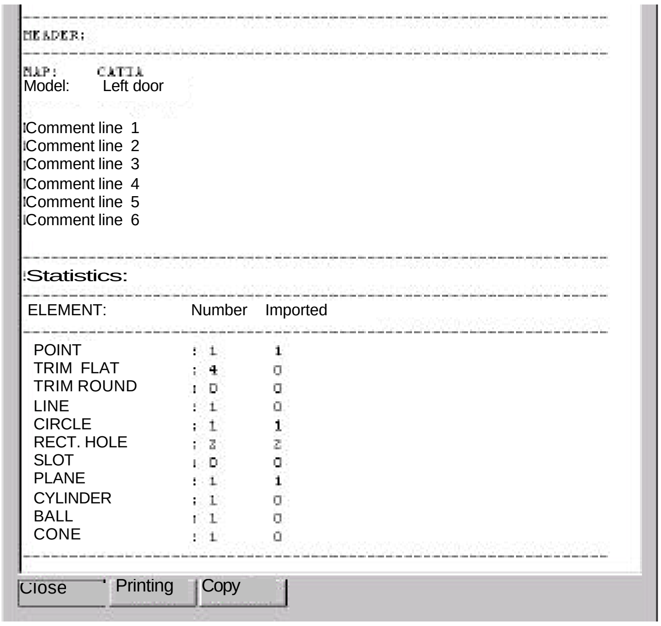


Figure 8-5, Text output: Imported elements

To export, select <File>-<Export Catia list> and allocate a file name under which the Catia list will be saved.

8.3 Process

The processing functions are mainly used to manage regular geometry elements and links within the list.

Cut

The selected elements and links are copied into the clipboard and removed from the list.
These data can only be reinserted into a HOLOS regular geometry list.

Copy

The selected elements and links are copied into the clipboard.

Insert

Elements or links which have previously been copied into the clipboard are added to the existing list.

Delete

The marked element or link is deleted. Deletion is only possible if the referenced elements or links are also deleted.

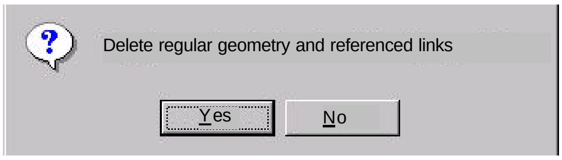


Figure 8-6

Rename

Changes the name of the marked element or link. Renaming can also be performed by selecting the element or link, waiting briefly and then reselecting it.

Process

Changes the parameters of the element or link. Processing can also be performed by selecting the element or link with a double click.

8.4 Elements

The functions of the <Elements> menu serve mainly to create and modify regular geometry elements and links.

> **NOTE**
>
> The circle element is described in detail below. For the other elements, only the additional functions and switches are dealt with.

General Information

1. When presetting normals, you do not need to preset any normalized vectors, as the preset vector is automatically normalized.
2. All normals have a meaningful preconfiguration.
3. All tolerances are preconfigured with the general tolerance.

8.4.1 Circle

Set up a new circle.

Click on the function <Circle>.

The dialog window for creating a new circle is displayed:

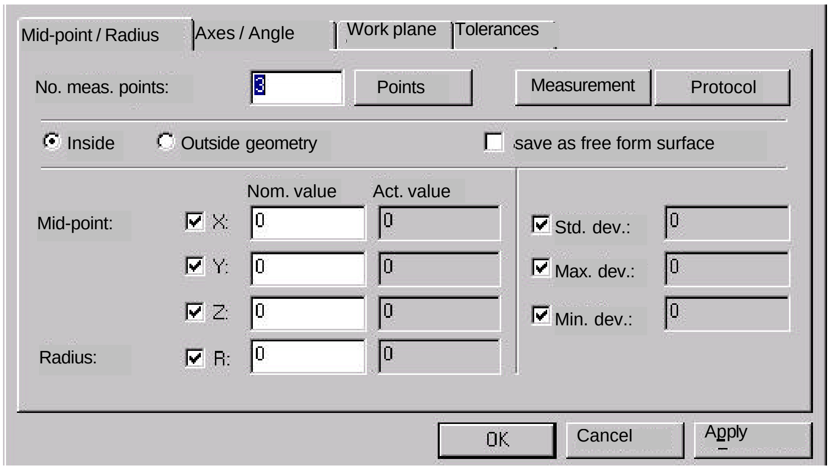


Figure 8-7, Regular geometry CIRCLE 1

You now have two options for defining the circle:

1. Preset the corresponding nominal values.
2. Probe the element manually with the measuring machine. (Element generation). Nominal values will be set during this process.

NOTE

Align the workpiece before generating elements. Without alignment, you cannot make any statement about the position of the element in space, only about absolute dimensions, such as e.g. the radius.

OK: Apply parameters and close dialog.

Cancel: Do not accept parameters and close dialog.

Apply: Apply parameters and do not close the dialog. You must always execute this function before executing further functions, such as measurement, record, points, etc..

Parameters for the mid-point and the radius

No. meas. points: Specify the number of points which are to be used to measure the element.

Points: Define the positions of the measuring points as well as the probe for this element and click on the <Points> function.
A window opens for defining the points:

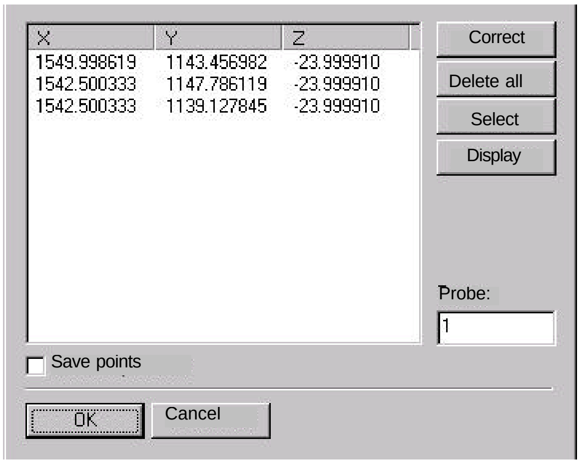


Figure 8-8, List of points

You can now select points on the regular geometry element. The selected points are automatically applied to the points list.
In multiple column operation, you can also select the column which is to perform measurement.

Correct: Remove the last point from the list.

Delete all: Remove all points from the list.

Select: Activate the "POINT ON GEO" selection mode, so that you can select points.

Display: Redisplays all selected points, if the screen content has been deleted by one of the graphic functions.

Probe: Specify the number of the probe that you wish to use to measure the element. The set probe is applied as standard.

Save points: The selected points are used to measure the element. Deactivate this control box if you wish to reject the selected points, so that the measured points are recalculated.

Measurement: If you wish to work with the CNC version, the defined element will be measured in CNC mode, including any working plane to be measured. First you must execute the <Apply> function, so that the set data are applied.

If no measuring points are defined, then the measuring points will be evenly distributed on the circle.

Example:

If there are three measuring points, a measuring point will be set every 120°.

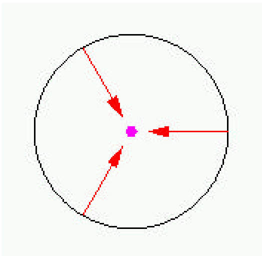


Figure 8-9

Record: A measuring record is generated for the current element, which also contains the set header.
First of all you must execute the <Apply> function, so that the set data are applied.

Save as free form geometry: The regular geometry is generated as free form geometry, i.e. as a surface on which you e.g. can define a regular geometry measuring run.
If the free form geometry is no longer available when you reopen this dialog and confirm with "OK" or "Apply", then it will automatically be regenerated.

Inside, outside geometry: Define whether the circle is an inside or an outside circle. The probe radius will also be corrected accordingly during measurement.

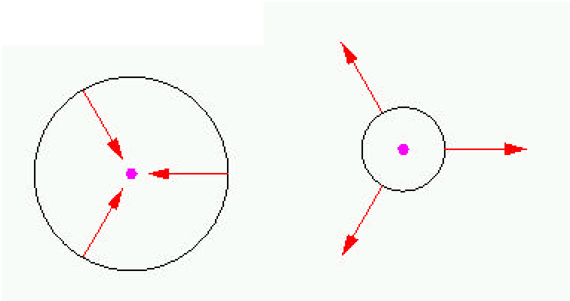


Figure 8-10, Inside circle (left) and outside circle (right)

Using the control box you can define which information is to be output in the measuring record for the element:

Mid-point:	Displays the nominal or actual value of the mid-point.
Radius:	Displays the nominal or actual value of the last measurement or evaluation of the radius.
Std. dev.:	Standard deviation of the last measurement or evaluation.
Max. dev:	Maximum deviation of the last measurement or evaluation.
Min. dev:	Minimum deviation of the last measurement or evaluation.

NOTE

If the element has not yet been measured and has been generated via the analysis, then the fields Std. dev., Max. dev. and Min. dev. will contain the values of the analysis.

Parameters for the axes / angles

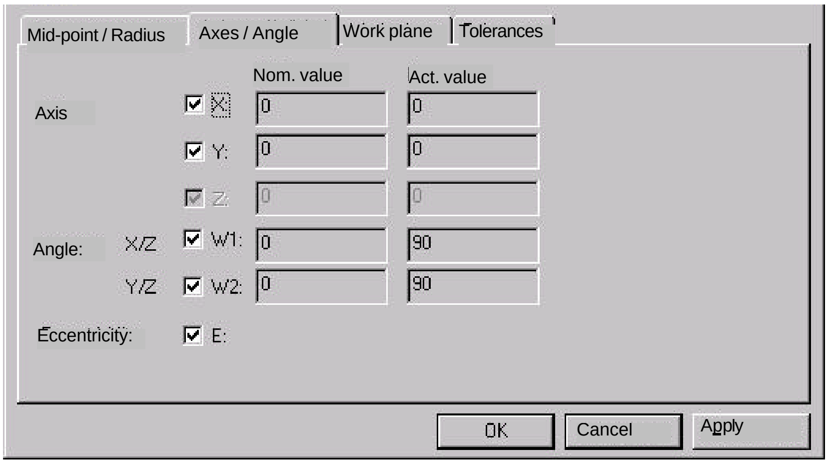


Figure 8-11, Parameters for axes/angles

Axis:	Gives the projection point of the mid-point of the circle onto the main plain of the circle.
Angle:	Gives the angle in space in relation to the main plane.
Eccentricity:	Gives the entire deviation of the circle from the mid-point.

Parameters for the working plane

A working plane must be defined for each two-dimensional element, so that the position is defined in space. The working plane is given in the Hessian normal form, i.e. it is defined by a point (mid-point of the circle) and a normal (U,V,W - vector).

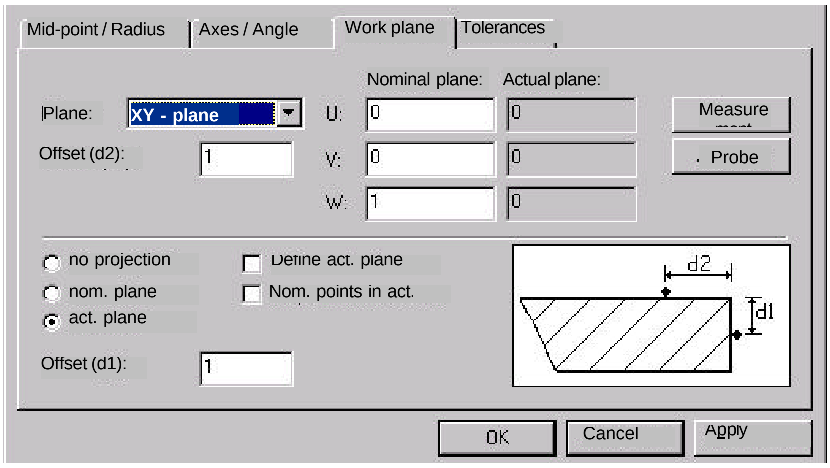


Figure 8-12, Parameters for working plane

Plane: Using the "Plane" control element, you can either link the element - in this case, the circle - with a previously generated plane. The circle is then projected into the plane and the normal of the plane is applied.

or

select one of the main planes. The normal vector is then set for the corresponding main plane (e.g.: XY-plane: U = 0, V = 0, W = 1), and there is no projection into the main plane.

The selection "free" is automatically set, as soon as you specify any normal vector.

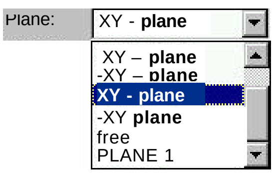


Figure 8-13

Nominal plane: Either preset the normal vector, or the normal vector of the linked plane will be displayed.

Probe: Function for determining the working plane, if you create the element via generation.

Click on <Probe>

Probe the working plane for the element with the measuring machine. As soon as you have probed three points, the working plane is calculated. The circle is projected into this working plane, and the normal of the nominal plane is set. The result is recalculated for each further probing point.

Click on <Probe> again, in order to end generation of the working plane.

Measurement: Measure the working plane in CNC mode if you are working with the CNC version of HOLOS.

Offset (d2): The offset (d2) specifies how far away the measuring point for measuring the working plane is from the circle, taking into account whether the geometry is internal or external.

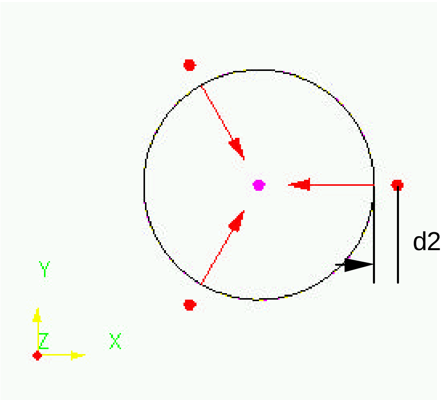


Figure 8-14

Offset (d1): The offset (d1) specifies how far the measuring point for measuring the circle is offset in the opposing normal direction of the working plane.

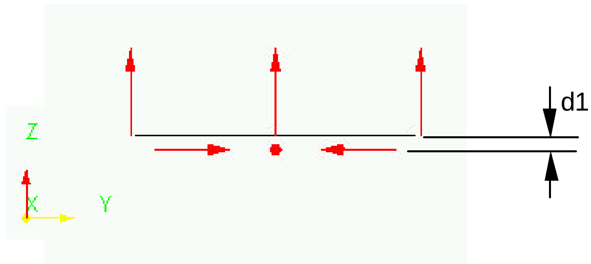


Figure 8-15

No projection: The measuring points of the circle are not projected after the measurement.

Actual plane: The measuring points of the circle are projected into the actual working plane after the measurement (CNC or manual).

Nominal plane: The measuring points of the circle are projected into the nominal working plane after the measurement (CNC or manual).

Define act. plane: Measure the working plane of the element. This can also be measured if a link with a "plane" element is present.
The control box can and must only be deactivated if a link with a "plane" element is present.

Nominal points in actual plane: This selection takes into account a modified normal direction and the bounce back in the case of bent or sprung metal parts. The measuring points are projected into the measured nominal plane, and the offset (d1) is still taken into account.

Parameters for tolerances

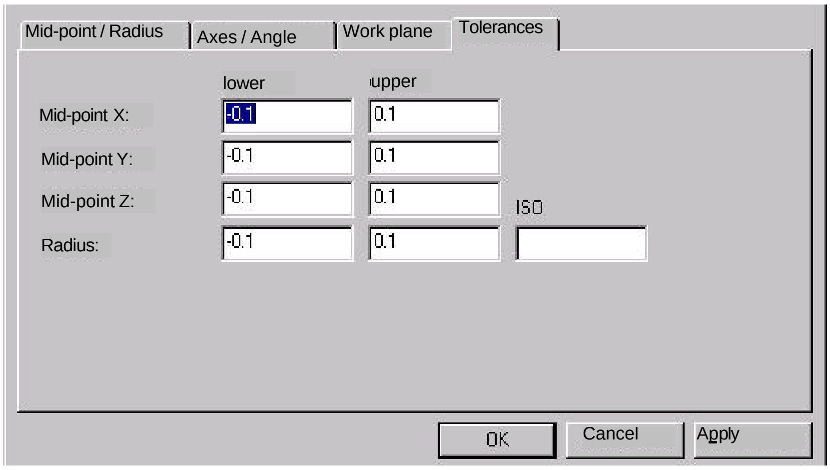


Figure 8-16

Mid-point X, Y, Z: Tolerance for the mid-point coordinates

Radius: Tolerance for the diameter of the circle

ISO: Enter the tolerance for the circle in accordance with DIN ISO 286 Part 2. The upper and lower tolerance for the diameter is calculated as soon as you click on any other field.

NOTE

Make sure that the tolerance is acceptable and possible for your defined geometry. It is not automatically checked whether you have made entries for an inside geometry (e.g. borehole) or an outside geometry (e.g. shaft). You could therefore specify h7 as the tolerance for a defined inside geometry.

8.4.2 Plane

Planes are always managed in the Hessian normal form: a plane is defined by a point and a normal. It always has an infinite extent, even if it is graphically represented as a limited surface.
During manual measurement it is therefore not checked whether the point lies within the delimitation. If elements are defined which use the plane as a working plane, then the elements do not have to lie within the delimitation.
However, measuring points for CNC measurement can only be defined within the delimitation.

Parameters for the plane

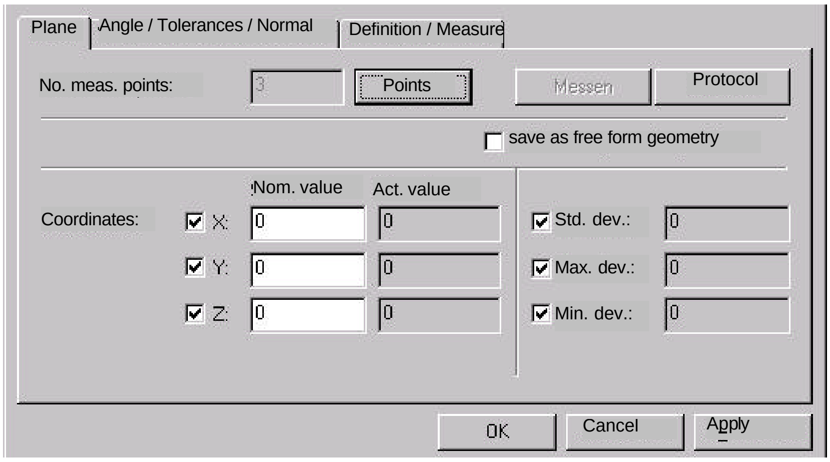


Figure 8-17, Parameters for the plane

Coordinates: Any point on the plane. In order to specify the plane by values, select the "Definition / Measure" register tab.

Regular geometry

Parameters for angles, tolerances and normal

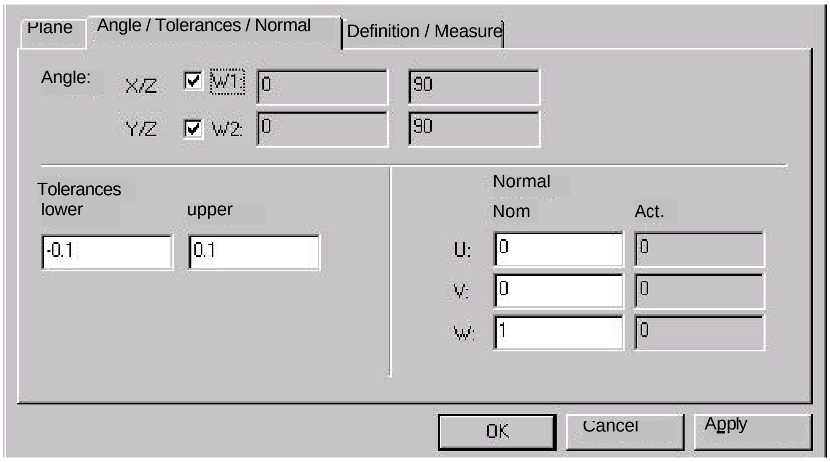


Figure 8-18, Parameters for angles, tolerances and normal

Parameters for definition and measurement

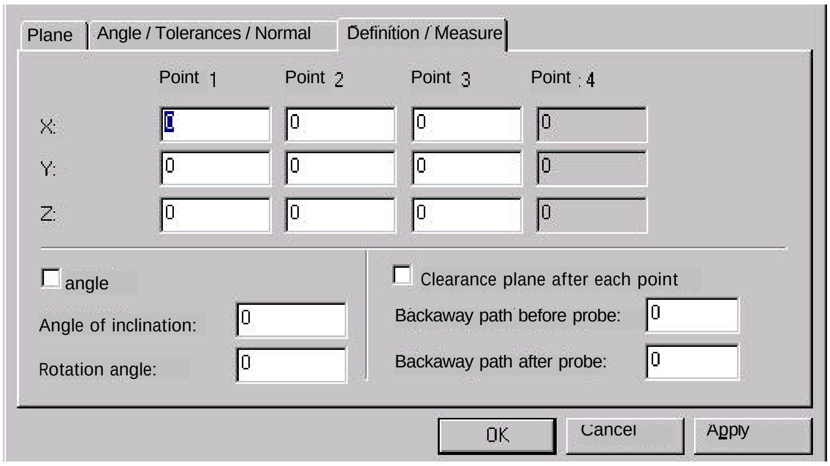


Figure 8-19, Parameters for definition and measurement

Point 1 - 4:	Delimitation points of the plane for graphic representation. In order to generate the plane using preset values, specify three points. The fourth point will be determined automatically.
Angle:	In order to define the plane via the inclination and rotation angles, select the control box. Both angles are only entered when defining the plane. If you reopen the plane element later, the values will be zero again.

Angle of inclination:	Angle of inclination around the Y/Z-plane.
Rotation angle:	Rotation angle around the Z-axis.

The following parameters are only required for CNC measurement of the plane and also only apply during measurement of this plane. For all subsequent measurements, the global parameters of the measuring run will then apply again.

Clearance plane after each point:	With this option, the coordinate measuring device moves into the clearance plane after each probing point.
Backaway path before probing:	Distance of an intermediate position from the target position. The intermediate position is located before probing, and is vectorially probed starting from it.
Backaway path after probing:	Distance of an intermediate position from the target position. The intermediate position is located after probing, and is vectorially moved away up to it.

8.4.3 Point

Geometrical points only require a surface if they are to be projected onto the workpiece during definition. You can define all other points in space at will; for these, the actual distance between nominal and actual point is determined.

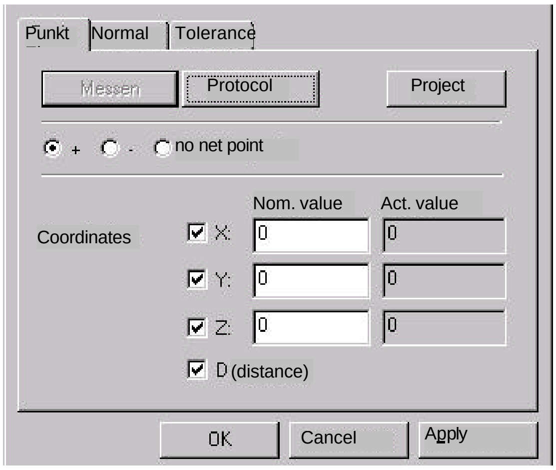


Figure 8-20, Regular geometry - POINT 1, Point

Parameters for the point

Coordinates: Coordinates of the point.

Project: The point with specified straight lines is projected onto the workpiece, in order to determine a point on the workpiece.

no net point: No net point is generated during projection of the point.

+ / -: If a coordinate is left e.g. on "Z", then the point will be projected from the positive (+) or negative (-) Z-axis onto the workpiece from above.
Accordingly: X = 0; positive X-axis; Y = 0; positive Y-axis

Enter the coordinates.
Choose between +; - and "no net point" and click on "Apply".
Click on Project.

The coordinates X, Y and Z now contain the values of the projected point.

Parameters for the normal

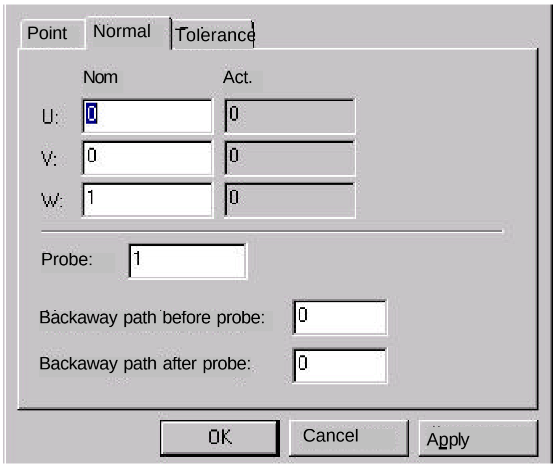


Figure 8-21, Regular geometry - POINT 1, Parameters for the normal

In multiple column operation you can also select the column which is to perform measurement.

U, V, W: Normal of the point.

Probe: Probe used to measure this point.

Backaway paths: See plane element

Parameters for tolerances

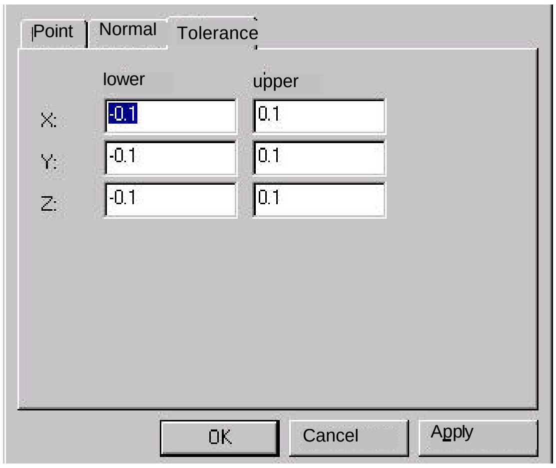


Figure 8-22, Regular geometry - POINT 1, Parameters for the tolerances

See circle element.

8.4.4 Slot

A slot can be specified via two methods:

1. By entering both mid-points of the circle
2. By entering the mid-point, the length and the width of the slot.

Parameters for the dimensions

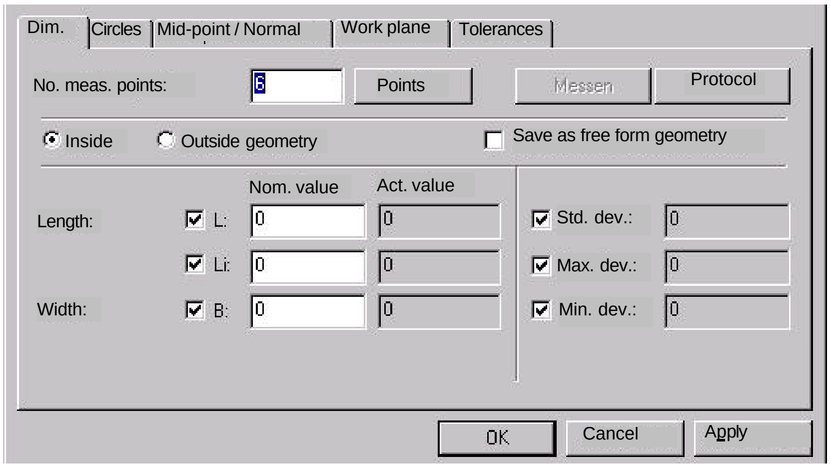


Figure 8-23, Regular geometry - Slot 1, Parameters for dimensions

Length/Width: L = Li + B

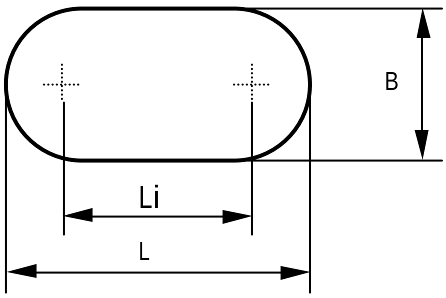


Figure 8-24

Parameters for circles

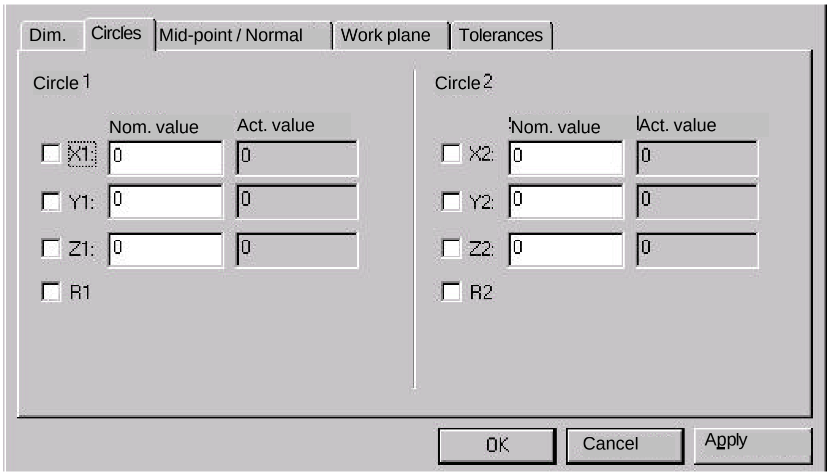


Figure 8-25, Regular geometry - Slot 1, Parameters for circles

Circle 1: Mid-point of the first circle
Circle 2: Mid-point of the second circle

Parameters for mid-point and the normal

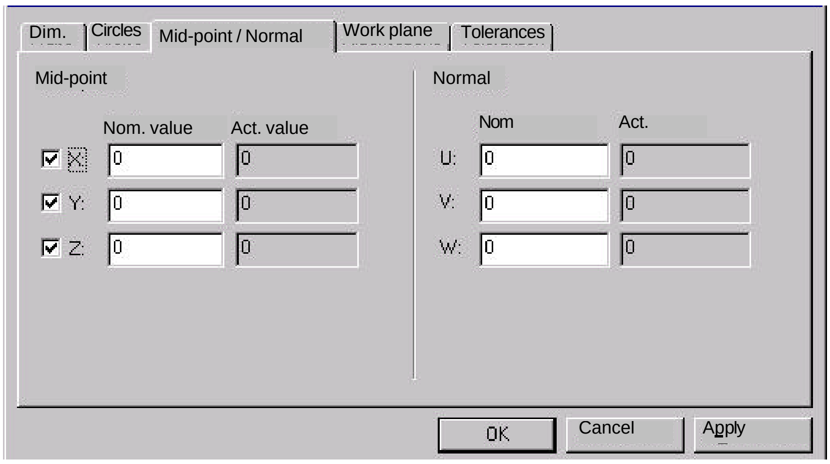


Figure 8-26, Regular geometry - Slot 1, Parameters for normal and mid-point

Mid-point: Mid-point of slot.

Normal: The normal of the slot corresponds to the normalized vector from the mid-point of the first to the mid-point of the second circle.

Parameters of the working plane

Correspond without exception to the work plane of the circle.

Parameters for tolerances

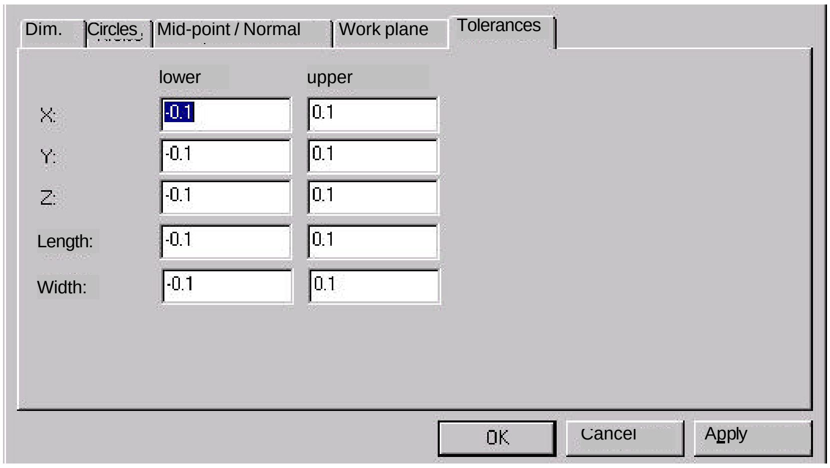


Figure 8-27, Regular geometry - Slot 1, Parameters for tolerances

X,Y,Z: Tolerance for the X-,Y-,Z-values for circle 1; circle 2 and the mid-point.

Length: Tolerance for the length data.

Width: Tolerance for the width data.

8.4.5 Rectangular hole

Parameters for the center and the size

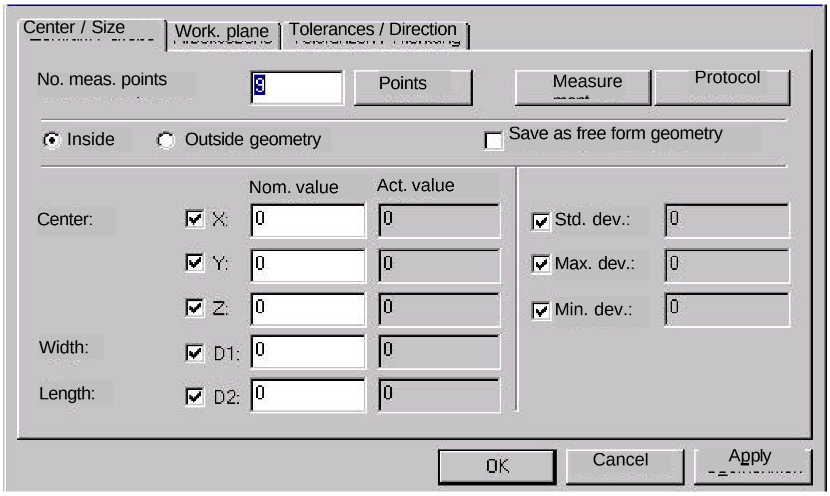


Figure 8-28, Regular geometry - RHOLE 1, Parameters for center and size

Center:	Mid-point of rectangular hole.
Width:	Width of rectangular hole.
Length:	Length of rectangular hole.

Parameters for the working plane

Correspond without exception to the working plane of the circle.

Regular geometry

Parameters for tolerances and direction

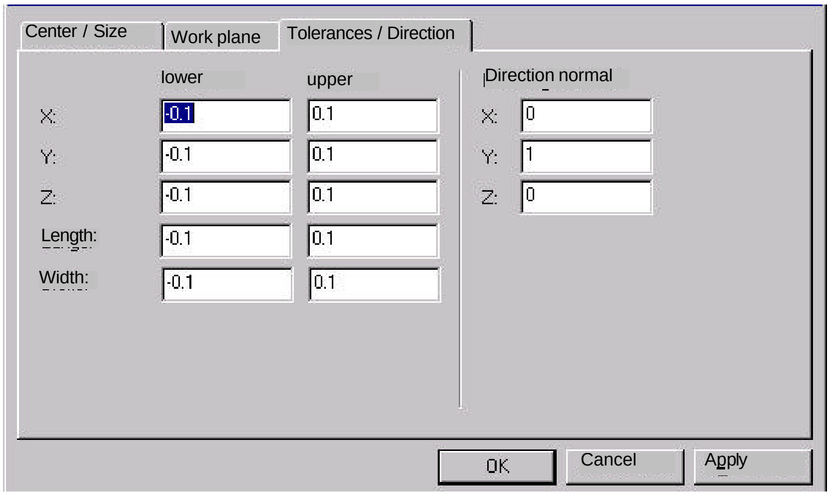


Figure 8-29, Regular geometry - RHOLE 1, Parameters for tolerances and direction

X,Y,Z: Tolerance for the center of the rectangular hole.

Length: Tolerance for the length of the rectangular hole.

Width: Tolerance for the width of the rectangular hole.

Direction normal:

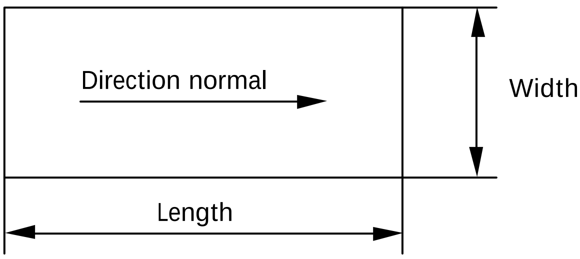


Figure 8-30

8.4.6 Hemisphere

With the hemisphere it is possible to measure circles, slots or rectangular holes, if the probe diameter is greater than the circle diameter.

Parameters for the mid-point / hemisphere

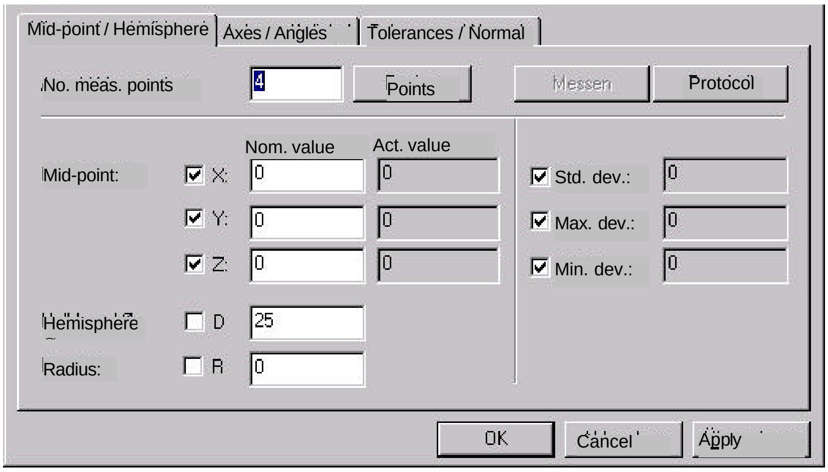


Figure 8-31, Regular geometry - HEMISPHERE 1, Parameters for the mid-point/hemisphere

Mid-point: Mid-point of the hemisphere which is to be measured.

Hemisphere ∅: Diameter of the hemisphere.

Radius: Only for information purposes, e.g. to mention the radius of the circle to be measured in the record.

Parameters for axes and angles

Correspond without exception to the axes and angles of the circle.

Parameters for tolerances and normal

See working plane and tolerances for the circle.

Note

The plane only needs to be defined if you require information about the axes and angles. If no plane is defined, then the axes and angles will not be evaluated.

8.4.7 Sphere

Parameters for the mid-point and the radius

Correspond without exception to the parameters of the circle.

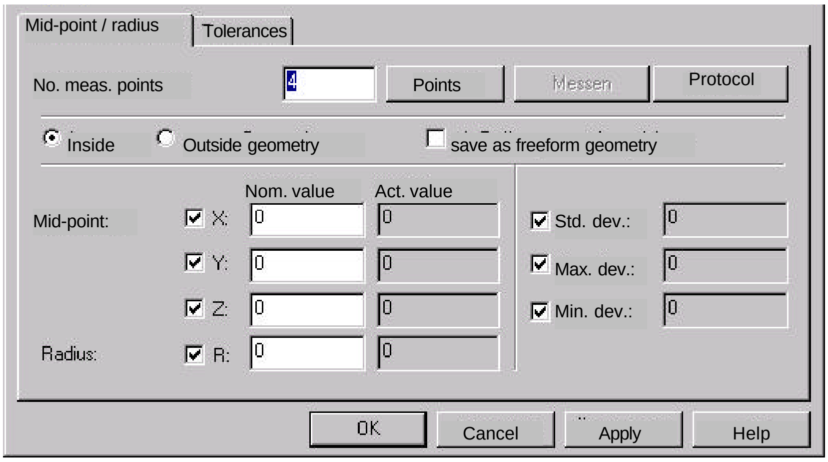


Figure 8-32, Regular geometry - SPHERE, parameters for mid-point and radius

Parameters for tolerances

Correspond without exception to the parameters of the circle.

8.4.8 Cylinder

Parameters for the center and the size

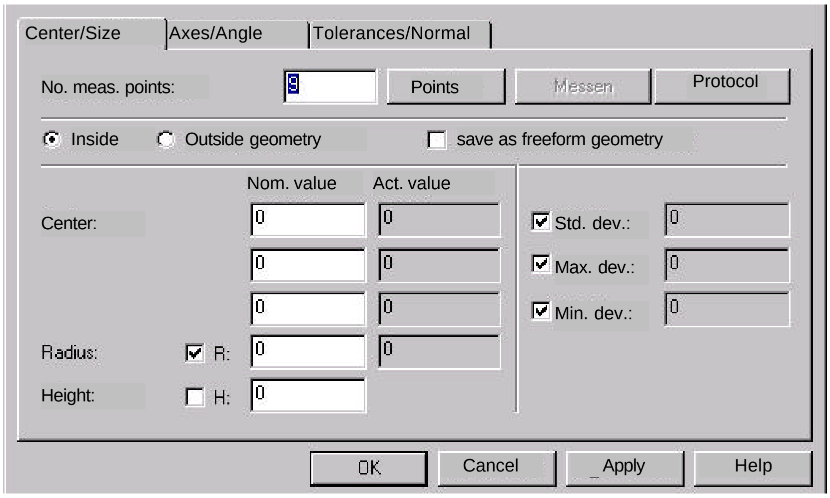


Figure 8-33

See mid-point and radius of the circle.

Center	Mid-point (Z) of the cylinder
Radius	Radius of the circle
Height	Height of the cylinder

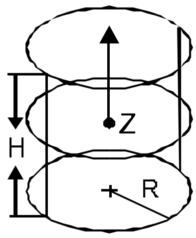


Figure 8-34

Parameters for axes and angles

See axes and angles of the circle.

Parameters for tolerances and normal

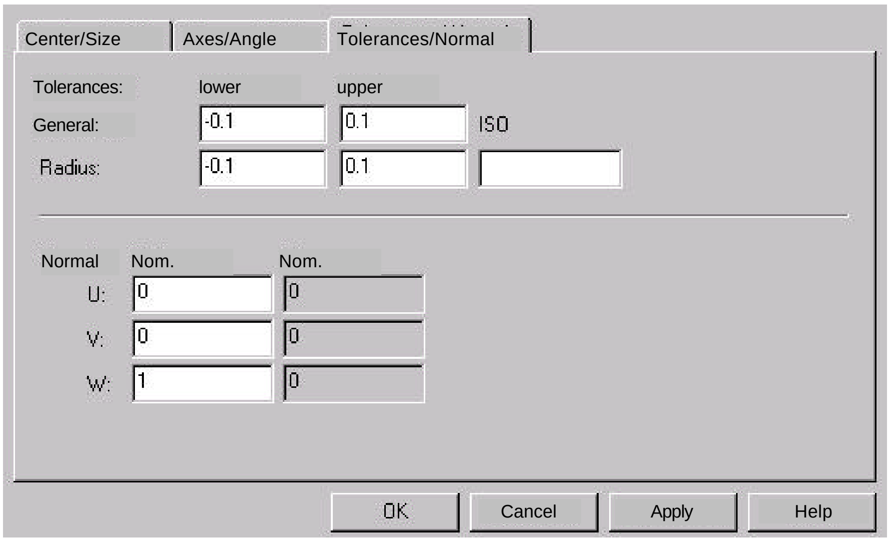


Figure 8-35

General:	Tolerances of the angle in space in relation to the main plane.
Radius:	Tolerance for the diameter of the circle.
ISO:	Enter tolerance for the cylinder in accordance with DIN ISO 286 Part 2. The upper and lower tolerance for the diameter is calculated as soon as you click on any other field.
Normal	Direction in u, v, w to define the position in space

8.4.9 Cone

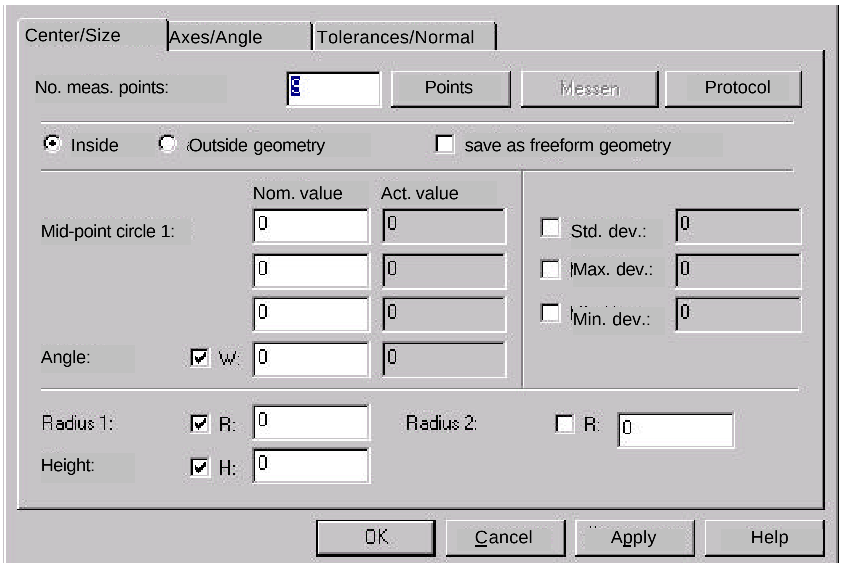


Figure 8-36

See mid-point and radius of the circle.

The cone is defined by three or four possible specifications: radius of circle one (R1), radius of circle two (R2), the angle (W) and the height (H).
Which circle is one and which is two depends on the direction.

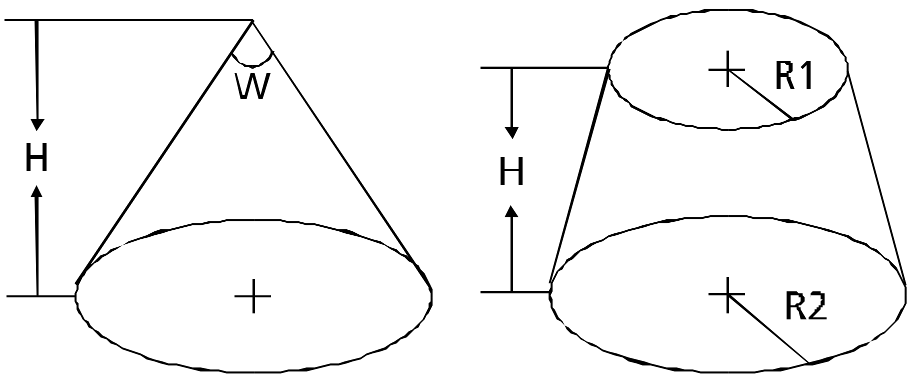


Figure 8-37

Parameters for axes and angles

See axes and angles of the circle.

Parameters for tolerances and normal

See axes and angles of the cylinder.

8.4.10 Intermediate position

For HOLOS CNC you can define movement paths within a regular geometry list, with the help of intermediate positions.
You can also perform a probe change at an intermediate position, and in the case of a DSE machine, a probe swivel is also possible.

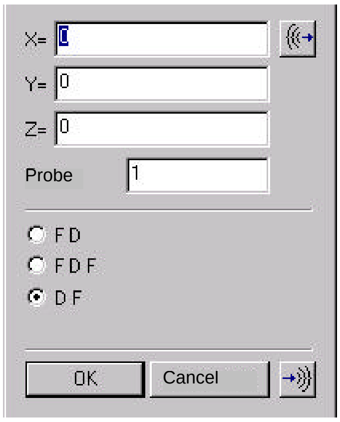


Figure 8-38

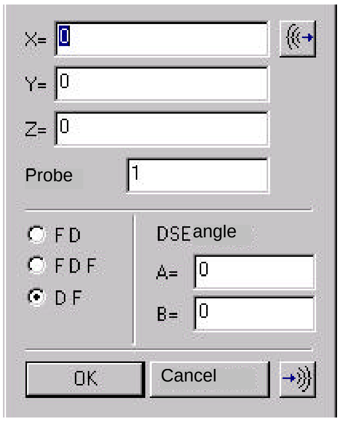


Figure 8-39

In multiple column operation you can also select the column which is to perform measurement.

X, Y, Z:	Intermediate position in workpiece coordinates.
Probe:	Probe number which you wish to change to.
FD (Move, Rotate):	First the intermediate position is moved to, then the probe is changed.
FDF (Move, Rotate, Move):	First the intermediate position is moved to, then the probe is changed and then the intermediate position is moved to again.
DF (Rotate, Move):	First the probe is changed and then the intermediate position is moved to. With a DSE machine, probe changing also means DSE rotation.
DSE angle (A,B):	Angle to which the DSE is rotated.
	Move to intermediate position and change probe.
	Accept current probe position and current probe from the coordinate measuring device. With a DSE machine, the angles of the DSE are also then accepted.

8.4.11 Comment

The "comment" is not a regular geometry element in the actual sense. It fulfils the following functions:

- In the first block, a comment can be entered for the measuring record.
- The other options serve to interrupt a CNC run, to amend it manually and then to record it again.

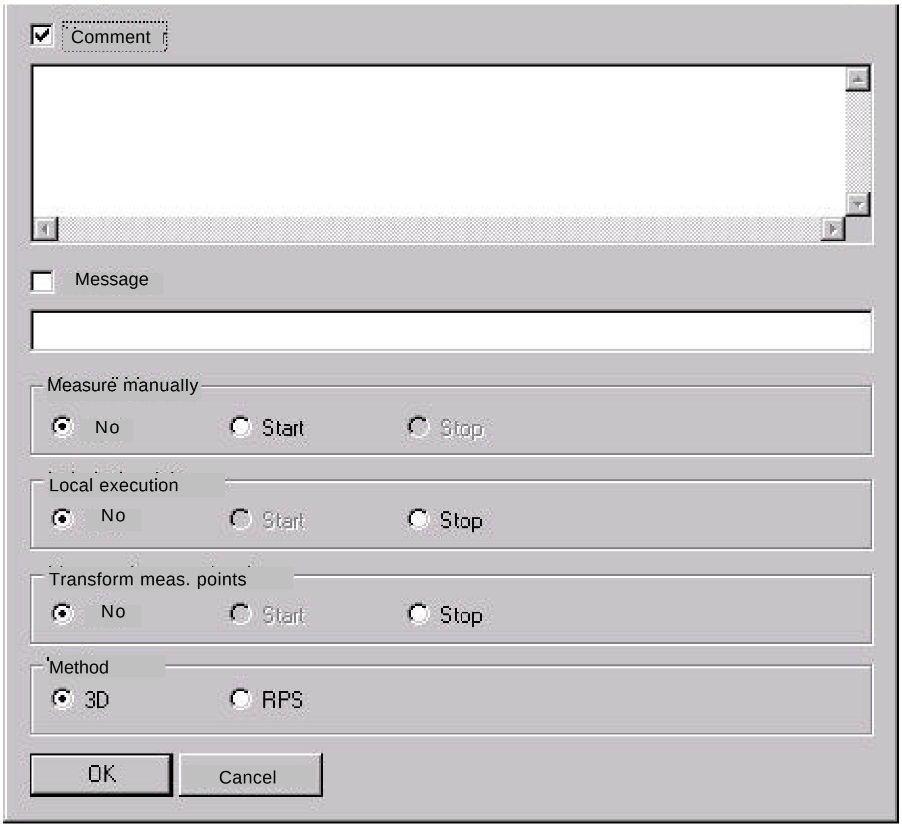


Figure 8-40

Comment — Switch the option on and enter a comment, which will be output in the measuring record.

Message — Switch the option on and enter a message which defines a stop in the CNC run and is displayed on the monitor. E.g. you can use it to indicate at an intermediate position that a clamping fixture needs to be removed, before further measurement can be performed.

If no entry is made, but the option is switched on at the same time as the "Manual measurement" option, then at this point in the process you will be asked whether manual measurement is to be performed.

manual measurement	If e.g. in a drilling pattern the position of the elements in relation to each other is correct, but the entire pattern can be moved depending on the workpiece, then you can interrupt the CNC run with this option and measure one or more elements manually. Then switch the option off again with "Stop" and the CNC run is continued. Before the CNC run is continued, you can define what should occur with the manual measurement.
	Select an option:
local alignment	Effects a temporary change in the workpiece system. Select the elements for the best-fit. Or
transform measuring points	Moves the measuring points within the workpiece system.
	For both options, you then define the method:
method	You can choose between 3D- and RPS-alignment method.

🕮 For alignment method, see Chap. 3.6

8.4.12 Linking

There are two ways of linking two elements or links with each other:

1. With the <Elements><Link> function
2. Directly select the required link e.g. point, distance, angle or straight line.

The link function allows you to quickly generate several links one after another:

Click on the link function.

You can now select two elements or links.

After selecting the second element or the second link, the window for links is displayed:

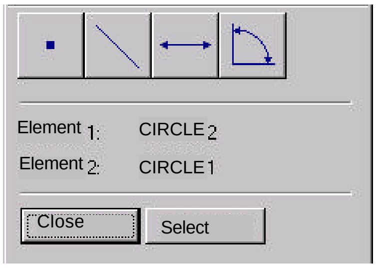
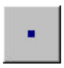


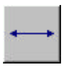
Figure 8-41, Regular geometry - Links

Select a link:

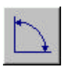


Generate the point link.

Generate the straight line link.



Generate the distance link.



Generate the angle link.

Generate the circle link.

Generate the plane link.

The link is generated with elements 1 and 2, the selected elements or links, and entered into the link list.
In order to be able to select further elements or links, click on the "Select" key.

When an element or a link is selected, the next selectable one is always marked with an asterisk, and you can generate a further link with other elements.
When you have generated all required links, close the window.

The following table provides an overview of the possible links.
The elements circle, slot, rectangular hole and hemisphere are always treated like a point, i.e. the mid-point of the element always has a further link. Cylinder and cone are further linked as a straight line (middle line).
The angle and distance links as well as the intermediate position element cannot be linked further.

Element	**Element**	**Link**	**Linking task**
Point	Point	Point	Mid-point between two points
Straight line	Straight line	Point	Point of intersection of two straight lines
Straight line	Plane	Point	Point of intersection of a plane and a straight line
Point	Straight line	Point	Perpendicular root point of a point on a straight line
Point	Plane	Point	Perpendicular root point of a point on a plane
Plane	Plane	Straight line	Intersecting straight line of two planes
Straight line	Straight line	Straight line	Middle straight line between two straight lines
Straight line	Straight line	Angle	Angle between two straight lines
Plane	Straight line	Angle	Angle between straight line and plane
Plane	Plane	Angle	Angle between two planes
Point	Point	Distance	Distance between two points
Straight line	Straight line	Distance	Distance between two straight lines
Point	Straight line	Distance	Distance of a point from a straight line
Point	Plane	Distance	Distance of a point from a plane
Point	Straight line	Plane	Clamp plane
Straight line	Straight line	Plane	Clamp plane
Plane	Plane	Middle plane	Clamp plane
Point	Point Point	Circle	

Figure 8-42

8.4.13 Distance

The <Distance> function generates a distance link.

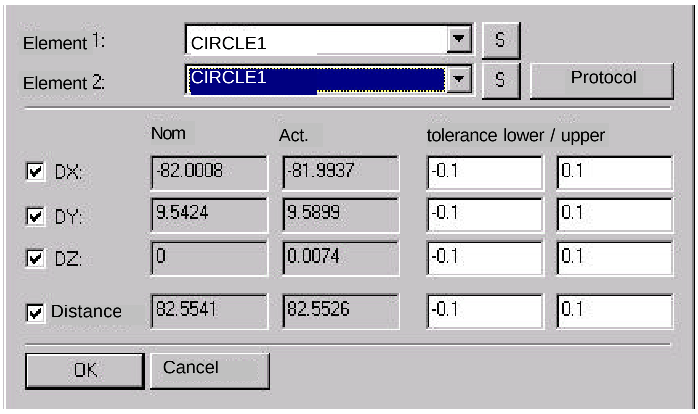


Figure 8-43, Regular geometry - link - distance

Element 1 and 2:	Elements which are linked to the distance link.
	Activate the selection mode, through which the next element or the next link in the graphic is to be selected as the element to be linked.
Record:	Evaluate link and generate a record for the link.
DX, DY, DZ:	Nominal and actual value of the main axial components are determined from the elements. You can only preset the tolerances for the corresponding component.
Distance:	Distance of the nominal point from the actual point in space.

8.4.14 Straight line

The <straight line> function generates a straight line link.

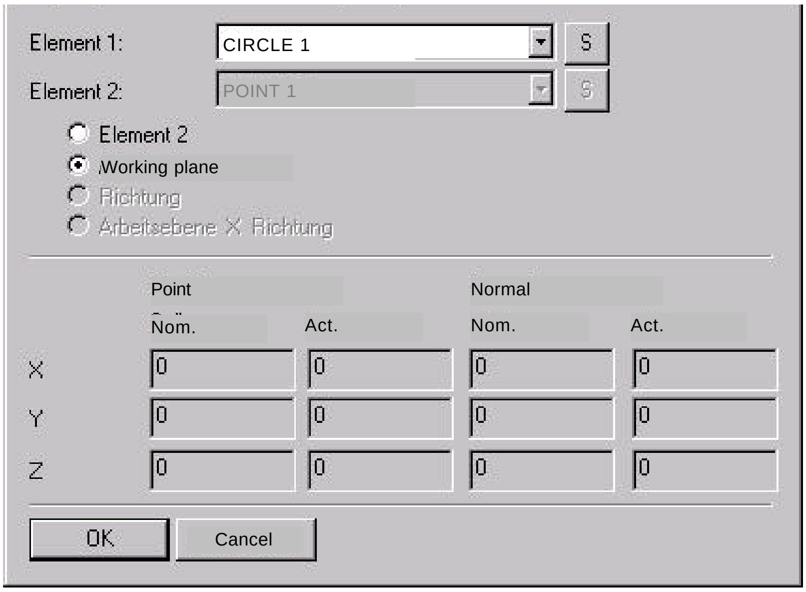


Figure 8-44, Regular geometry - link - straight line

Selection of the link type:

Element 2	Forms the straight line between two elements
Work plane	Only for circle, oblong and rectangular hole. The straight line is formed from the mid-point and the normals of the work plane.
Direction	Only for oblong and rectangular hole. Forms the straight line from the mid-point and the direction of the element.
Work plane X-direction	Forms the straight line as a cross-product of the work plane and the direction of the element.

Normal: The normal results from the order of the selected elements. It always points from <Element 1> to <Element 2>.

For further parameters, see distance link.

There is no measuring record for straight line links, but the straight line is mentioned for information in the record.

8.4.15 Point

The <Point> function generates a point link.

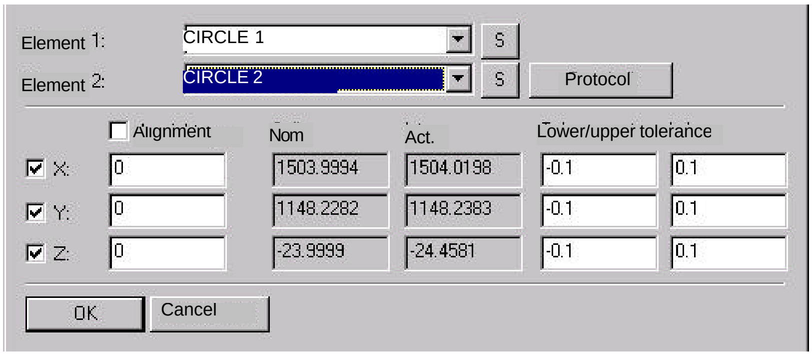


Figure 8-45, Regular geometry - link - point

Alignment: As a rule, the nominal and actual point are used for alignment. If no data set is available, but the coordinates of a theoretical point are known, you can preset a nominal point here especially for the alignment.

Example:

It is known that the point of intersection of three planes lies exactly on the zero point.

Generate the three planes.

Form two intersecting straight lines between the planes.

Form the point of intersection from the two intersecting straight lines.

Activate the "Alignment" control box in the point of intersection and preset the coordinates X = 0; Y = 0; Z = 0.

The coordinates of two further points must also be known.

Now you can align the part.

For describing further parameters, see distance link.

8.4.16 Angle

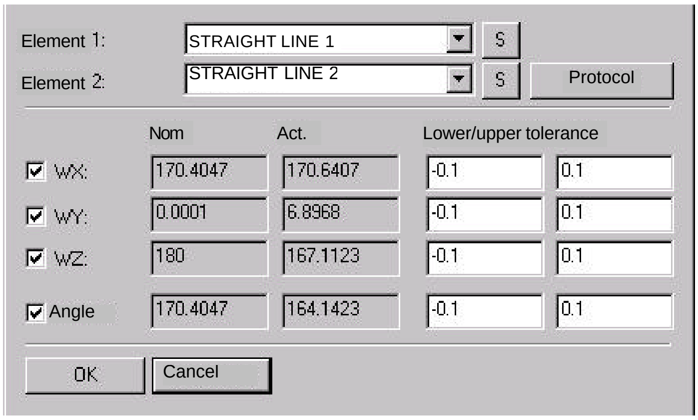
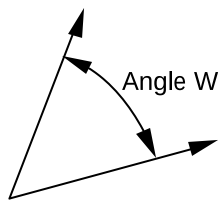


Figure 8-46, Regular geometry - link - angle

WX:	Angle of the projected vectors onto the YZ - plane.
WY:	Angle of the projected vectors onto the XZ - plane.
WZ:	Angle of the projected vectors onto the XY - plane.
W:	Angle in space.

The normal directions of the elements to be linked are decisive for the result, as the angle enclosed by the normals is calculated:



For describing further parameters, see distance link.

8.4.17 Circle

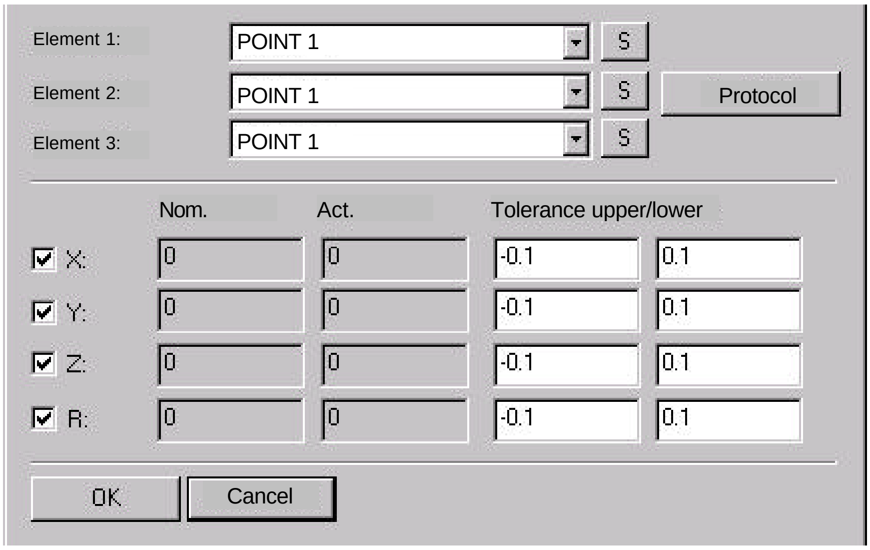


Figure 8-47

A circle is calculated from three points: coordinates of the mid-point and radius

8.4.18 Plane

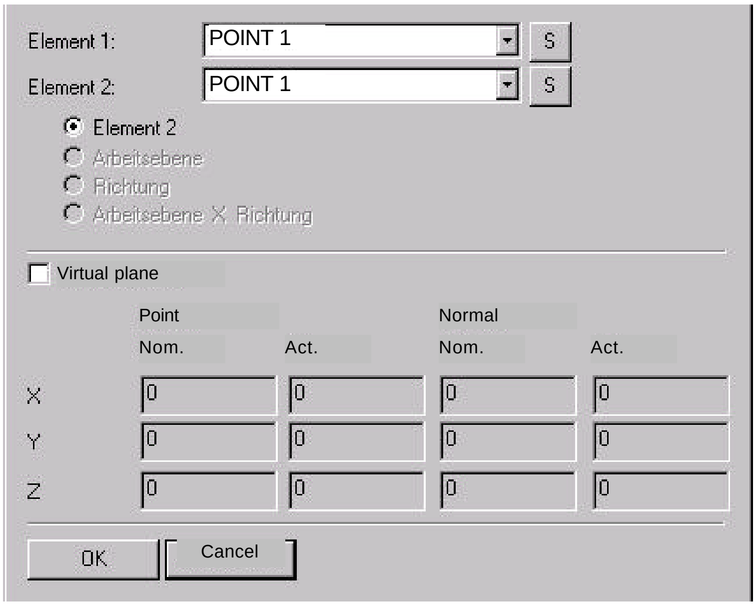


Figure 8-48

Selection of the link type:

Element 2	Forms the plane between two elements
Work plane	Only for circle, oblong and rectangular hole. The plane is formed from the mid-point and the normals of the work plane.
Direction	Only for oblong and rectangular hole. Forms the plane from the mid-point and the direction of the element.
Work plane X-direction	Forms the plane as the cross product of the work plane and the direction of the element.

Virtual plane: No existent plane, but a defined plane is used, e.g. a symmetry plane.

Preselect the coordinates of one point and three normals for this purpose.

8.4.19 Transformation

Regular geometry elements and their links can be moved, rotated or mirrored.

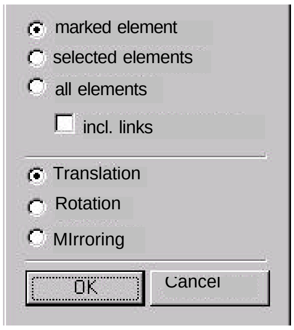


Figure 8-49, Transformation through translation

marked element:	Only the marked element is transformed.
selected elements:	All selected elements are transformed.
all elements:	All elements contained in the list are transformed.
incl. links:	The selected links are transformed.

8.4.20 Rotate normal

The element is rotated by 180°.

8.4.21 Element as hemisphere

Circle, oblong and rectangular hole can be calculated as a hemisphere with the help of a mid-point and the normals.

8.5 Measurement

The functions of the <Measurement> menu serve mainly for the measurement of regular geometry elements.

Functions:

- Measurement (only HOLOS CNC)
- Selected measurement (only HOLOS CNC)
- Man. measurement
- Selected man. measurement
- Simulation (only HOLOS CNC)
- Cancel measuring run (only HOLOS CNC)

8.5.1 Measurement

With this function the complete regular geometry list is measured. The following must be observed:

1. The regular geometry list must be saved before measurement.
2. If a measurement is active, you cannot start any other measurement at the same time.
3. Each element, as well as any working plane present, is treated as an independent measuring run. Therefore, the coordinate measuring machine moves into the clearance plane after each element.
4. The parameters "Backaway path before and after probing" and "Clearance plane after each point" are provided with each element. If you have set global values which cannot be used to measure an element, then the parameter is automatically corrected.

   Example:
   You wish to measure an inner circle with diameter 8 mm and have set a backaway path before probing of 10 mm: The backaway path is automatically reduced to 4 mm, so that the circle can be measured without problem.

Example:
You wish to measure a hemisphere with diameter 30 mm and have set a backaway path before probing of 10 mm: The backaway path is automatically increased to 17 mm, so that you can measure the hemisphere, without having to move onto the clearance plane.

5. The parameter "Clearance plane after each point" can only be changed for the plane element. Otherwise, the global setting always applies.
6. The measuring points are corrected according to the set offset value and the normals of the element.
7. The actual data are saved after the measurement.

8.5.2 Selected measurement

Corresponds to the <Measurement> function, but only the selected elements are measured.

8.5.3 Man. measurement

With the <man. measurement> function you can measure the entire regular geometry list by manually probing the elements.

Select the <man. measurement> function.

The assistant for manual measurement is displayed:

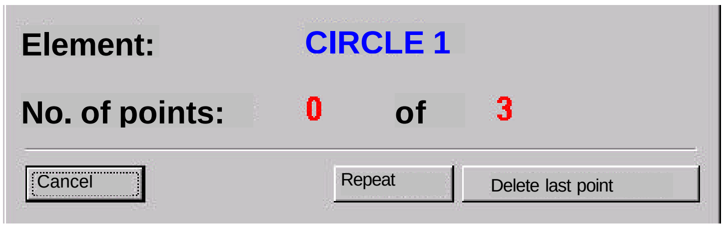


Figure 8-50, Manual measurement

Element: Displays the name of the element to be measured. The element is also selected at the same time. If the name of the element ends with "AE", this means the working plane of the element and you must probe the working plane of the element.

No. of points: Shows the number of points already probed, as well as the total number of points to be probed. If you have reached the number of points to be probed, the assistant automatically changes to the next element.

Cancel: Cancel the entire manual measurement.

Repeat: Repeat the measurement of an element. If you have already probed points for the current element, these points are rejected and you can perform a new probing.
If the number of already probed points is zero, then you switch automatically to the previous element and you can measure it again.

Delete last point: If you have for some reason probed at the wrong point, then you can immediately reject this point again. If you have probed the last point of an element incorrectly, then you must repeat the entire element again.

8.5.4 Selected man. measurement

Like the man. measurement function, but only the selected elements are displayed by the assistant.

8.5.5 Simulation

Each regular geometry measurement which you start in CNC mode is only graphically simulated and is not sent to the coordinate measuring machine.

This applies e.g. for the <Measurement> and <Man. measurement> functions, for regular geometry measurement from a macro run, as well as for measurement via the function keys in the element dialogs.

8.5.6 Cancel measuring run

Cancel the CNC measurement of a regular geometry list.

8.6 Evaluation

The functions of the <Evaluation> menu serve mainly for subsequent evaluation of existing actual data.

Functions:

- Evaluation
- Evaluate list
- Evaluate sel. list
- Measuring record
- RPS alignment
- 3D alignment

8.6.1 Evaluation

You can subsequently evaluate earlier measurements of the loaded regular geometry list. The evaluation is graphically displayed, and a measuring record is generated.

Call up the <Evaluation> function.

A window opens for evaluating the existing actual data:

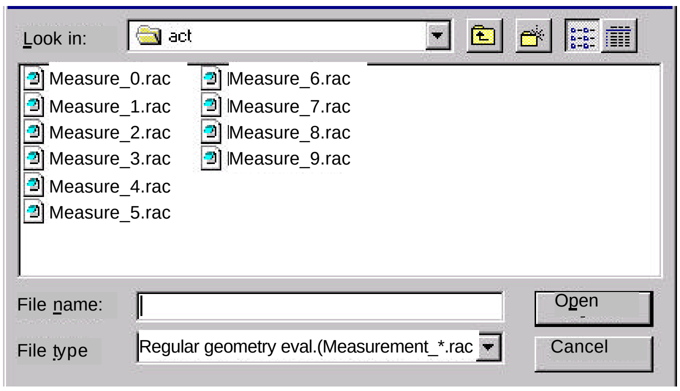


Figure 8-51, File name for evaluation

The actual data of a measurement are always saved under the name of the regular geometry list with consecutive numbering.

Select a file and click on <Open>.

The data are evaluated, and the result is displayed on the screen:

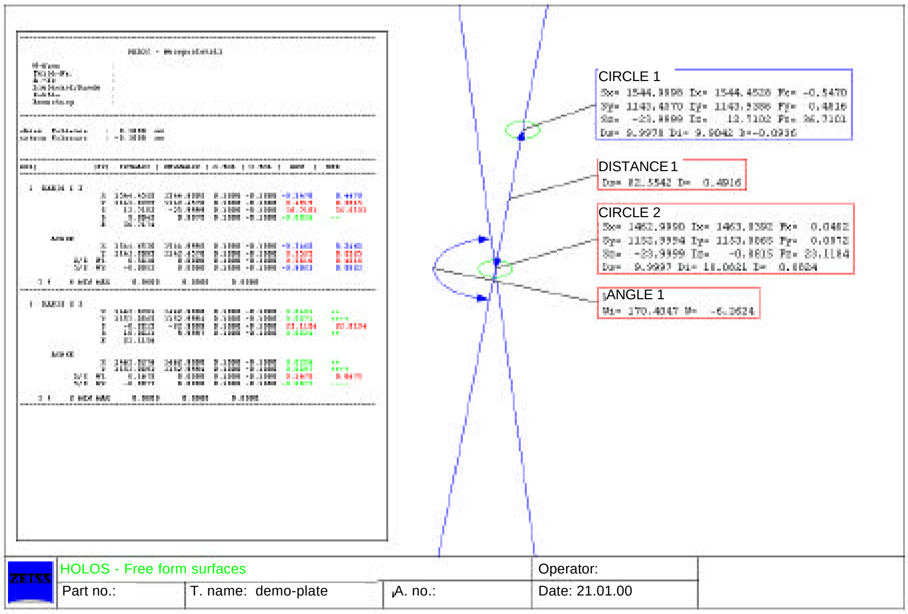


Figure 8-52, Actual data of a measurement

If elements have been removed from the regular geometry list or renamed, then they will not be evaluated. If there are more elements in the regular geometry list than are saved in the actual file, then the evaluated elements will automatically be selected.

8.6.2 Evaluate list

The currently loaded regular geometry list is evaluated with its actual values. The data are graphically evaluated, and a measuring record is generated.

This function is helpful if you have generated a regular geometry list with elements, then measure these and subsequently only generate the links. You can then evaluate the elements and the links from the list.

8.6.3 Evaluate sel. list

Like the <Evaluate list> function, but only the selected elements and links are evaluated.

8.6.4 Measuring record

Like the <Evaluation> <Measuring record> function in the HOLOS main menu. The record of the last evaluation (regular geometry or free form geometry evaluation) appears.

8.6.5 RPS alignment

With the RPS alignment procedure, you can align a workpiece at defined points. The nominal planes selected for the alignment may only change within the range of the measuring accuracy, or when repeating measurement of these points.

⇘ Define the elements or links for RPS alignment. The plane and intermediate point elements cannot be used, and only the point of the links can be used.

⇘ Check the normals of the elements. These are important for the offset correction and the probe radius correction, particularly with elements whose working plane is being measured.

⇘ In the element window select the corresponding axes of the elements for which the nominal value to actual value variance is to be set to zero.

Please note the following points:

1. You need at least six nominal planes, i.e. each selected axis corresponds to a nominal plane.
2. Use at least three elements.
3. Distribute the six nominal planes according to the 3-2-1 procedure, e.g. three nominal planes for the Z-axis; two for the Y-axis and one for the X-axis.
4. Where possible, only select elements for an axis that are a short distance from each other in this axis, or check the deviation of the distance of the elements used in this axis. Too great a deviation can lead to unintentional rotation of the workpiece.

   Example:

You are measuring the following points:

	Point 1	Point 2
X - nominal value	200	400
Y - nominal value	100	100
Z - nominal value	25	25
X - actual value	1200.56	1399.47
Y - actual value	1100.45	1100.23
Z - actual value	125.25	125.25

Figure 8-53

If you select both points for RPS alignment in X and have only defined six nominal planes altogether, then you will get a rotation around the Y-axis, due to the deviation of the distance of 1.09 in the X direction.

Perform the <Evaluation><RPS alignment> function.

The best fit is performed, and the result is displayed in a window:

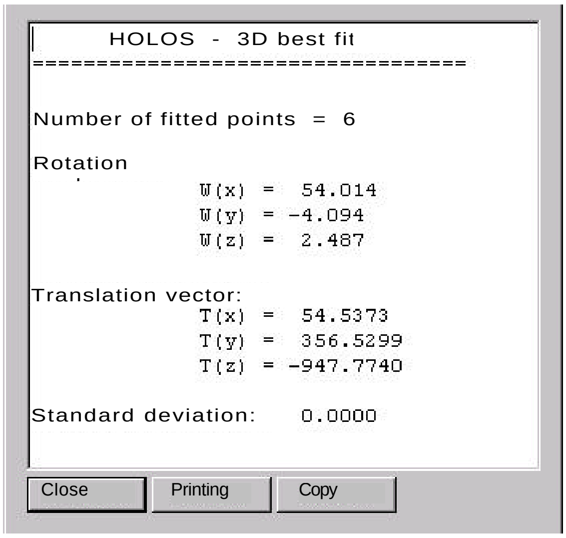


Figure 8-54, 3D best fit

If the number of fitted points is equal to six and the standard deviation is not zero, then check the above mentioned points again. Close the function.

If you answer <Yes> to the following query, the stored workpiece system will be corrected.

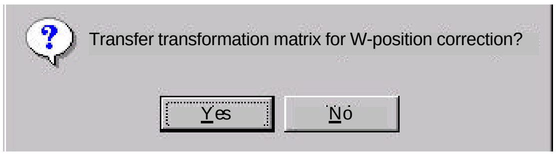


Figure 8-55, Transfer transformation matrix?

If you answer <Yes> to the next query, the measured data will be transformed.

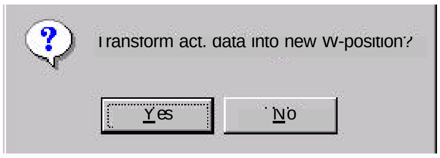


Figure 8-56

In order to re-evaluate the transformed values, use the <Evaluation><Evaluate list> function.

8.6.6 3D alignment

3D alignment is similar to the 3D best fit with free form geometry measurements. However, here the points (mid-point in the case of circle, slot, etc.) of the elements can be used.

Define the elements or links for the 3D alignment. The plane and intermediate point elements cannot be used, and only the point of the links can be used.

Select the elements or links for the 3D alignment. Even if all elements and links are to be used, they must be selected.

Perform the <Evaluation><3D alignment> function and proceed as described for the 3D best fit.

8.7 Extras

Functions:

- Analysis
- Select group
- Parameters
- Import Catia list parameters
- Reset 2D
- Reset 2D sel.
- Set values (options: only selected, offset, tolerance, probe, column)
- Upwards
- Downwards

8.7.1 Analysis

With the <Analysis> function, you analyze the regular geometry elements from an existing data set.

Select the objects to be analyzed (curves, faces, surfaces) and perform the <Extras><Analysis> function.

A selection window appears for the regular geometry elements:

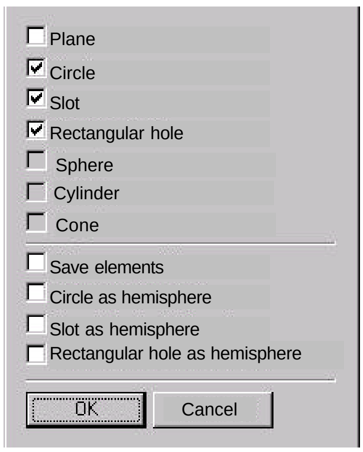


Figure 8-57, Selection window for regular geometry elements

Regular geometry

⇘ Select the elements which are included in the free form description and which you wish to filter out.

Please note the following:

1. The analysis of a rectangular hole can take some time, therefore you should analyze rectangular holes separately if possible.
2. If it is established during the analysis that an element is lying on an already existing or an analyzed plane, then this plane is entered as the working plane of the element.
3. The selected objects are analyzed one after another. Each object is checked, in the order shown in the illustration, to see whether it contains the corresponding element.
4. The number of selectable objects can affect the analysis as follows:

   Two curves: These are treated as one curve, enabling analysis of a slot, which consists of several curves. Please select only those curves which contain a circular segment of the slot.

   Four curves: These are also treated as one curve, enabling analysis of a rectangular hole which consists of several curves.

⇘ Switch the remaining options on or off:

Save elements: With this option, all regular geometry elements that have been found are immediately transferred into the regular geometry list. Otherwise, for each found element the parameters of the element are displayed, and you can modify these accordingly or completely reject the element.

Circle as hemisphere / Slot as hemisphere / Rectangular hole as hemisphere: Instead of the relevant element the hemisphere element, which contains the mid-point of the element, is set.

⇘ Click on <OK> to start the analysis.

The analysis is performed.

> ***NOTE***
>
> During the analysis, the offset attribute of an object is applied to the found element.

Depending on the <Save elements> option, you obtain either the corresponding dialogs for definition of elements or the following analysis results window:

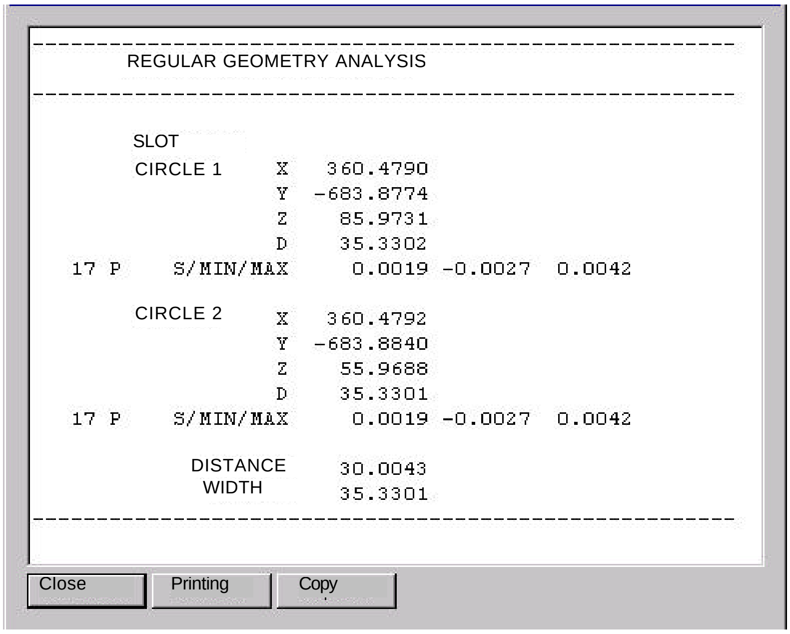


Figure 8-58, Regular geometry analysis

8.7.2 Select group

With this option, you can select several regular geometry elements which are then marked and selected.

8.7.3 *Parameter*

Change the parameters for the regular geometry list.

Display parameters

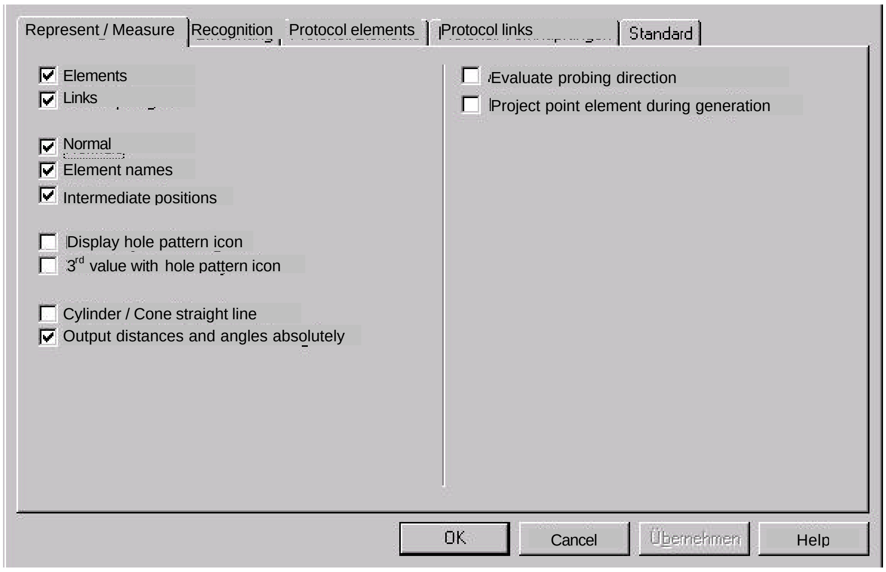


Figure 8-59, Regular geometry, display parameters

Elements	The elements are graphically displayed.
Links:	The links are graphically displayed.
Normal:	The normal of the elements or links is displayed. Keep this option switched on while you are generating the regular geometry list, in order to avoid subsequent measuring errors.
Element name:	The designation of the element or link is displayed.
Intermediate position:	The position of the intermediate position is displayed.
Display hole pattern icon:	The hole pattern icon is displayed for circles, slots and rectangular holes in the 2D view. The content of the graphic symbol is dependent on the settings in the parameters.

	The arrows give the direction of the deviation in the corresponding coordinate. Here the circle has a positive deviation of 0.0402 in the X-direction and a positive deviation of 0.0872 in the Y-direction.
3rd value for hole pattern icon	The value of the third (the missing) coordinate is also output.
Cylinder / cone straight line	The straight line from cylinder and cone for the link is shown.
Absolute distances and angles	With this option, the spatial distance is output for distances, not the delta value. For angles, only the value is output, not the preceding sign.
Evaluate probing direction	During manual probing, the probing direction is set identically to the normal direction.
Project point element during generation	If a new point element is generated during manual probing, then the point is first projected onto the existing data set, before being applied to the element. This means that the projected point is applied to the list of regular geometry elements.

Regular geometry

Recognition parameter

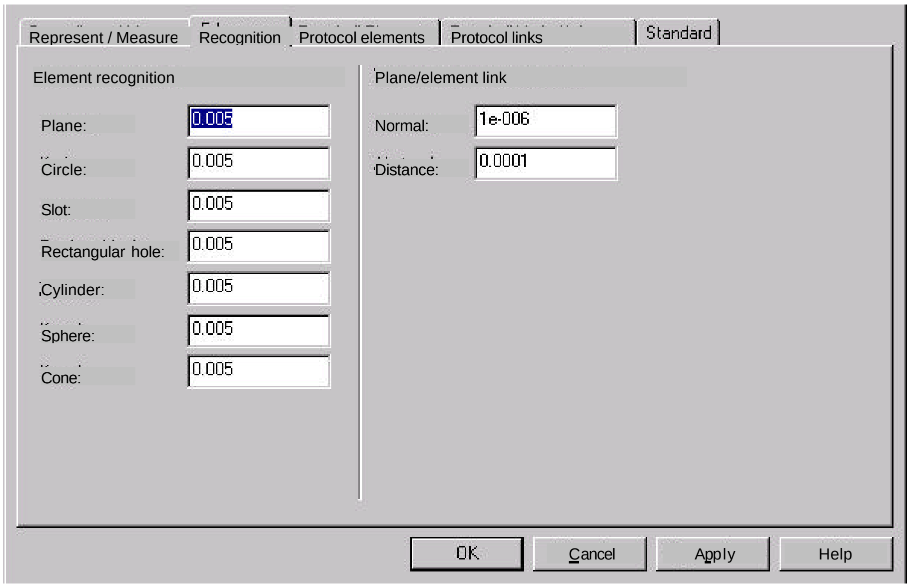


Figure 8-60, Regular geometry, recognition parameter

Element recognition:	This value specifies the standard deviation beneath which the relevant element is recognized. If a 2D element is analyzed, then the plane setting can also have an effect on the element recognition, as it will be checked first of all whether the points to be analyzed lie on a plane.
Plane/element link:	If a 2D element (e.g. a circle) is found, then it will be checked whether the element is already lying on an existing plane. The check is dependent on this parameter setting.
Normal:	If the direction of a normal deviates by more than this amount, no assignment of element to plane is performed.
Distance:	If the distance of the mid-point of an element from the plane in the orthogonal direction is greater than this value, no assignment occurs.

Protocol element and protocol links parameters

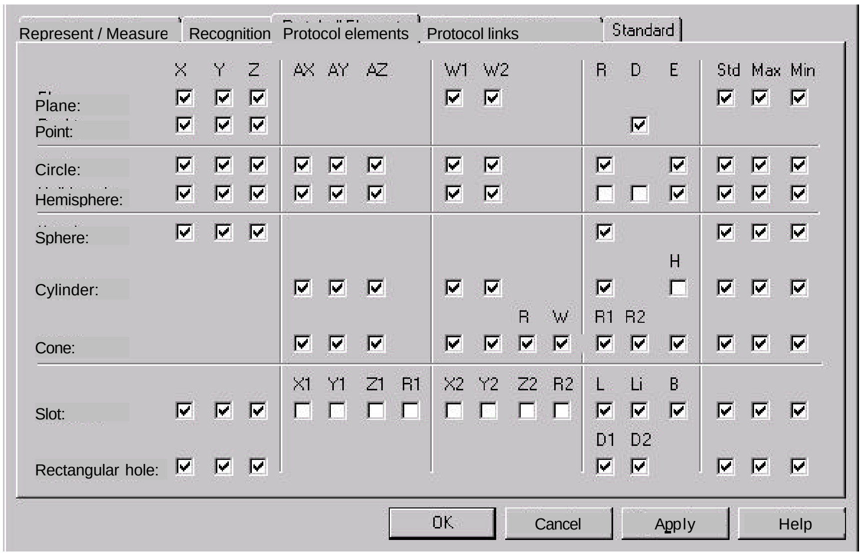


Figure 8-61, Protocol element parameters

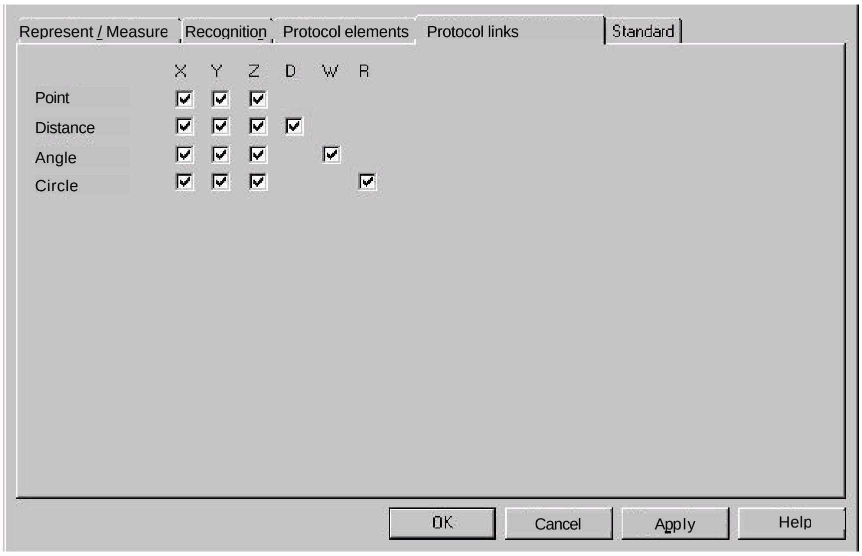


Figure 8-62, Protocol link parameters

Here you specify which information is to appear as standard in the measuring record. You can then overwrite this standard setting for each element or link. The setting of the already existing elements and links is not changed.

Regular geometry

Standard parameters

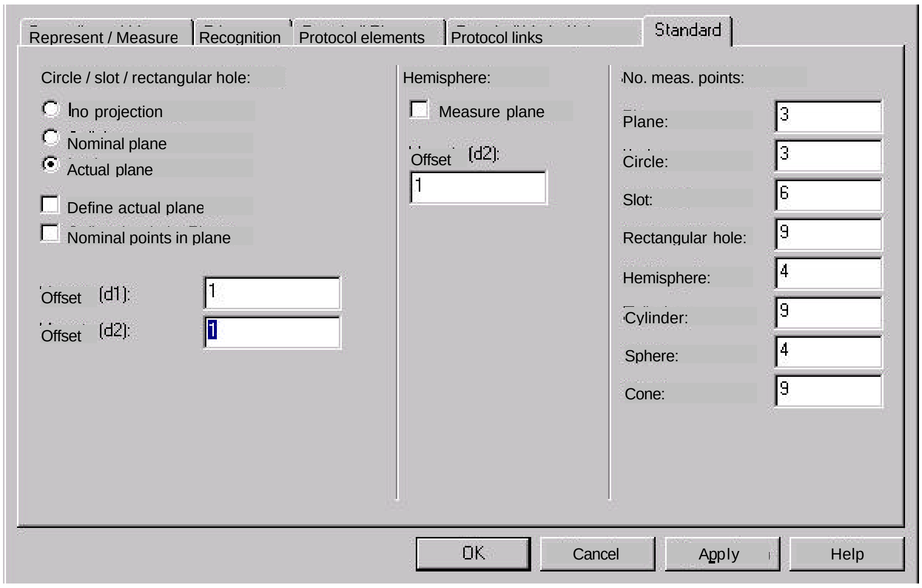


Figure 8-63, Regular geometry standard parameters

Here you enter the standard configuration of the parameters for the elements. The individual parameters are described for the elements.

NOTE

If you activate the <Actual plane> option, the <Define actual plane> option is not also automatically activated, which can lead to problems in determining the measuring results in the case of elements which do not have any link to a plane.

No. meas. points: Specify the number of measuring points which are to be used to measure the element.

8.7.4 Import Catia list parameters

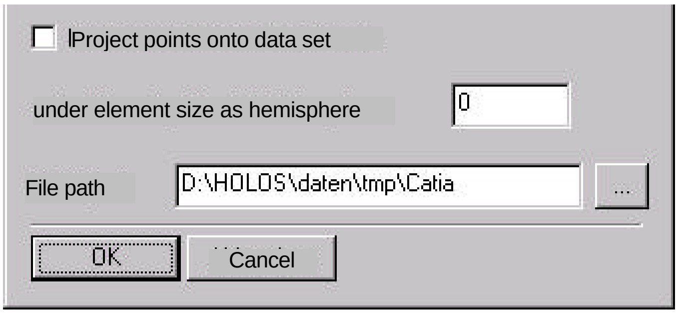


Figure 8-64

Points onto data set	With this option, individual points are projected from the list onto the data set. They are not evaluated, but applied to the list of regular geometry elements.
As hemisphere	You can enter a value in mm. If an element is smaller than this value, then a hemisphere is generated automatically.
File path	Specify the standard directory from which to read.

8.7.5 Reset 2D

In the two-dimensional view you can drag distance dimensions to another location with the mouse pointer. This option also undoes all such movements.

8.7.6 Reset 2D sel.

In the two-dimensional view you can drag distance dimensions to another location with the mouse pointer. All such movements are undone again with this option for those distances which are entered in the regular geometry list under links and selected.

8.7.7 Set values

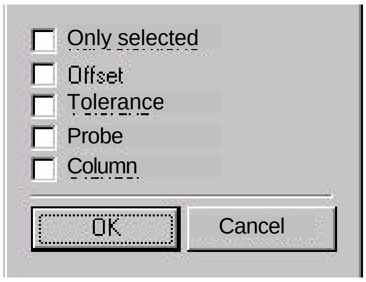


Figure 8-65

Only selected	The following options only apply for the selected elements.
Offset	The globally set offset is adopted.
Tolerances	The tolerance settings are set to the set global value. The modified tolerance settings are also overwritten.
Probe	The active probe is allocated to the elements to be measured.
Column	The active column is allocated to the elements to be measured.

8.7.8 Up

The selected element or link is moved upwards in the list by one position.

8.7.9 Down

The selected element or link is moved downwards in the list by one position.

9 Evaluation of measuring runs

This chapter describes the functions of the <Evaluation> menu. The functions are used to evaluate existing actual data at a later point in time.

The main function <Evaluation> is sub-divided into the following functions:

- Evaluate actual data
- Evaluate sections
- Evaluate regular geometries
- 3D best fit
- Measuring record
- Distance calculation

9.1 Evaluating actual data

9.1.1 Standard evaluations

The <Actual data> function has a branch leading to the <Evaluation> subfunction. With the <Evaluation> function, the deviations of the actual data on the nominal surfaces are calculated and subsequently displayed on the screen.

The type of representation is dependent upon the settings that you have made for the parameters in the <Graphics> parameter function.

📖 See Chap. 12.1

The measuring record will be created during the evaluation.

Operation

⇘ First of all, select the deviation representation via the <Graphics> parameter function.

Invoke the <Evaluation> - <Actual data> - <Evaluation> function.

A window opens for the selection of existing actual data (= measuring runs).

Select the actual data and click on <Open>.

The data will be evaluated and the result then displayed on the screen.

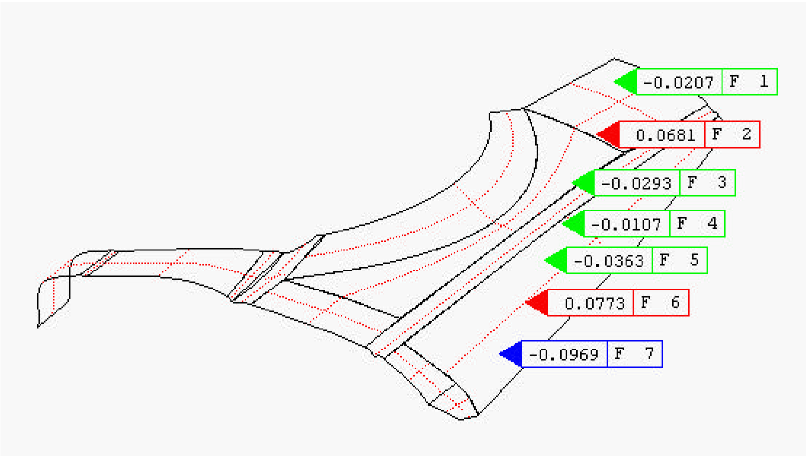


Fig. 9-1, Deviation representation with symbols

The content (number of points, deviation, nominal point, actual point, deviation vector and point type) and the typeface for the graphic symbols are defined via the evaluation parameters (NT) or the parameters for graphic symbols (UX).

Graphic symbols can also be moved at random or automatically arranged in the margin of the graphic window.

More detailed information can be found in the Extras chapter.

The following deviation representations are also possible:

- numerical
- vectorial
- with colored symbol
- with colored markings
- with colored marker (NT)
- tripod (NT)

9.1.2 Actual data as chromatic coordinates

For representation in chromatic coordinates, areas with the same tolerance zone position will be shown in the same color. In contrast to the deviation representation, here the entire surface is colored with a colored "marking“.

This type of evaluation can be utilized

- in the raster and grid measuring runs (here the measured patches is completely colored)
- for several manual probing points (here a convex envelope is formed around the points and the enclosed surface then colored).

Background information

In order to color the entire surface, the program needs, in addition to the information about the probing points, other information about the interspaces. Of the calculations that are required for this matter, mention will only be made at this point of those aspects that are of interest to you, the user.

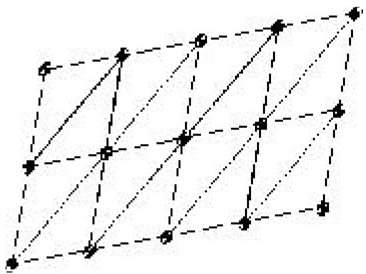


Fig. 9-2

First of all the points are joined by a so-called triangular meshing. The closeness of the triangular meshing depends on the nesting depth, which you can define yourself.

Nesting depth 1: a triangle is subdivided into four further triangles.

Evaluation of measuring runs

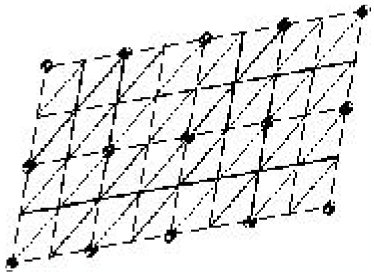


Fig. 9-3

For nesting depths of 2 and higher: the subdivided triangles will be further subdivided in the same manner etc.

> ***NOTE***
>
> The greater the nesting depth, the finer the subdivision of the surface and resulting precision in the evaluation. The nesting depth should however not exceed 5, since the computer speed might not then be adequate.

By means of various equalization calculations (Interpolations)the deviation of the probed points is now extended to the interspaces. Because of the flowing transitions, however, outliers on the surface cannot be recorded and a false impression can result. For this type of evaluation the probed points should therefore lie relatively close to one another.

NT version: The representation can be displayed illuminated for better three-dimensional representation, if this is set in the parameters.

Operation

Using the <Graphics > parameter function, first of all select whether the chromatic representation is to be filled or not.

Invoke the function <Evaluation> - <Actual data> - <chromatic coordinates>.

A selection window is opened with the existing actual data (= measuring runs).

Select the measuring runs to be evaluated and click on <Open>.

The dialog window for entering the nesting depth opens.

Enter the nesting depth (max. value of 5) and click on <OK>.

The data are evaluated and displayed on the screen in color.

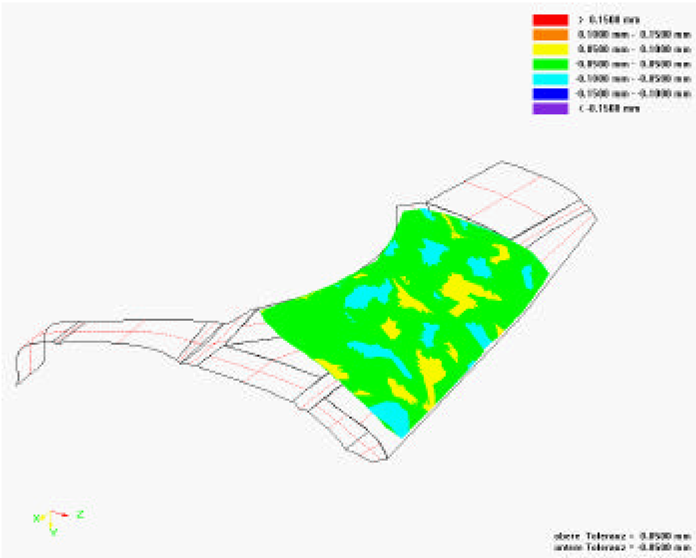


Fig. 9-4

9.1.3 Save actual data as measurement (NT)

Enter a "measurement name", under which a new directory is set up. All measured data of the currently loaded model will automatically be copied into this directory. In this way, you can save several measuring procedures in different directories for better clarity.

9.2 Evaluation of sections

HOLOS supports the Zeiss curve measuring program KUM, by making available possibilities for the evaluation of sections.

In principle, sections are represented as the superimposition of a polyline of nominal values and the polyline of the respective actual values that belong to it.

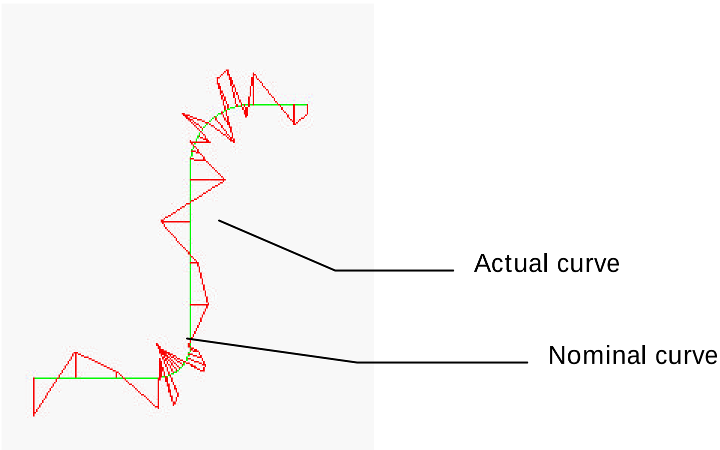


Fig. 9-5

Since the generation of nominal values in HOLOS is, in terms of design, not generally aimed at sections or curves, you must for the moment define sections via the measured actual values.

9.2.1 Definition of sections

Sections can be defined in two ways.

The simplest and fastest method for defining sections is in the evaluation parameters. .

Operation

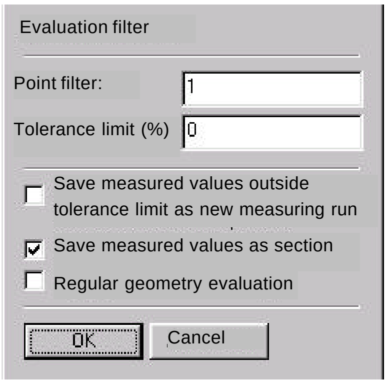


Fig 9-6

Activate <Save measured values as a section>.

In the next evaluation that you perform, the measured values will automatically be stored as a section and can be evaluated as a section.

If this type of section definition is not possible, because, for example, you cannot use all evaluated points in the section definition, you must generate the section on the screen.

A precondition is a previous evaluation of the actual values, i.e. evaluated actual values must be displayed on the screen. You define the sections by selecting the actual values.

Operation

Select the <Evaluation>-<Section> function.

A selection window opens.

Select <Define>.

A dialog window opens for organizing the sections.

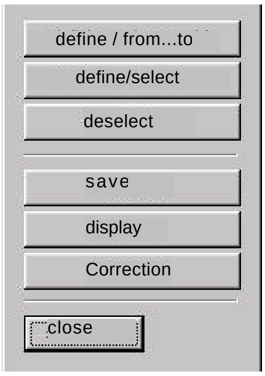


Fig. 9-7

define / from... to...

Using this function, you define sections from actual values that are positioned one behind the other. In the evaluation of measured values these are filed away in an internal structure for the graphic representation. Consequently measured points that lie one behind the other on a line or a raster/grid, also lie one after another in this structure.

The selection of the first value defines the start point of a section, the next value defines the end point. All the values that lie in between are applied for the section. Values that have been applied are designated by a circle.

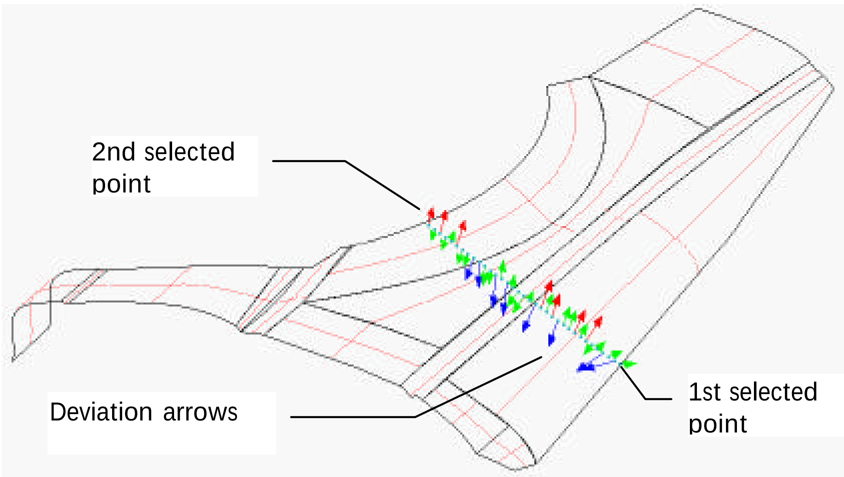


Fig. 9-8

define / select

With this function you select individual points for the definition of a section. At the same time a counter is initialized, i.e. after that you can begin with the definition of a new section at any given time.

deselect (NT)

The <deselect> function deselects the actual values selected up to this point.

save

The <Save> function takes care of the storage of sectionsin a file. The sections are filed with consecutive numbering (section0, section1, ...) in the respective workpiece directory. You can determine the name of a section yourself if the parameter for saving files is defined accordingly in the system parameters.

display

This function displays the last defined section together with the defined parameters for representation of the section on the screen. The representation of the section will be superimposed on the current graphical representation.

correction (NT)

The last selected actual value is deselected using the correction key.

delete (UX)

A dialog window opens, in which the sections present in the current workpiece directory can be deleted.

9.2.2 Evaluation and graphic representation of sections

With this function you select existing sections and display them on the screen. The representation of the sectionstakes place with the parameters that were set up for representation of the section.

For setting the parameters, see Chap. 12.17

Operation

Select the <Evaluation>-<Sections><Evaluate> function.

A window opens for the selection of existing sections.

Select the desired sections and confirm with <Open>(NT) or <OK> (UX).

The sections are displayed on the screen.

9.3 Evaluation of regular geometries

The functions for evaluating regular geometries allow you to perform regular geometry evaluations with any actual values.

You need these functions if you have no surface elements that are expressly recognized as regular geometry elements in HOLOS.

Probe points on the workpiece surface manually, or start an evaluation.

The measured values are displayed on the screen.

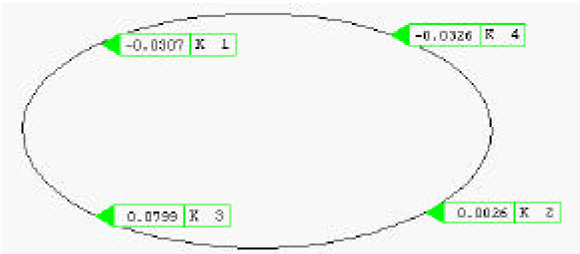


Fig. 9-9

As direct evaluation is not possible in manual mode, you must select the displayed measured values for the evaluation first of all:

In the <evaluation> menu select the function <regular geometry>(NT) or <regular geometries> - <Evaluation with selected actual values>.

This will take you into the mode for selecting actual values: “Select: POINT“ is displayed in the status bar.

Select the actual values by clicking on the arrow point with the left mouse button or use the box to select all actual values within an open window.

After selecting the first actual value, the dialog window opens for managing the selected points:

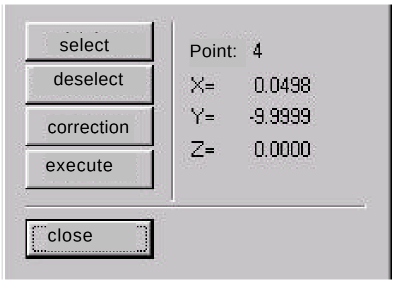


Fig. 9-10

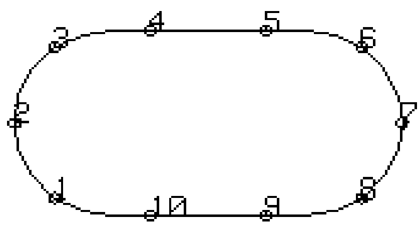
The coordinates of the selected points are displayed in the window.

Select — Mode for selecting actual values.

The following guidelines apply for the evaluation of regular geometry elements:

Slot

- 10 actual values must be selected for evaluation.
- The first 3 points must lie on the first semi-circle.
- The next 2 values must lie on the first straight section.
- The next 3 values must lie on the second semi-circle.
- The last two values must lie on the second straight section.
- A logical sequence must be observed for the point distribution.



Rectangular hole

8 values must be selected for evaluation, with 2 values lying on each of the four straight sections. A logical sequence must be observed for the point distribution.

Circle

At least 3 points must be selected for evaluation.

Plane

At least 3 points must be selected for evaluation.

Sphere

At least 4 points must be selected for evaluation.

Cylinder

At least 9 points must be selected for evaluation.

Cone

At least 9 points must be selected for evaluation.

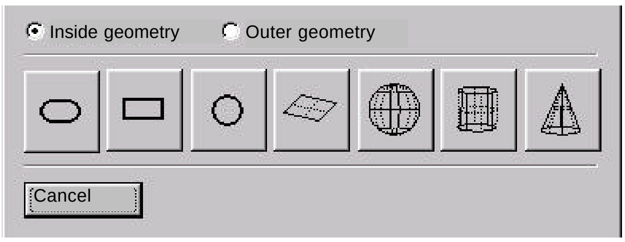
Deselect — Cancels the selection of all currently selected actual values. **Note that all actual values must be “deselected” before a new element is evaluated.** The selection of the previously selected values is not cancelled automatically.

Correction — Cancels the selection of the last selected actual value.

Execute

Start the evaluation when all required points have been selected.

The regular geometry selection panel is displayed:



Click on the icon of the element that you wish to evaluate.

The element is evaluated. The result is displayed on the screen and output in the measuring record:

Evaluation of measuring runs

Display of results for a circle

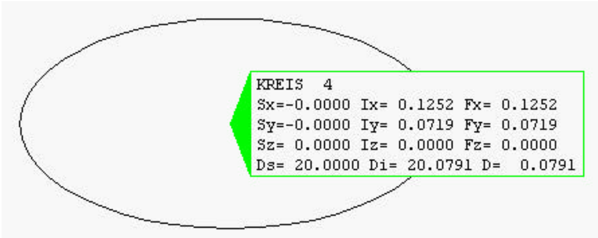


Figure 9-11

Measuring record for a circle in space

ADR		SY	ACT.DIM.	NOM.DIM.	U.TOL.	L.TOL.	DEV.	UEB
6	CIRCLE I	X	0.0476	0.0000	0.1000	-0.1000	0.0476	++
		Y	0.1399	0.0000	0.1000	-0.1000	0.1399	0.0399
		Z	0.0000	0.0000	0.1000	-0.1000	0.0000	+-
		D	19.9456	20.0000	0.1000	-0.1000	-0.0544	---
		E	0.1477					
	AXIS	X	0.0476	0.0000	0.1000	-0.1000	0.0476	++
		Y	0.1399	0.0000	0.1000	-0.1000	0.1399	0.0399
	X/Z	W1	-0.0000	-0.0000	0.1000	-0.1000	0.0000	+
	Y/Z	W2	-0.0000	-0.0000	0.1000	-0.1000	-0.0000	-
4 P		S/MIN/MAX	0.0201	-0.0201	0.0203			

Figure 9-12

You can specify which information is to be displayed on the screen for the graphic representations of the regular geometry evaluations.

Select the <Evaluation> - <Regular geometry graphic icons> functions in the <Parameters> menu.

The selection panel is displayed:

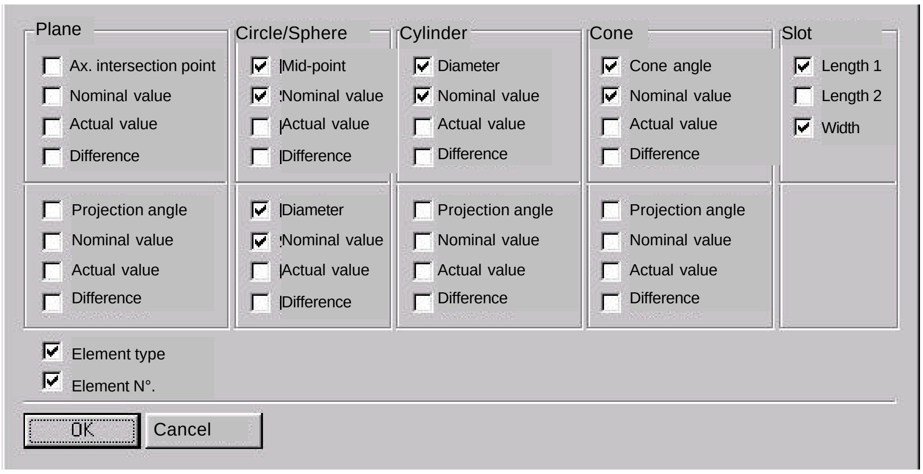


Fig. 9-13

Select the options for the desired result displays.

9.4 Execute 3D best fit

The 3D best fit denotes a best fit of the workpiece on the basis of the actual values. By means of a computational rotation and translation of the workpiece the actual and nominal values will be made to coincide to such an extent that the sum of the error squares is a minimum. (Gauss criterion). From this best fit HOLOS generates a transformation rule, that improves on the original alignment (= workpiece position, workpiece correction system).

The result of the best fit is displayed in a window. Rotation angle, translation vector and standard deviation are output. By means of the standard deviation you can decide whether you want to accept the best fit or execute another one using further actual values.

NOTE

If the existing workpiece correction system is to be corrected on the basis of the transformation rule, then you must send it back to the measuring software without fail. HOLOS itself does not notice the transformation rule.

In the measuring software itself, the modified workpiece correction system must again be stored, since otherwise the information will be lost on exiting the measuring software.

Precondition for the 3D best fit: the workpiece must be coarsely aligned. It is not possible to best fit a workpiece, that is not aligned beforehand, since it is not possible to carry out probings in this case.

Operation

Example: the actual values already exist.

⇘ **UX:** Define first of all with the parameter function

🕮 See Chap. 12.9

⇘ Invoke the <3D best fit> function in the <Evaluation> menu.

A selection window opens with the existing actual data.

Select the actual values and click on <Open>(NT) or <OK>(UX).

NT:The dialog window for defining the degrees of freedom for the 3D best fit opens.

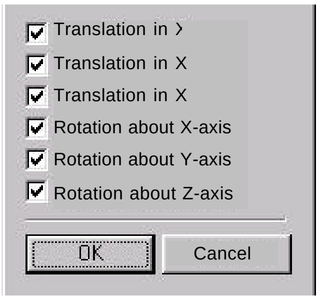


Figure 9-14

NT Define the degrees of freedom and click on <OK>.

The best fit is carried out and the result displayed in a window.

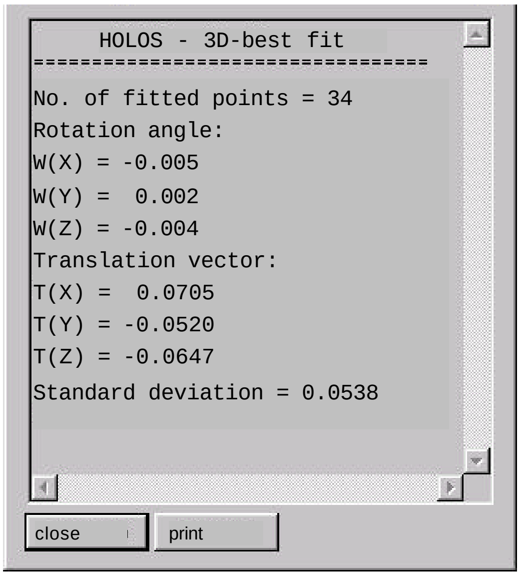


Fig. 9-15

For checking purposes you can now probe points or decide on the basis of the standard deviations, whether the best fit should be improved upon again.

The program queries whether or not to "transfer transformation matrix for W-position correction?".

⇘ Only answer the question with <Yes>, if you want a correction to the stored workpiece positioning system.

If you reply with <Yes> a further question will follow.

⇘ Only answer the question with <Yes>, if you want a correction to the actual data.

The data are transformed.

9.4.1 3D best fit with selected values

For a 3D best fit with weighting of individual values, actual values that can be seen on the screen are selected.
Values which should be weighted more heavily during the 3D best fit can be selected several times and as a result have greater influence on the best fit result.

> ***Prerequisite***
>
> Before defining the points of a weighted 3D best fit, the actual values must be evaluated.

Operation

⇘ Evaluate the actual values.

The deviations are displayed on the screen.

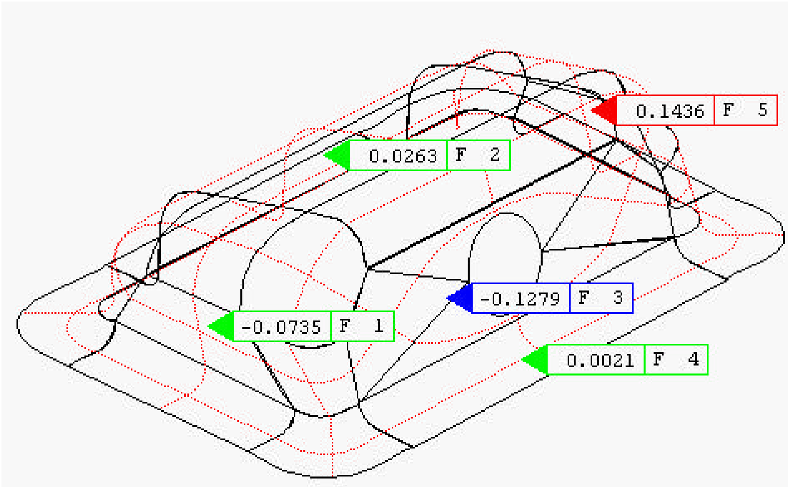


Fig. 9-16

Select the function <3D best fit>-<3D best fit with selected actual values> in the <Evaluation> menu.

The request “Select actual values” appears in the status line at the bottom of the screen.
You are now in the mode for selecting the actual values which are to be used for the 3D best fit.

Select the required actual values by clicking on them with the left mouse button. Actual values which are to be taken more into consideration during the 3D best fit should be clicked on several times.

After the first value has been selected, a dialog window appears, in which you can obtain information about the selected points:

Evaluation of measuring runs

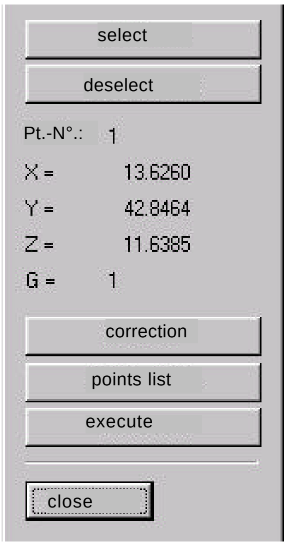


Fig. 9-17

Function	Meaning
select	Calls up the mode for selecting the actual values, if you have exited it by performing another action.
deselect	Cancels the selection of all selected actual values.
pno.	Number of the last selected point.
X, Y, Z	Coordinates of the last selected point.
W	Weighting of the last selected point.
correction	Cancels the selection of the last selected point.
points list	Displays list of all selected points:
Execute	Execute the 3D best fit with the selected values. To do this, **at least four different points** must be selected.

The result of the best fit is displayed on the screen:

Confirm the result with “Ok”.

A query appears, asking whether the result of the 3D best fit is to be transferred to the UMESS measuring software:

⇘ To do this, click on <YES>, in order to start the transfer.
If you do not wish to transfer the transformation matrix, click on <NO>.

⇛ After transfer of the transformation matrix to the UMESS measuring software, the current workpiece position system is corrected accordingly.

> ***NOTE***
>
> It should be noted that this correction is only temporary.
> You can use the functions for saving the W-position in the measuring software to permanently store the new workpiece position system.

9.4.2 W-position (Calypso)

The program functions only apply for the Calypso measuring software.

Reset

Resets the machine coordinate system as the workpiece system.

Set

A workpiece system saved for the workpiece is reset for Calypso.

A workpiece system is always automatically saved for the workpiece when an alignment has been completed.

Adopt

The current workpiece system from Calypso is adopted and saved.

9.5 Display measuring record

During evaluation of actual data a measuring record is created. Depending on the setting in the <Measuring Record - Output> parameter function, this will be output on the screen, the printer or to a file. If you have set output to the screen, the record will be output to the screen via the <Measuring record> function described here.

> **NOTE**
>
> It is always the record of the actual data which you last evaluated that is displayed. Selection is not possible.

Record layout

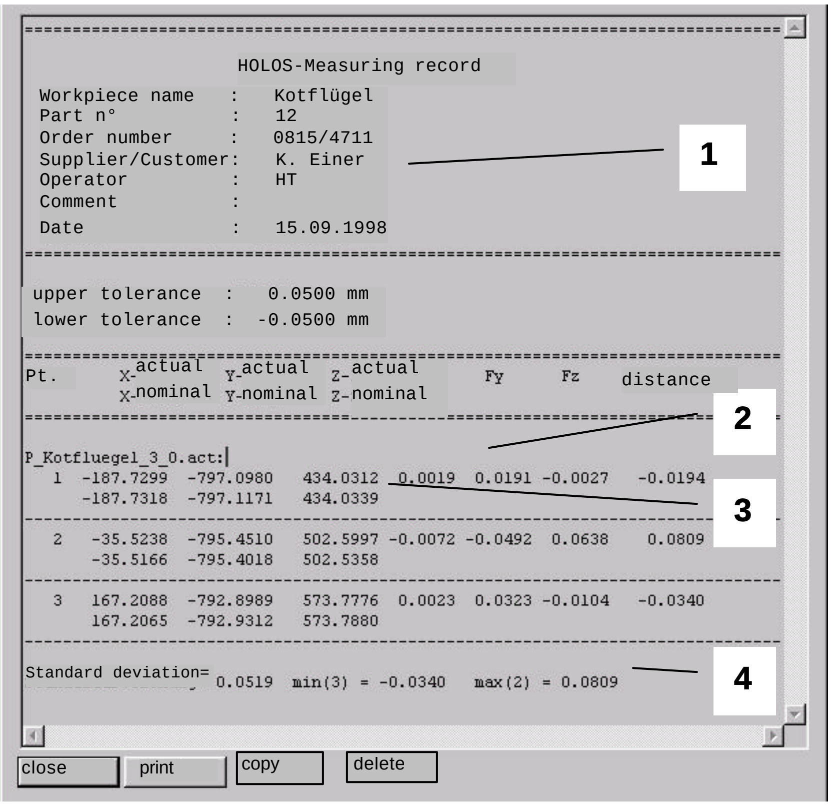


Fig. 9-18

1. Create the record header using the <Record header> parameter function.
2. The first line contains point number, coordinates of the actual point, deviation vector in Cartesian coordinates and distance of the actual point to the nominal point. The distance sign relates to the normal direction of the surface: positive sign = too much material,
   negative sign = too little material.
3. The second line contains the associated surface and the nominal point on the surface. The nominal point derives from the perpendicular projection of the actual point onto the surface.
4. At the end of the measuring record is the standard deviation together with the largest and smallest deviations.

Operation

First of all, with the <Measuring Record - Output> parameter function, select "Output to terminal".

Then evaluate the desired actual data using one of the evaluation functions.

Invoke the <Measuring record > function.

The record of the last evaluation is displayed. From this point on, you can also output it to the printer.

NT

The "Copy" key enables the record to be copied into the Windows clipboard and then to be inserted into other Windows applications. If you only wish to copy part of the record, mark it with the cursor before clicking on the key.

The "Delete" key deletes the entire record.

9.6 Distance calculation

Select the <Distance> function in the <Evaluation> menu.

The selection window for the various distance calculations is displayed:

Point → point

Distances between points are calculated. All points that can be displayed in HOLOS or points on the workpiece surface can be selected for this purpose.

Click on the <Point -> point> function.

Click on the first point and then on the second.

The distance between the two points is calculated and then displayed on the screen.

Model → selected point

The distance between a selected point and the workpiece surface is calculated.

Click on the <Model -> selected point> function.

Click on a point.

The distance between the point and the workpiece surface is calculated and displayed on the screen.

The results of the distance calculation are output in the measuring record.

Model → CAD point

The distance between the imported CAD points and the workpiece surface is calculated.
If you do not wish to use all points for the calculation, enter a value in the point filter input field.

Click on the <Model -> CAD point> function.

The distance between the CAD points and the workpiece surface is calculated and displayed on the screen.

The results of the distance calculation is output in the measuring record.

Model → digitization point

The distance between the digitizing point and the workpiece surface is calculated.
If you do not wish to use all points for the calculation, enter a value in the point filter input field.

Click on the <Model -> digitization point> function.

The distance between the digitization points and the workpiece surface is calculated and displayed on the screen.

The results of the distance calculation are output in the measuring record.

Model → scan point

The distance between points of scanning lines and the workpiece surface is calculated.
If you do not wish to use all points for the calculation, enter a value in the point filter input field.

Click on the <Model -> scan point> function.

The distance between the points of the scanning lines and the workpiece surface is calculated and displayed on the screen.

The results of the distance calculation are output in the measuring record.

Point filter

If not all points are to be taken into consideration for distance calculation with CAD, digitization or scan points, define a filter value in this field.

A value n means that only each n-th point is evaluated. In the case of scanning points, the filter relates to the points of a scanning line.

10 Working with objects

This chapter describes the functions in the <Objects> menu. The functions are used for the processing of objects, e.g. for selection, masking or changing the orientation.

Objects include curves, surfaces, patches and faces.

The main <Objects> function branches off to the following functions:

- Delete selected objects
- Mask selected objects
- Demask selected objects
- Delete masked objects
- Show masked objects
- Catalog
- Analysis
- Rotate orientation
- Show orientation (NT)
- Search for object
- Remove CAD points (NT)
- Select point (NT)

10.1 Select objects

In numerous functions you need to select an object before it is processed. There are two ways in which you can select objects:

- by clicking on them in the graphics window using the mouse
- by selecting the name of the object from a catalog (<Catalog> function).

To select objects in the graphics window you need to click on the objects in certain places:

Object	**Click**
Surfaces and patches	on the isoparametric lines
Curves	on the curve
Faces	within the face
Points	on the point
Lines	on the line

A selected object is deselected by clicking on it again.

NT

The Windows functions "cut", "copy" and "insert" are also available. These allow you to insert objects e.g. into another model or into "Dimension" via the clipboard.

10.2 Delete selected objects

This function deletes all selected objects from the hard disk. Before the delete action a list of all currently selected objects appears. In addition, you need to confirm a safety query.

Objects cannot be deleted if they have other objects that depend on them.

Example: a curve that is referenced by a FACE cannot be deleted before the FACE is.

10.3 Mask selected objects

This function masks selected objects. Masked objects will no longer be taken into account in any calculations or graphic representations. They are nevertheless still physically present and can be demasked again at any time.

Masked objects are either blank or shown colored brown, depending on the "masked objects" parameter in the <Representation> function on the graphics menu bar.

> ***NOTE***
>
> In certain functions, surfaces are also masked by the program itself. This is the case if a new surface is calculated through "Re-parametrization".

In the present program version, during conversion to a VDA file the masked objects will also be converted. Therefore, they must be deleted beforehand if they are not to appear in the VDA file.

Operation

- First select the objects using the <Define group> function.
- Then select the <Mask selected objects> function.
- The selected objects are masked.

10.4 Demask masked objects

With this function you can demask masked objects. They will then be available again for all functions. You can only ever demask all objects of a type, not a selected individual object.
The types of the objects are selected via subfunctions:

- all objects
- all surfaces
- all patches
- all faces
- all curves

10.5 Delete masked objects

Using this function you can delete masked objects without re-selecting them.

You can choose whether you wish to delete all masked objects, only masked surfaces, masked faces or masked curves.

Call up one of the functions for deleting masked objects.

The masked objects are displayed in the graphic display.

In the window which appears you must confirm the function for deleting masked objects.

Click <YES> to delete the masked objects or <NO> to cancel the function.

10.6 Show masked objects

You can use the <Representation> function on the graphics menu bar to set beforehand whether masked objects will on principle be shown or not.

If on principle they will not be shown, you can show them on the screen colored brown by using the <Show masked objects> function.

10.7 Catalog of objects

The <Catalog> function lists all existing objects of a particular type by their name and then permits the selection of a particular object. The object types are selected via the subfunctions:

- Surfaces
- Faces
- Curves

Purpose of the function

The function serves to select objects when clicking on them with the mouse is difficult. This is the case, for example, when several objects are lying one on top of the other. Selected objects will be immediately marked in color.

In the NT version, masked objects can also be displayed in the catalog, and individual objects can be demasked.

Operation

Click on the <Catalog> function and then the desired object type.

A window with all objects of this type will appear.

Select the object name.

The object will be marked in color in the graphics window.

By renewed selection, an already selected object is deselected again.

10.8 Analysis of objects

The analysis of an object provides you with a whole series of information that is listed in the following table.

Object	Information
Surface (SURF)	Name of surface Number of elements in S and T Direction of the surface normal
Patch	Name of surface Position of the patch in S and T of the surface Polynomial degree in U and V (iordu and iordv) Direction of the surface normal
Curve (CURVE)	Name of curve Number of curve segments Orientation of the curve
FACE	Name of FACE and SURF element Number of surface curve lines Orientation of the surface

The surface normal and the direction of the curve or orientation of the surface will be shown on the object itself in the graphics window. You will find all other information in the status line.

NOTE

To obtain information about a segmented surface, you must click on the isoparametric lines of this surface. These however only become visible when you have turned off the patch representation (<Representation> function in the graphics menu bar).

Purpose of the function

- If you are generating surfaces interactively on the screen (functions in the <Digitize> menu) you need to know the orientation of the curves for a correct definition.
- If you need to "re-parameterize" one or more surfaces ", then to execute the function correctly you need to know the U and V directions, i.e. the orientation of the surface.

Operation

First click on the <Analysis> function.

The program switches to analysis mode and this is shown in the status line with "Select: Analysis".

Now select the object to be analyzed by clicking on it or via the catalog.

The results of the analysis will appear.

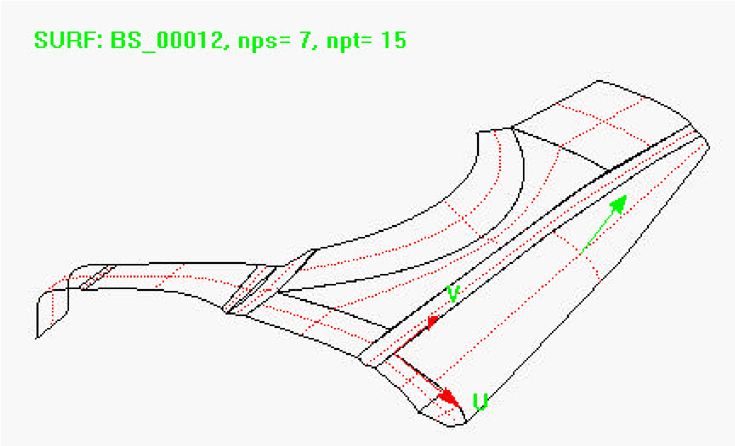


Figure 10-1

Whilst the program is in analysis mode you can have as many objects analyzed as you like.

To exit the analysis mode, press the middle mouse button whilst the mouse cursor is positioned within the graphics window.

The program switches back to Single-Select mode.

10.9 Attributes for surface elements

You can assign the following attributes to surface elements (surfaces, faces):

Color
When rendering is switched on, the surface element appears in the defined color in the graphic representation.

Tolerances:
The assigned tolerances are taken into account when evaluating measuring runs. Each element is evaluated with its defined tolerance.
In this case, no evaluation is possible with defined tolerance classes, just as during evaluation with chromatic coordinates no different tolerance considerations can be used.

Offset:
The offset serves for correction of sheet thicknesses. During evaluation of measuring runs, the entered offset is taken into account for each element.

If no attributes are entered, then representation and evaluation are in accordance with the standard settings.

Click on the <Attributes> function in the <Objects> menu.

At the bottom left the message SELECT: ATTRIBUTE appears.

Select a surface element.

The selected element is marked in color, and a dialog window appears for entering the attributes:

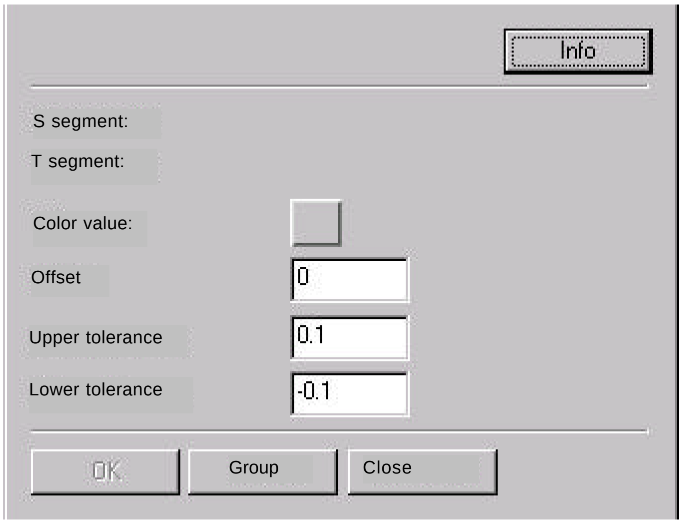


Figure 10-2

At the top of the window, various information is displayed about the selected element, e.g. element name, number of segments.

Color value A selection list with the available colors is displayed.

Offset Enter the offset value in the text field.

Tolerances Enter the values for upper and lower tolerance in the text fields.

Group If a group of elements is selected, use this function to define attributes for the entire group.

Info A list containing the elements for which attributes have already been defined is displayed:

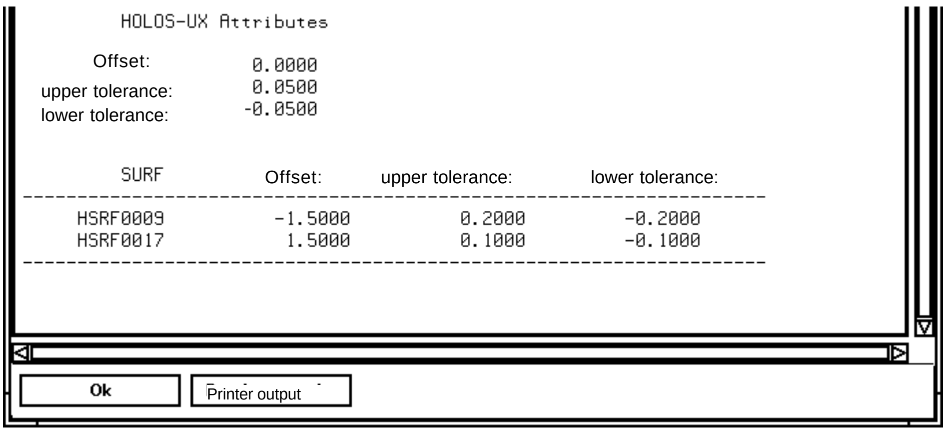


Figure 10 -3

The set attributes are taken into account in the next evaluations for actual data and sections:

Example:

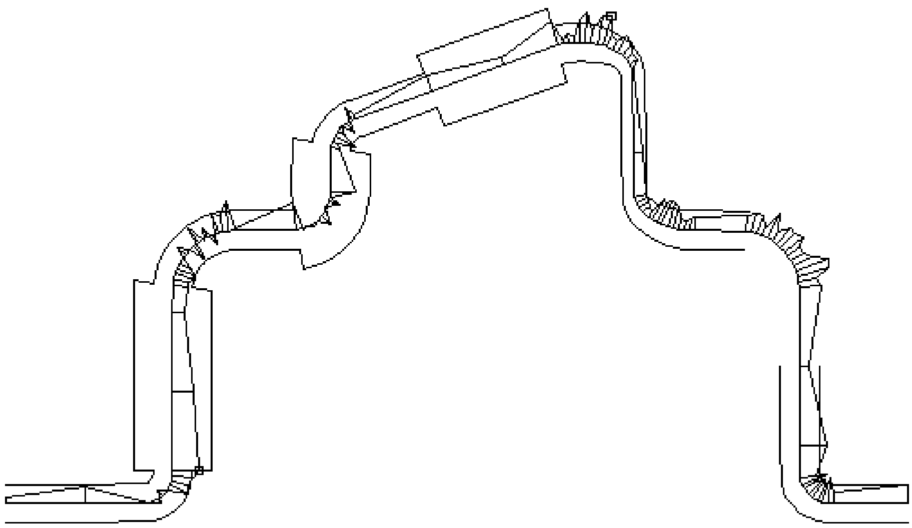


Figure 10-4, Section evaluation with different tolerances

10.10 Rotate orientation

With this function you rotate the orientation of the selected object. A rotation of the orientation can be necessary if you generate surfaces from curves or if you "reparameterize" several surfaces (see <Analysis> function) or if the orientation in the VDA file is incorrectly defined, which would lead to collisions in the measuring run.

The function has the following effects:

- For surfaces, the parameters S and T are interchanged. The surface normal is rotated.
- For patches, the parameters u and v are interchanged. The normal of the entire surface is rotated.
- For faces, the orientation of the associated surface is rotated.
- For curves, the running direction is rotated.
- The orientation is displayed immediately.

The rotation of the orientation cause a change in the data structure. The data structure is stored in accordance with the system parameters in the <Parameters> menu.

NOTE

In manual probing ("Patch identification") the program recognizes whether the probing direction agrees with the surface normal of the identified patch. If this is the case, the orientation of the patch is automatically rotated.

Operation

Select the object that is to be modified.

Click on the <Rotate orientation> function.

The orientation is rotated and displayed immediately.

10.11 Show orientation (NT)

The normals of all surfaces are displayed. If more than just one element is displayed, then only the normals of the selected objects are displayed.

10.12 Search for object

With the <Search for object> function, you can look for objects using their names.

Operation

Click on the <Search for object> function.

A window opens for entering the object name.

Enter the text and accept the entry with <OK>.

The object is color marked and selected in the graphic window.

By clicking on <OK> again you deselect the object again.

10.13 Remove CAD points (NT)

Using the <Remove CAD points> function you can remove CAD points from the main memory of HOLOS. Following a safety query, you can also completely delete the stored information.

In order to re-display CAD points after removing them, they must first be re-imported.

10.14 Select point (NT)

The point selection mode is activated via the <Select point> function, in order to select points for the generation of scanning lines, for example (these functions are only available if the Digitization option has been installed on your system).

11 Working with groups of objects

This chapter describes the functions in the <Group> menu. They are used in the definition and management of groups.

If certain functions are to have an effect on a number of objects, combine these objects together in a group beforehand.

The main <Group> function is subdivided into the following functions:

- Define group
- Extend group
- Delete group
- Display group
- Store group (NT) / Save group (UX)
- Load group
- Select group
- Copy group
- Rename group
- Delete group
- Bézier polygon (UX)

11.1 Local groups

A local group is valid until the "Select mode“ (group, single, analysis) is changed, but has a maximum duration of one work session.

Groups are formed by selecting objects.
🕮 You will find further information about how to select objects in Chap. 10.1.

Purpose of the group

A group is a prerequisite for numerous interactive functions. A defined group is a requirement for the definition of certain measuring runs and in the interactive generation of surfaces from curves.

11.1.1 Define group

The <Define> function activates the "Group select mode”, which can be recognized by the "Select: GROUP”message in the status line. As long as this mode is active, you can select objects by clicking on them and then combine them into a group.

Operation

Click on the <Define> function.

Any existing group will be deleted.. You can now define a new group.

Click on the objects that you want to combine into a group.

The objects will be marked in color.

If you want to remove an object from the group, click on it once again.

If you want to add objects to an existing group, utilize the <Extend> function.

To return to the "Single select mode", press the middle mouse button whilst the mouse cursor is in the graphics window.

The status line then shows "Select: SINGLE“.

11.1.2 Extend group

With the <Extend> function you can add objects to an existing local group. This function is only necessary if the program is no longer in "Group select mode“.

If the program is still in the above mode, a group can be extended at any time by clicking on further objects.

Operation

The <Extend> function activates the "Groups select mode “ and is then operated like the <Define> function.

11.1.3 Delete group

With the <Delete> function you deselect all objects in the current group and as a result delete the group. The objects themselves are not deleted in the process.

11.1.4 Display group

The <Display> function lists all objects in the current group together with their names in a special window.

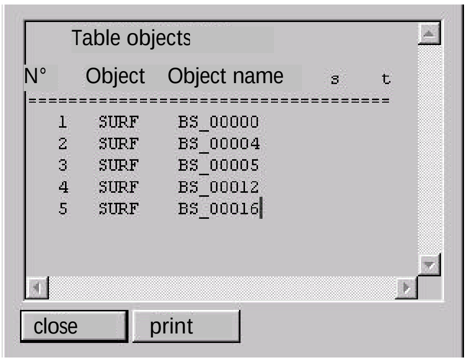


Fig. 11-1

11.2 Display Bézier polygon(UX)

The <Bézier polygon> function displays the Bézier polygon for the surfaces and curves of the current group. The Bézier polygon shows the support points of a surface.

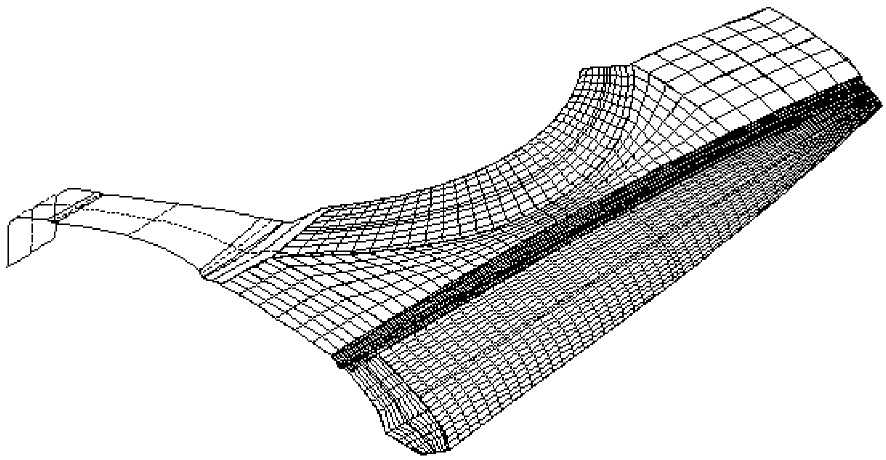


Figure 11-2

Further information on the Bézier polygon is contained in the keyword index in the Appendix.

> ***NOTE***
>
> If the representation is modified (rotate, cutout, refresh etc.) the polygon is deleted again.

11.3 Global groups

A global group is saved under a name and is therefore available again at any time.

> ***NOTE***
>
> If a VDA file contains the element GROUP, it will be applied as a global group

A global group can be selected and deleted as a whole and consequently provides faster access.

11.3.1 Store group(NT)/Save group (UX)

With the <Save> function you file away a local group as a global group. The group is saved in conjunction with the entry of a name.

Once you have entered the name in the input window and confirmed it, the remaining part of the model will be blanked out. All subsequent actions then apply only to the group.

You can always obtain access to the entire model by the management of global groups. If you want to work with the entire model once again, load it by means of the <Load group> function.

11.3.2 Load group

With the <Load> function you load a global group into the main memory. A window listing all the groups for a model is displayed for selection of the group.

The model itself is loaded by clicking on <Model>.

The operation of selection windows is described in Chap. 1.9.5.

11.3.3 Select group

With the <Select> function you select a group from the list of existing groups. In the process you automatically select all of the objects in this group. This has the advantage that you no longer have to click on the objects individually.

> ***NOTE***
>
> If you select the entire model as a group, **all** of the objects that the model contains will be selected.

11.3.4 Copy group

With the <Copy> function you copy the group temporarily available in the working memory, and file it under another name on the hard disk.

To create the copied group, you have to enter a group name.

11.3.5 Rename group

With the <Rename> function you allocate a new name to the currently existing group in the main memory.

Enter the new name and confirm with <OK>.

11.3.6 Delete group

With the <Delete> function you delete global groups from the hard disk. The objects in a group are not deleted in the process. To select the group, a window is displayed listing all the groups for a model.
You cannot delete the model itself by using this function. For this purpose utilize the <Delete model> function.

NOTE

You cannot delete an active group. If you want to delete the active group, you must first load a different group or the model.

12 Transformation of objects

This chapter describes the functions in the <Transformation> menu. The functions are used to modify the position of objects or to generate new objects in a modified position.

The main <Transformation> function is subdivided into the following functions:

- Mirroring
- Translation
- Rotation
- Scaling
- Offset surface
- Mirror nominals (UX)
- Invert actuals

Basic information on transformations

> ***NOTE***
>
> Take care when using these functions, since surface data will be modified in the process. Also bear in mind that subsequently defined nominal data will no longer apply in some cases!

Where objects are dependent on other objects, these are automatically transformed at the same time. Example: a FACE is to be transformed. As a result of this, the associated surfaces and curves will also be transformed.

12.1 Definition of type for transformation

During activation of a transformation function, a further window appears, with which you can determine the elements to be transformed:

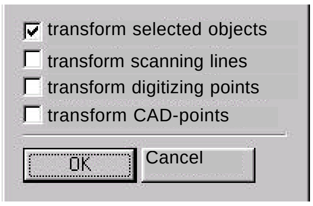


Fig. 12-1

transform selected objects

Activate this function in order to transform selected objects (surfaces, faces, curves).

transform scanning lines

Activate this function in order to transform scanning lines. If scanning lines are selected, only these scanning lines are transformed.

transform digitizing points

Activate this function in order to transform digitizing points.

transform CAD points

Activate this function in order to transform CAD points.

12.2 Mirroring objects

With the <Mirroring> function, all selected objects are mirrored. The "mirror axis" is one of the three basic planes. You select one of them via the function's input window.

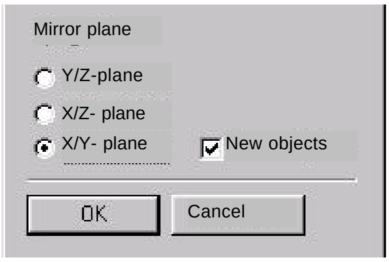


Fig. 12-2

If the “New Objects” option is activated, new objects will be set up in addition to the original ones. If the option is not selected, the original objects will be overwritten by the mirrored objects.
If you want to transform objects which are referenced from several other objects, it may be necessary to generate copies which will not be overwritten.

Example: A curve can be referenced from several FACE elements. However, if only one of these FACES is transformed, then the original curve is preserved.

Operation:

First select the object or the group of objects that are to be mirrored.

Click on the <Mirroring> function.

The dialog window for mirroring opens.

Select the plane in which mirroring is to take place.

Decide whether new objects are to be created, and if so, click on "New objects".

Accept the entered options with <OK>.

The action is carried out.

12.3 Translation of objects

With the <Translation> function, you translate all of the selected objects. You enter the direction and magnitude of the translation in the function's input window as the "Translation vector.

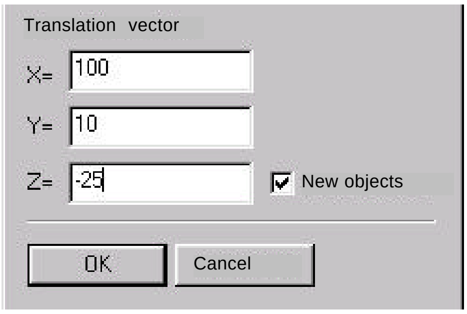


Fig. 12-3

If the “New objects” option is activated, new objects will be set up in addition to the original ones. If the option is not selected, the original objects will be overwritten by the translated objects.

Operation:

First select the object or the group of objects that are to be translated.

Click on the <Translation> function.

The dialog window for translation opens.

Enter the translation vectors by which the selected elements are to be translated.

Decide whether new objects are to be created, and if so, click on "New objects".

Accept the entered options with <OK>.

The action is carried out.

12.4 Rotation of objects

With the <Rotation > function, you rotate all selected objects. You enter the rotation point and rotation angle in the function's input window. The rotation angle always refers to an axis about which rotation then takes place.

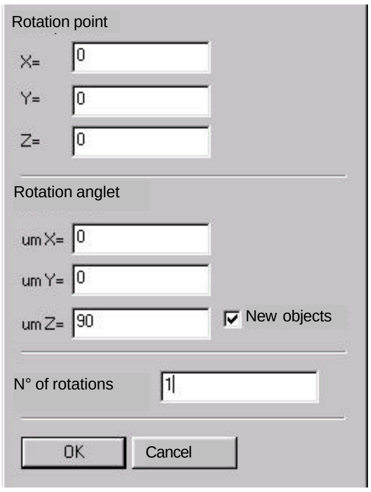


Fig. 12-4

If the “New objects” option is activated, new objects will be set up in addition to the original ones. If the option is not selected, the original objects will be overwritten by the translated objects.

Operation:

First select the object or the group of objects that are to be mirrored.

Click on the <Rotation> function.

The dialog window for rotation opens.

Enter the point of rotation, the rotation angle around the relevant axis and the number of rotations.
Decide whether new objects are to be created, and if so, click on "New objects".
Accept the options that were entered with <OK>.

The action is carried out.

12.5 Scaling

To scale selected objects, activate the <Scaling> function in the <Transformation> menu.

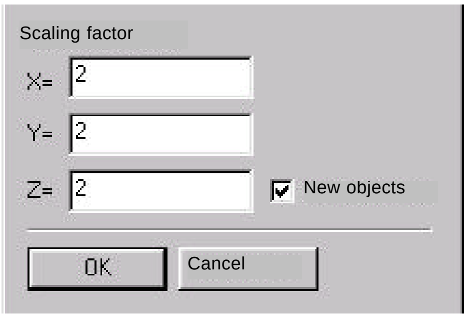


Fig. 12-5

If the “New objects” option is activated, new objects will be set up in addition to the original ones. If this option is not selected, the original objects will be overwritten by the translated objects.

Operation:

First select the object or the group of objects that are to be mirrored.

Click on the <Scaling> function.

The dialog window for scaling opens.

Enter the scaling factor for the respective axes around which the selected elements are to be scaled.

Decide whether new objects are to be created, and if so, click on "New objects".

Accept the entered options with <OK>.

The action is carried out.

12.6 Offset surface

An offset surfacedenotes a surface that is translated in the normal direction. For this type of translation the direction is fixed. You simply specify the value of the translation. The value can be positive or negative.

positive value: Offset surface will be shifted in the direction of the normal.

negative value: Offset surface will be shifted in the opposite direction to the normal.

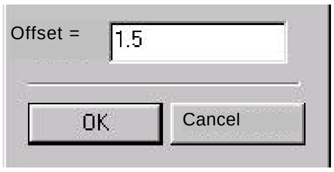


Fig. 12-6

Operation:

First select the surface.

Click on the <Offset surface> function.

A window is opened for entering the offset.

Enter the value and confirm with <OK>.

The offset surface is calculated and shown in addition to the old surface. The old surface is marked.

12.7 Mirror nominal values

In the case of axially symmetrical workpieces, as a rule only the data of one half are saved and supplied to systems for further processing. Therefore, not all data that are required for complete quality assurance are available in HOLOS.

Using the <Mirroring of measuring runs**>** function, you can mirror already defined nominal values on a mirror plane. If no surfaces exist for the mirrored zone, then the associated surfaces are also mirrored automatically.

Operation

Click on the <Mirror nominal values> function.

A window opens for selecting existing measuring runs.

Select the relevant measuring run and confirm with <OK>.

The nominal values are displayed on the screen and a window opens for defining the mirror plane (Y/Z or X/Z or X/Y):

Figure 12-7, Mirroring nominals

Change probe — With this option, the mirrored nominals are measured with another probe.The currently set probe is then assigned to them.

Use old name — With this option, you save the mirrored measuring run under the same file name.

Select the required mirror plane and option(s) and confirm with <OK>.

The displayed measuring points are mirrored.
An attempt is made to identify a surface in the target area.

If no surface is found, then the surface belonging to the nominal point is also mirrored.

12.8 Invert actuals

With this function, the direction of the deviations is inverted. The actuals are positioned in the normal direction of the surface onto the other side of the surface and saved in a new file.

13 Definition of parameters

This chapter describes the functions of the <Parameters> menu. The functions are used for setting up fundamental parameters for the many different areas.
The main <Parameters> function subdivides into the following functions:

NT	UX
• Graphics	• Graphics
• Clipping plane	• Rendering
• FACE isolines	Rendering Parameters
• Rendering	Rendering colors
• Markings	• Markings
• Measuring procedure	• Measuring procedure
• Evaluation	• Evaluation
Tolerances	• Graphic icons
Evaluation	• Macro run
Graphic icons	• Clearance planes
Chromatic representatioin	• Digitization
• Patch identification	• Patch-Ident
• Probe	• Probe
• Measuring record	• Tolerances
• Clearance planes	• Tolerance classes
• Digitizing	• 3D best fit
• Scale plot	• Measuring record output
• Printing	• record type
• System	• Record head
• Sections	Standard record head
• Output	User record head
• DMIS output	• Output
• Measuring software	• Sections
	• System
	• Printer setup
	• Serial interface (only with UMESS300 /UMESS 1000)

All parameter windows are operated by the entry of text or the selection of options.

📖 Read up on operation in Chap 1.9.4 and Chap. 1.9.3.

13.1 Graphics parameters

Graphics parameters are linked both to a model and to the subgroups of the model. When you load a model or a group, the associated graphics parameters will be selected as a result of this. If you have not yet defined any graphics parameters for a model or a group, the last parameters that were defined will still be active.

The graphics parameters apply to various elements of the graphic representation.

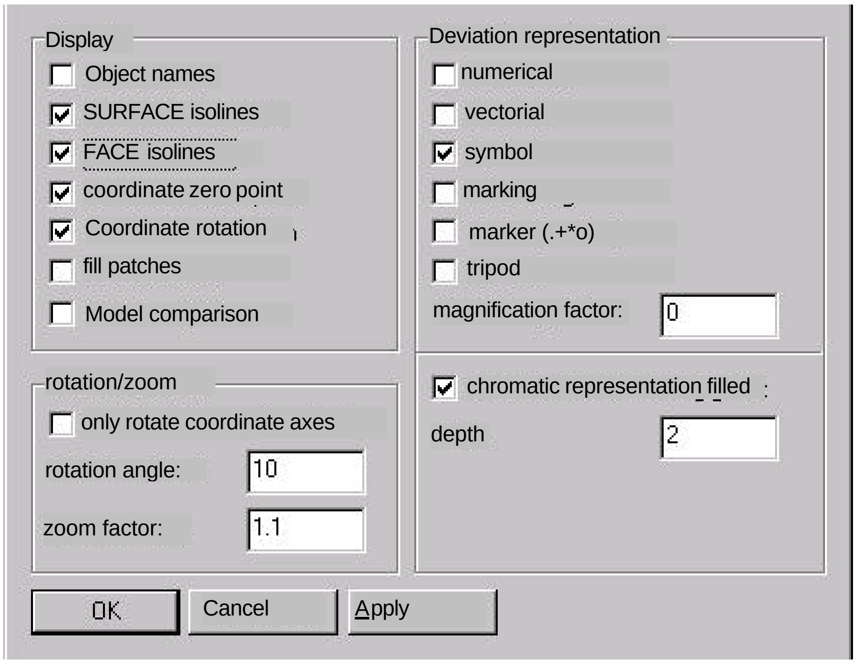


Figure 13-1

You can also open the window for the graphic representation parameters by clicking on a free surface on the graphics menu bar with the right mouse button.

Object names

The names of the graphic objects, curves, surfaces and faces are displayed. Precondition: their representation must be turned on (<Representation> function in the graphics menu bar).

SURFACE isolines

The SURFACE isolines of surfaces and patches are displayed. Significance: iso lines SURFACE are useful for clicking on surfaces and patches.

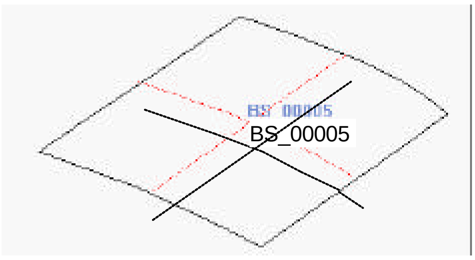


Figure 13-2

FACE isolines

Select whether the iso-lines are to be displayed. Preselect the relevant parameters under "FACE iso-lines".

Coordinate zero point

The origin of the workpiece coordinate system is displayed on the screen.

Raster dimension

In the view of the model in the three basic planes (<Sectional image> function) and in the 3D-view, a raster can be additionally displayed.

Select the basic planes that are to be rastered in a window, and enter the width of the raster.

Coordinate rotation(NT)

The point of rotation about which the graphic representation is rotated is displayed on the screen. The point of rotation is defined using the graphics menu bar.

🕮 See Chapter 3.6, Image rotation.

Fill patches

The patches are filled. The representation takes place in the order in which the patches were saved, i.e. patches lying at the back will not be represented if they are defined as "hidden surfaces".

Model comparison (NT)

The information resulting from a model comparison (different coloring of surfaces) is graphically represented on the screen.

13.1.1 Parameters for rotation/zoom

Rotate coordinate axes only

You rotate the entire graphic interactively on the screen by moving the mouse whilst keeping the right mouse button pressed down. As the mouse is moved, the graphic rotates at the same time. For large models however, the screen construction will be carried out more slowly. To save time, this parameter enables just the coordinate axes to be rotated at the same time. The model will only be built again when you release the right mouse button.

Rotation angle

The rotation angle preselects the step width for the rotation.

Zoom factor(NT)

The zoom factor preselects the magnification factor.

13.1.2 Deviation parameters

The deviation parameters come in useful for the evaluation of actual data (<Actual data> function in <Evaluation> menu).

You can also open the window for the graphic representation parameters by clicking on a free surface on the graphics menu bar with the right mouse button.

numerical

The deviation of a measured point is output as a numerical value.

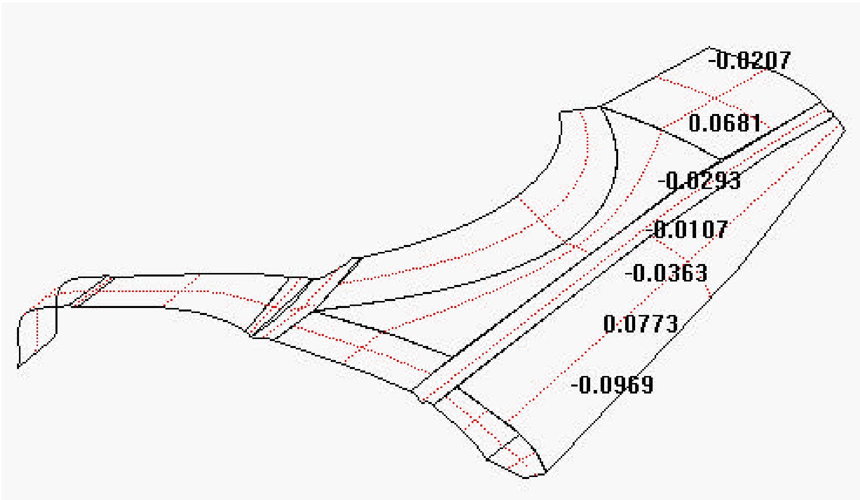


Figure 13-3

vectorial

The deviation of a measured point will be shown by a vector of a certain length. The size of the vector can be changed using the magnification factor.

Definition of parameters

symbol

The deviation of a measured point will be output as a numerical value in a colored symbol. The color corresponds to the tolerance zone position.

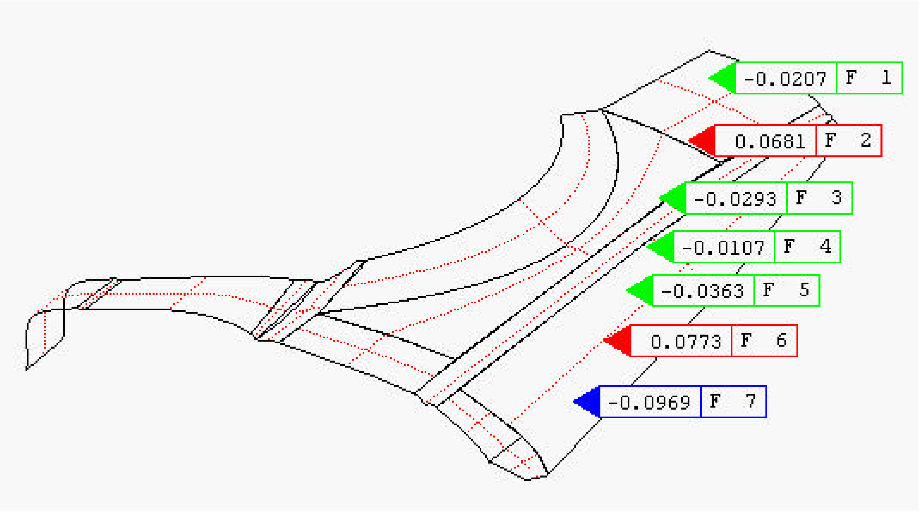


Figure 13-4

The content and position of the graphic icons can be changed in various ways:

The graphic icons allow you to display the nominal value, actual value and deviation vector as required. You can move the positions of the graphic icons at will or arrange them automatically in the margin.

For serial evaluations, you can save the position of the graphic icons. The icons will then be output at the defined position during the next evaluation of the same measuring run.

Further information on this is given below and in the chapter containing the descriptions of the Extras menu.

marking

The deviation of a measured point is displayed as a colored square around the measured point. The size of the square will be automatically adjusted, although it will always be clipped at a patch boundary. The color corresponds to the tolerance zone position.

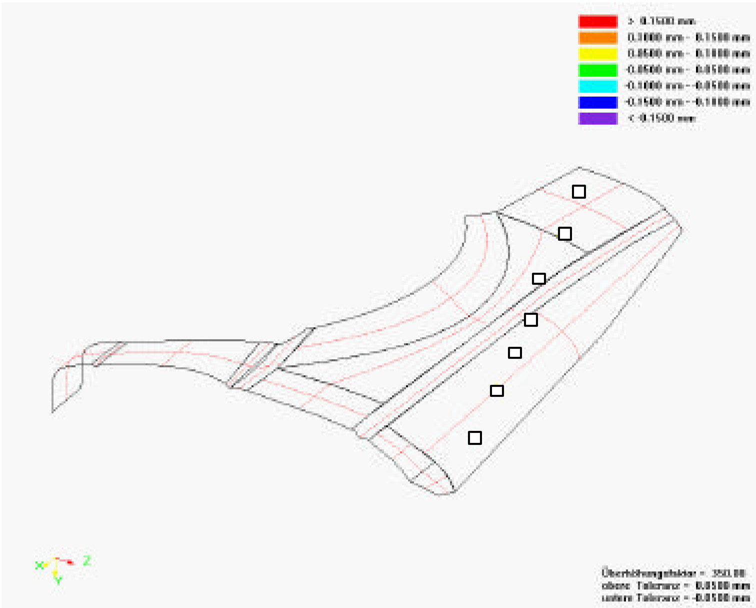


Figure 13-5

The marking is suitable both for manually probed points and for points that have been probed in a CNC run.

marker(NT)

The deviation of a measuring point is represented as a colored marker. The marker type is defined in the <Markings> parameter page.

Definition of parameters

tripod(NT)

The deviation of a measuring point is represented as a "tripod", i.e. you will obtain a vectorial representation of the deviation in all three coordinates.

The size of the vectors can be set using the magnification factor.

magnification factor

The deviation can be displayed magnified vectorially, chromatically and as a section. This factor specifies the magnification.

chromatic representation filled

During evaluation of measuring runs the deviation can be represented in chromatic coordinates. In contrast to the above-described marking the chromatic coordinates yield, through interpolation, a continuous colored surface. The surface consists of triangles. These triangles can be filled or not filled.

depth (NT)

This value specifies the depth for the chromatic deviation representation.

🕮 See Chapter 9.1.2, Evaluation with chromatic coordinates.

extended output (UX)

In the graphic record, the following are also output for evaluations:

min./max. deviation, form error, standard deviation, number of evaluated points, upper and lower tolerance.

Best Fit (UX)

The result of the last performed 3D best fit is also displayed in extended output.

13.2 Parameters for the clipping plane

You can define a plane for the representation, in order to “clip” the model, which will then enable you to define measuring points on internal sections.

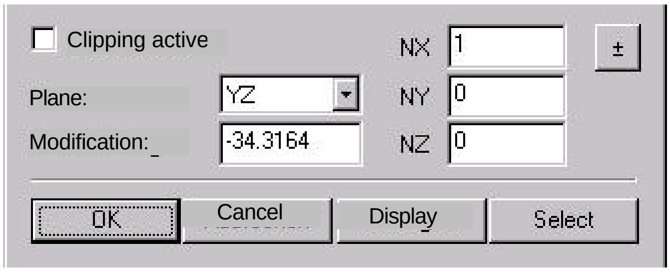


Figure 13-6

Clipping active

Activates the clipping function for the display. The view is updated by pressing the **"Display"** key.

To rotate the model to the other side, use the **"+/-"** key.

Plane

Define the cutting plane from a main plane.

NX / NY / NZ

Normal vector of the cutting plane. Preselect the vector movement in the normal direction.

Movement

Enter an amount by which movement is to occur in the normal direction, in relation to the zero point.

Instead of directly entering the movement, you can also press the **"Select"** key and either directly select one or three points in the model. If one point is selected, the program calculates the movement, if three points are selected, the entire plane is selected for cutting.

13.3 Parameters for the FACE isolines

Trimmed surfaces are standardly displayed only with their boundary curves. If you require more graphic information, you can switch on isolines for trimmed surfaces via this option.

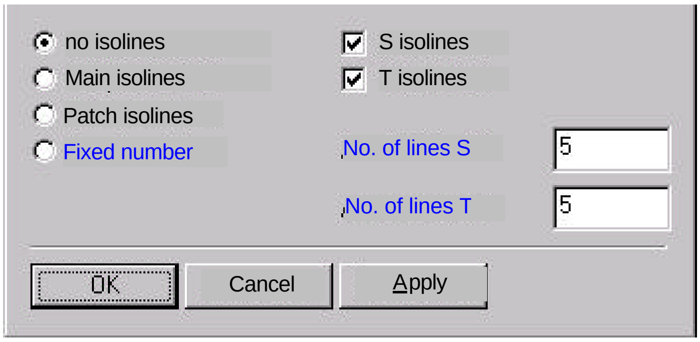


Figure 13-7

Main isolines

The main isolines of a FACE are displayed.

Patch isolines

The main isolines of all patches which run under the corresponding FACE are displayed.

Fixed number

Set a fixed number of isolines for a FACE. Enter the number of isolines in the accompanying input fields.

S- / T-isolines

Switch the isolines for the S and T parameter direction on / off.

No. of lines S / T

If you select "Fixed number", you can preset the number of lines in the S- and T-direction.

13.4 <Rendering> function parameters

With the parameters for the rendering function you define the parameters for the illuminated and colored display of the workpiece surface. In this display, hidden edges of the workpiece geometry can be overlaid.

13.4.1 Rendering Parameter

Operation

To define the parameters for the rendered display click on the <Rendering>function and the "Parameters" panel (NT), or the <Rendering> - <Rendering parameters> functions (UX).

A window appears for parameter definition.

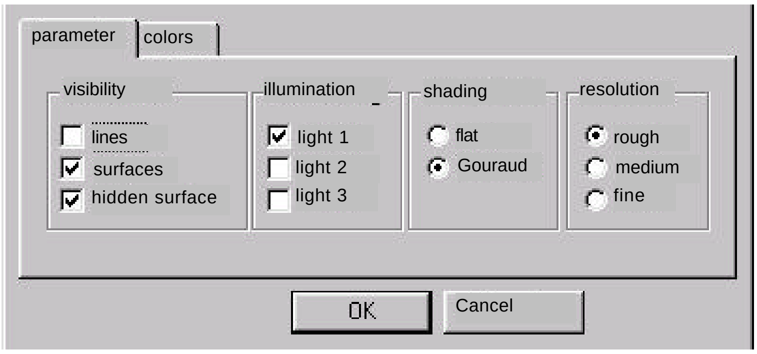


Figure 13-8

Lines

The lines of the calculated triangles for the rendering are graphically displayed on the workpiece surface (wire model). They can also be displayed with a switched on surface model.

Surfaces

The workpiece surfaces are displayed graphically (surface model). They can also be displayed with a switched on wire-frame model.

hidden surface

Hidden edges are removed.

illumination

The light source can be switched on and off with this parameter. A total of three light sources can be defined, which are staggered over an angle of 120 degrees above the workpiece.

shading

This parameter defines the illumination model of the graphic display.

resolution

With this parameter you define the resolution of the displayed workpiece surface. A coarse resolution is sufficient as a rule. In some cases a higher resolution may be necessary. A higher resolution means a larger number of points and, as a result, increased computing times.

Read data (UX)

The data of the last rendered representation are applied.

Automatically save data (UX)

The data calculated for rendering are automatically saved on the hard disk.

OK

This function closes the window and displays the rendered representation on the screen.

Cancel

This function closes the parameter window.

Reset (UX)

The rendering function is deactivated via the reset function. The memory required for calculation for the rendered representation is released.

Save (UX)

Save the data for the rendered representation on the hard disk.

13.4.2 Rendering colors

With this parameter you define the respective colors for the illumination model.

Operation:

Click on the <Rendering> function and the "Colors" panel (NT), or the <Rendering> - <Rendering colors> functions (UX).

A window appears for defining the colors of the elements.

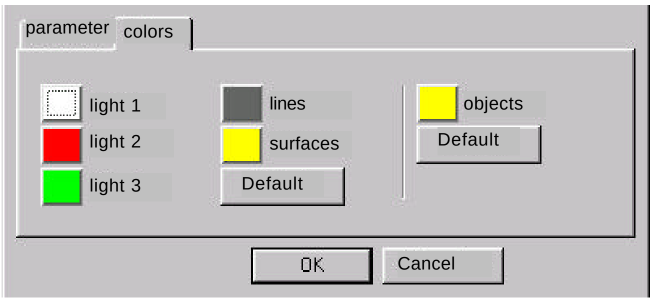


Figure 13-9

In order to change the color for a defined element, click on the corresponding element.

The <Default> key always gives you the preset color value for the element in question:

lines:	grey
surfaces:	yellow
light 1:	white
light 2:	red
light 3:	green

NT: A further window appears for defining the colors.
UX: Element and color selection are located on a panel.

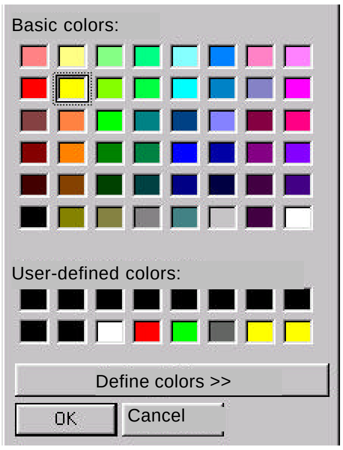


Figure 13-10

Then define which color is to be assigned to the element in question. Click on the corresponding color.

NT: Click on <Define colors>, if the color range is not sufficient and you wish to select another color.
Define a new color and click on <Add color>.

Confirm the selection with <OK>.

13.5 Parameters for markings

This function defines the symbols with which different markings are graphically displayed.

Operation:

Click on the <Markings> function.

A window appears for defining the markings.

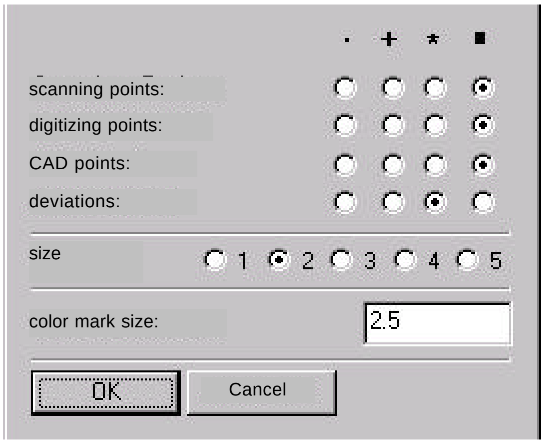


Figure 13-11

Select the required markings and confirm with <OK>.

NT: You can also define the size of the "markers" and that of the color marks in "Marking".

13.6 Measuring run parameters

Measuring run parameters are linked both to a model and to the subgroups of a model. If a model or a group is loaded, the associated measuring run parameters will be activated. If up until now, no parameters have been defined for a model or a group, the last parameters that were defined will still be active.

> ***NOTE***
>
> If you define one of these parameters, in principle you overwrite the definition in the CADLINK module of the UMESS measuring software. If you want to utilize the parameters from CADLINK, you are not permitted to define any parameters here!

The measuring run parameters are specified with the <Measuring run> function.

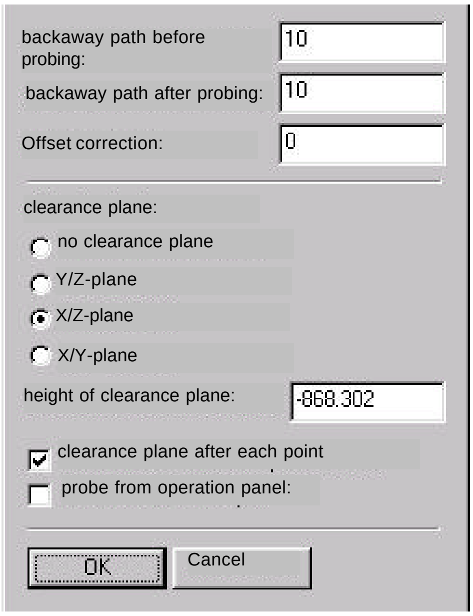


Figure 13-12

Definition of parameters

Backaway path before probing

Distance of an intermediate position from the target position. The intermediate position lies ahead of the probing. Vectorial probing will take place from this position.

Backaway path after probing

Distance of an intermediate position from the target position. The intermediate position lies after the probing. Vectorial backaway will take place up to this position.

Offset correction

Correction value for offset surfaces, e.g. to take into account sheet thicknesses. The sign of the offset value determines the direction for the correction.
Offset positive: Correction in direction of surface normal.

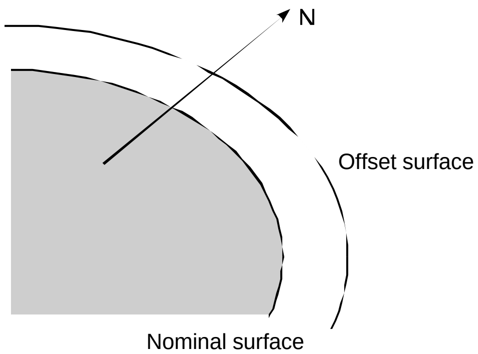


Figure 13-13

Offset negative: Correction in opposite direction to surface normal.

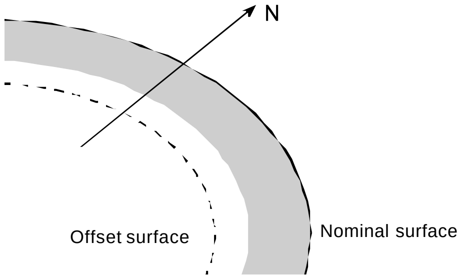


Figure 13-14

Clearance plane

To avoid collisions during CNC operation, the coordinate measuring device travels to a clearance plane at the end of a measuring run. Depending on the setting, this can be after each surface or after each point. This clearance plane should be specified here. The basic planes XY, YZ and ZX of the workpiece coordinate system are the options.

> **NOTE**
>
> The clearance plane is a plane in the workpiece coordinate system, not a plane in the machine coordinate system

You can define any plane you like with the <Clearance plane in space> function in CADLINK .

Height of clearance plane

Enter the value for the height of the clearance plane.

The height of the clearance plane is always measured from the origin of the workpiece coordinate system, not as the distance to a probing point.

You can also define a clearance plane graphically on the screen.

⇘ Select the display of clearance planes in the representation parameters.

- Using the clearance plane parameter screen, define which clearance planes are active and their distance from the workpiece.
- Click on the cross in the center of the relevant clearance plane.
- The clearance plane is adopted and entered into the parameters for the measuring run.

Clearance plane after each point

If you select this option, the coordinate measuring machine will travel into the clearance plane after each probing point.

Probe from operation panel(NT) / from UMESS (UX)

If you select this option, a measuring run will not use the probe that you have defined when programming the measuring run, but always the probe set at the operation panel or the probe defined in UMESS.

13.7 Evaluation parameters

Different parameters can be defined for evaluating measuring results.

13.7.1 Enter tolerances

Tolerances are linked to a model and to the subgroups of that model. When a model or group is loaded the associated tolerances are also selected. If no tolerances have yet been defined for a model or a group, the last defined values will still be active.

The <Tolerances> function allows you to enter the upper and lower tolerance values for the evaluation of the measuring runs.

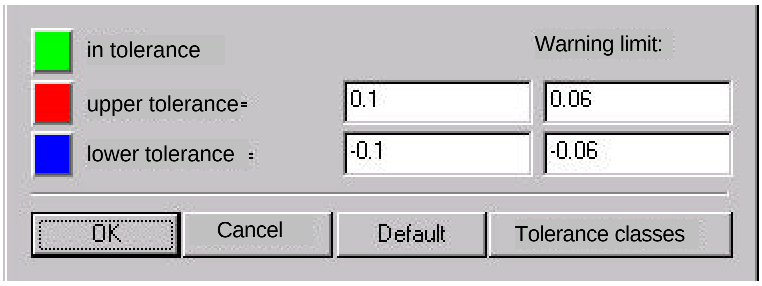


Figure 13-15

Enter negative tolerances together with their sign.

Warning limit (NT)

You can specify values within your selected tolerance band as the warning limit. Deviations in the deviation window within the warning limit are displayed in green. Deviations between the warning limit and the input tolerance are displayed in a different color.

If you wish to evaluate deviations without a warning limit, enter the same value for the warning limit as for the tolerance.

13.7.2 Tolerance classes (NT)

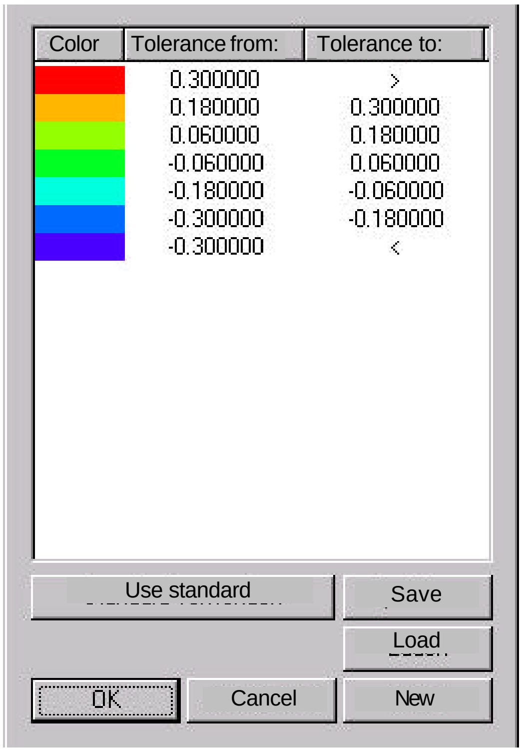


Figure 13-16

Color

Double clicking on a color opens the color selection.

Tolerance from/to

Double clicking on a value in the "Tolerance from" column opens a window for entering values. "Tolerance to" is automatically adjusted and cannot be modified.

Standard

"Use standard" restores the standard setting.

Save / Load

You can create different tolerance classes and save them under a separate file name with the ending .tol. Saved tolerance classes can be reloaded at any time.

New

Opens an input window for defining the number of tolerance classes and the tolerance range:

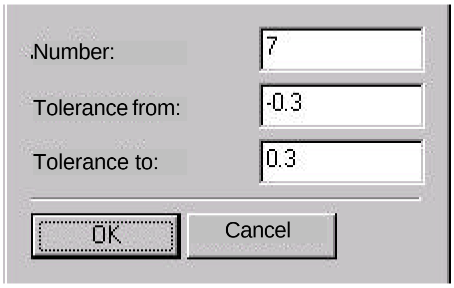


Figure 13-17

13.7.3 Tolerance classes (UX)

This function serves to display deviations in chromatic coordinates in tolerance classes. Areas with the same tolerance zone position are shown in the same color.

You can define a minimum or three and a maximum of 21 tolerance classes. The average tolerance class is always limited by the lower and upper tolerance.

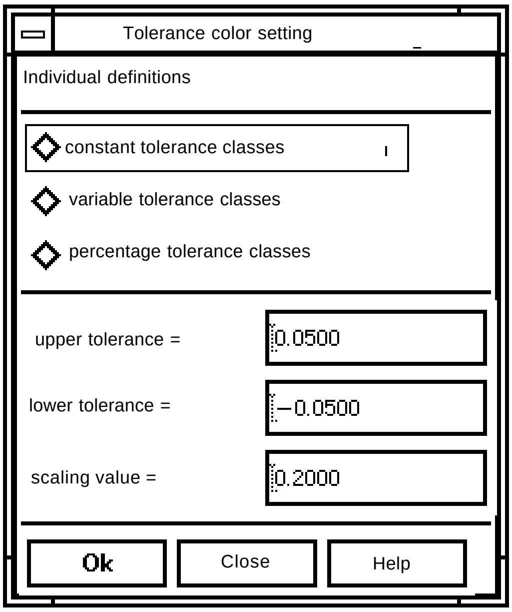


Figure 13-18

You can define the following types of tolerance classes:

Constant tolerance classes

The tolerance class width is entered in millimeters or inches and is identical for each tolerance class.

Example:	Tolerance width = 0.1 mm
Class n	from 0.2 ... 0.3 mm
Class n+1	from 0.3 ... 0,4 mm
Class n+2	from 0,4 ... 0,5 mm

Variable tolerance classes

The tolerance class width is entered in millimeters or inches and can be any size for each class.
The only restriction is that all values within the first and last class must be covered.

Example:

Class n	from 0.2 ... 0.3 mm (class n+1 *must* start at 0.3 mm)
Class n+1	from 0.3 ... 0.6 mm
Class n+2	from 0.6 ... 0.9 mm ...

Percentage tolerance classes

The tolerance class width is entered in millimeters or inches and depends on the upper and lower tolerance.
If you enter 100 as the percentage factor, all tolerance classes are set with a width of 1 * tolerance.
If you enter 30 as the percentage factor, all tolerance classes are set with a width of 0.3 * tolerance.

Example: upper tolerance 1,0; percentage factor = 100

Class (n-1/2)	from lt ... 1,0 mm
Class (n-1/2)+1	from 1.0 ... 2.0 mm
Class (n-1/2)+2	from 2.0 ... 3.0 mm

Example: upper tolerance 1.0; percentage factor = 30

Class (n-1/2)	from lt ... 1.0 mm
Class (n-1/2)+1	from 1.0 ... 1.3 mm
Class (n-1/2)+2	from 1.3 ... 1.6 mm

Definition of parameters

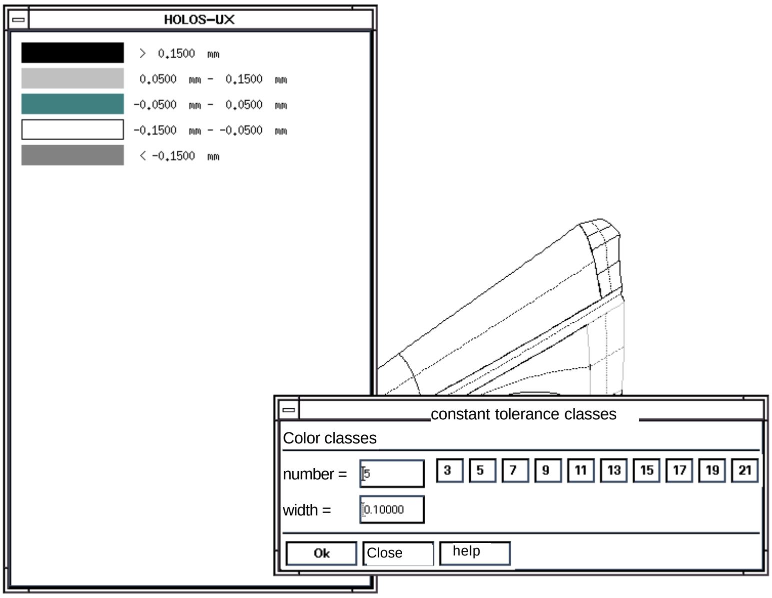


Figure 13-19

Scaling value

When measuring individual points, deviations are output in a deviation diagram as well as in the graphic (tolerance band with deviations). If you have preset both the upper and the lower tolerance with "0.0", then you must state another dimension for graphic representation of the deviation. This is the scaling value. This is the scaling value.

13.7.4 Parameters for evaluation

Different parameters can be defined for evaluating measuring results.

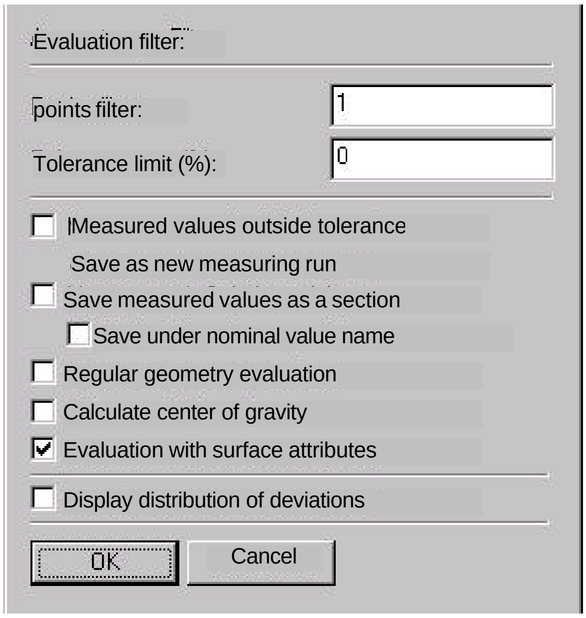


Figure 13-20

points filter

The value defines the distance of the measurement points to be evaluated.
A value n means that each n-th point is evaluated.

Tolerance limit

The value defines the tolerance limit as a % during evaluation of the measurement results.
The value "0" means that all points are evaluated. The value "100" means that only those points are evaluated which lie outside the defined tolerances.

Measured values outside tolerance

If you select this option, all measurement values which lie outside the tolerance limit entered above are saved as a new measuring run.

Save measured values as a section

Select this option in order to save the evaluated values as a section. It can then be used to evaluate sections.
Save under nominal value name:
When a section is saved, no new file name is generated, but the section evaluation is saved under the name of the nominal value file. If this file already exists, then it will be overwritten.

Regular geometry evaluation

With this option, when measured values are evaluated, the regular geometry elements are automatically also evaluated. Regular geometries are elements that are recognized exclusively as regular geometry elements in HOLOS.

Calculate center of gravity

Switch on this option if you also wish to calculate the center of gravity of the measured values when calculating the deviations. It will be output in the measuring record.

Evaluation with surface attributes

The values entered under "Objects" - "Attributes" are used for evaluation. Without this option, the general settings apply.

Display distribution of deviations

With this option, the distribution of deviations is displayed during an evaluation, as for manual measurement.

13.7.5 Parameters for graphic icons

The graphic icon parameters define how the icons for the deviation representation are displayed on the screen.

Graphic icons for regular geometries

Use this dialog window to define the parameters of the graphic icons for the deviation representations of regular geometries.

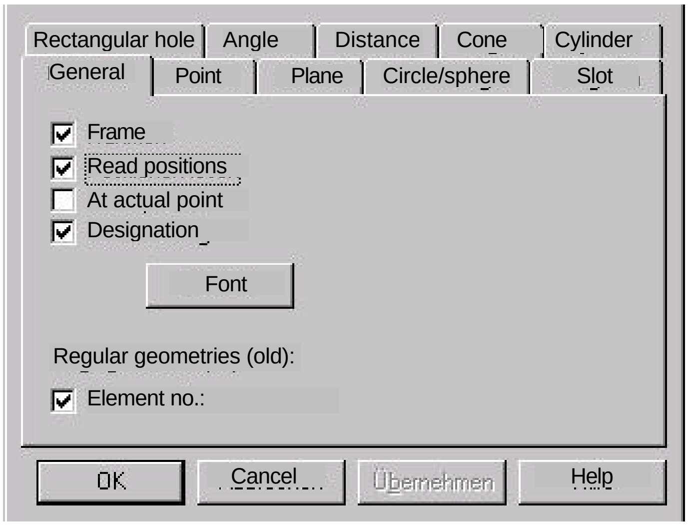


Figure 13-21

⇘ Define the representation of the measuring section by activating the relevant options:

- with / without frame
- read in and observe saved position
- at actual point
- with / without designation, i.e. point or element name
- with / without element number: the selection applies for regular geometries which have been measured from a CNC-run.

⇘ Click on <OK>, if you want to adopt the settings.

Definition of parameters

Example: Point

The index card is used to define the parameters of the graphic icons for representing the deviations of individual points.

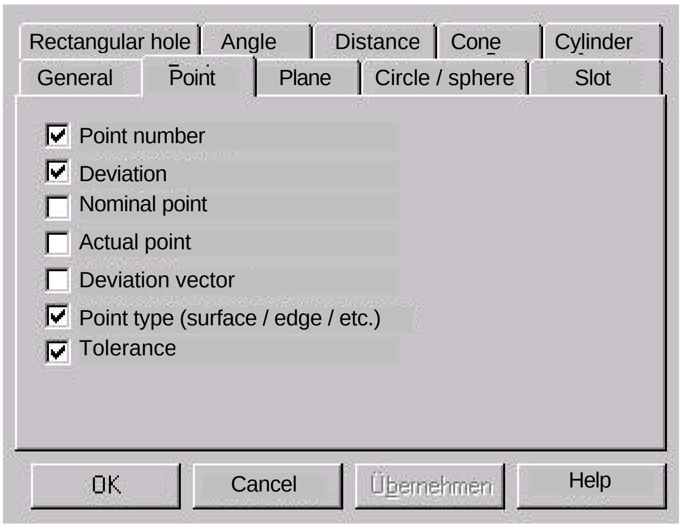


Figure 13-22

Point number Display the number of the measured point.

Deviation Display the deviation of the measured point.

Nominal point Display the nominal value of the measured point.

Actual point Display the actual value of the measured point.

Deviation vector Display the deviation vector for the measured point.

Point type Display the type of measured point:
F = surface points
K = edge points.

Tolerance Tolerance display: value within, above, below tolerance.

Save positions (UX) You can move graphic icons at random on the screen or arrange them automatically at the edge of the screen.
With this option, you can save the defined positions for individual measuring runs. During the next evaluation the graphic icons will automatically be displayed in the defined positions again, if the measuring runs based on the corresponding sequence and the <Read positions> option is switched on.

Font(NT) This function allows you to set the typeface and the type size for the texts in the graphic icons. Do not use any proportional typefaces, as otherwise the text formatting will be lost.

Click on <OK>, if you wish to apply the settings.

13.7.6 Chromatic representation (NT)

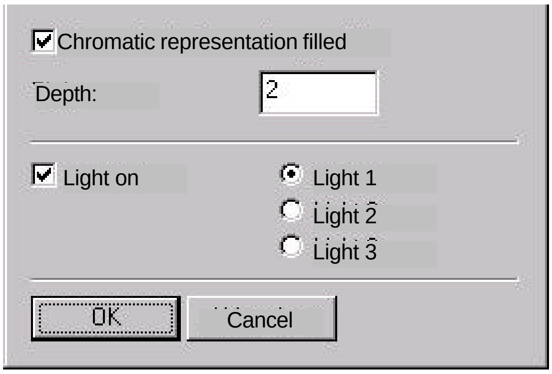


Figure 13-23

Set how the actual data are to be represented as chromatic coordinates during the evaluation.

filled

The meshed triangles are displayed not just as frames, but filled.

depth

The depth defines the number of triangles that are to be used for division.

light

You can select an illuminated display and define a total of three light sources which are offset by an angle of 129 degrees over the workpiece.

13.8 Patch identification parameters

"Patch identification" denotes the following process: during probing of a known workpiece the nearest patch will be searched for in a defined area around the probing point. This search area takes the shape of a sphere around the probing point. The parameter you specify is the radius of this sphere. On system startup the setting for the search area has a default setting of 5 mm. If no patch can be found in the search area the probing will be rejected.
Use the function <Patch ident> to enter the search area.

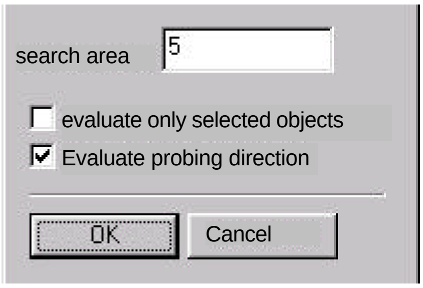


Figure 13-24

evaluate only selected objects

During manual probing of points on the workpiece surface, a surface segment which contains the measurement point is automatically determined by HOLOS.
To do this, HOLOS defines the surface segment with the shortest distance to the probed measurement point.
If only particular objects (surfaces, patches or faces) are to be taken into consideration when defining a surface segment, select this option.

Prerequisites

The selected objects must be at the location of the chosen points. Otherwise no evaluation can be performed.

Then select the objects which are to be taken into consideration in the evaluation and probe points on the workpiece surface. When determining the deviation only the selected objects are taken into consideration.

Evaluate probing direction

During manual probing with a coordinate measuring device, in addition to the position of the measured point, the probing direction is also transferred. By evaluating this probing direction the system can determine whether or not the orientation of the found surface is correct. If this is not the case the orientation will be changed automatically.

When using with diverse optical systems the transfer of a valid probing direction cannot always be guaranteed. For such cases, the evaluation of the probing direction must be turned off to prevent incorrect results being obtained.

The precondition for getting correct measuring results remains however, that the orientation of the surfaces is correct!

13.9 Defining probes

The <Probe> function specifies the probe, with which the subsequently generated measuring runs will be measured.

Only probes that are currently active can be selected.

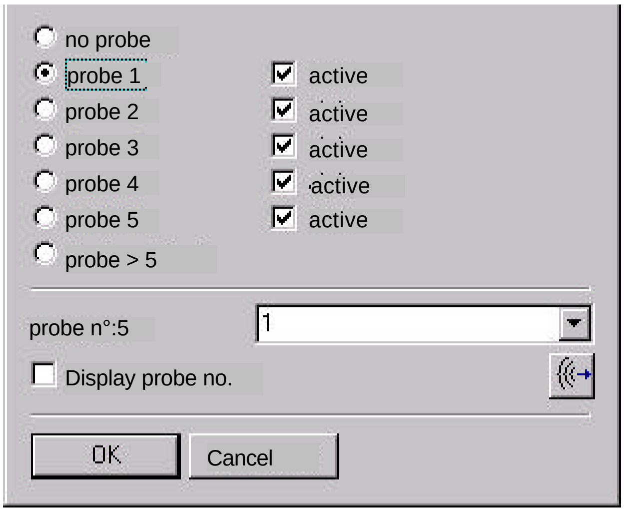


Figure 13-25

Probe selection

If no probe is to be linked to particular measuring runs, select the option "No probe". In the measuring run, the probe that is set up at the control console will then be used.

A color is assigned to each probe. The nominal points from the measuring runs will be displayed in this color. If no probe is specified the nominal points will be displayed in white.

You can also select the probes using the symbol button on the graphics menu bar.
📖 See Chap. 3.10

Probe entry

When working with "Calypso", you can manually enter a probe number greater than 5. However, only the actually calibrated probes are contained in the selection list.
In the case of DSE, the A- and B-angles are also displayed.

Display probe no.

The probe number is displayed at the nominal point.

Transmission symbol

Carry out probe change on the machine.

13.10 Parameters for 3D best fit (UX)

With the <3D best fit> function you can activate or block degrees of freedom for the 3D best fit.

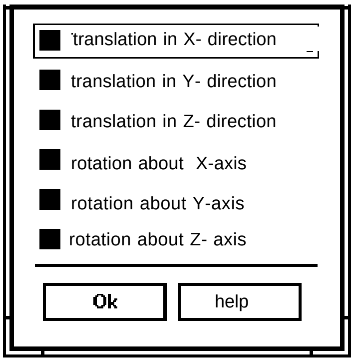


Figure 13-26

If the display field is light colored, the degree of freedom is active, i.e. translation or rotation is permitted during the best fit. If the display field is dark, the degree of freedom is blocked, i.e. translation or rotation is not permitted.

Significance of the degrees of freedom

In the case of rotationally-symmetrical and translation-symmetrical workpieces, the "rotation" and "translation" degrees of freedom must be restricted. In the case of a hemisphere, rotation around the middle axis must be blocked, for example, as otherwise, when calculating the transformation there would be an infinitely large number of equivalent results and the program would not be able to determine the optimal transformation.

13.11 Controlling output of the measuring record

13.11.1 Output of the measuring record

The <Measuring record output> function specifies where your measuring records will be output after they have been generated. A measuring record will be generated as soon as you evaluate the measured values with the functions of the <Evaluation> menu.

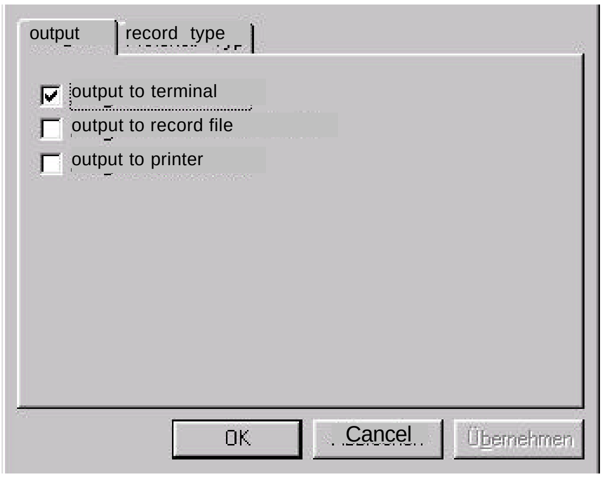


Figure 13-27

Output to terminal

The measuring records are output on the screen. They can be listed on the screen by using the <Measuring record> function in the <Evaluation> or <Patch ident> menus and then printed out from this window. Since the records are not saved for the output to the screen, you can only ever look at the record of the last evaluated measuring run.

Output to printer

The measuring records are output to the printer that is installed in the system. This takes place automatically when a measuring record has been generated using the functions of the <Evaluation> menu.

Output to record file

The measuring records are output to files. The files are automatically designated with the name "protocol_x", where x is a consecutive number. The record files can be displayed later in the above window with the <Print record file > function.

Print record file

With this function you can print any record file at a later point in time. A list of all stored record files appears for selection of the file. When you have made your selection and confirmed with <OK>, the file is printed out on the installed printer.

13.11.2 Measuring record type

Click on <Record type> in the parameter window for measuring records.

The dialog screen for the record type is displayed:

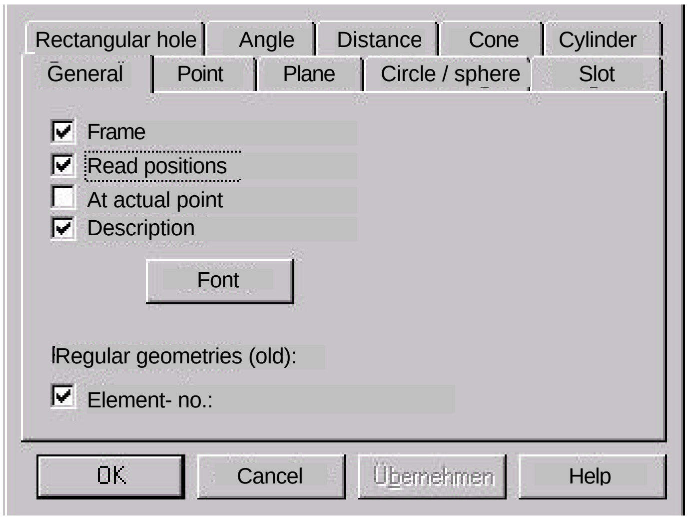
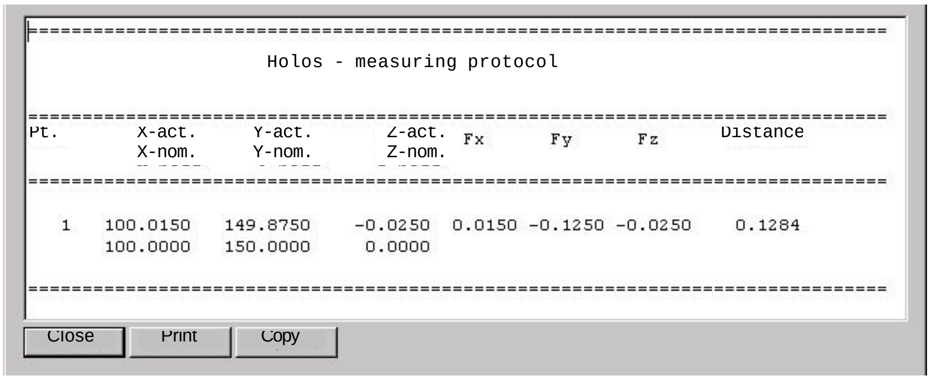


Figure 13-28

Protocol 1 Standard record output. You can use the <Display> function to show what the measuring record looks like.

Definition of parameters



Holos - measuring protocol

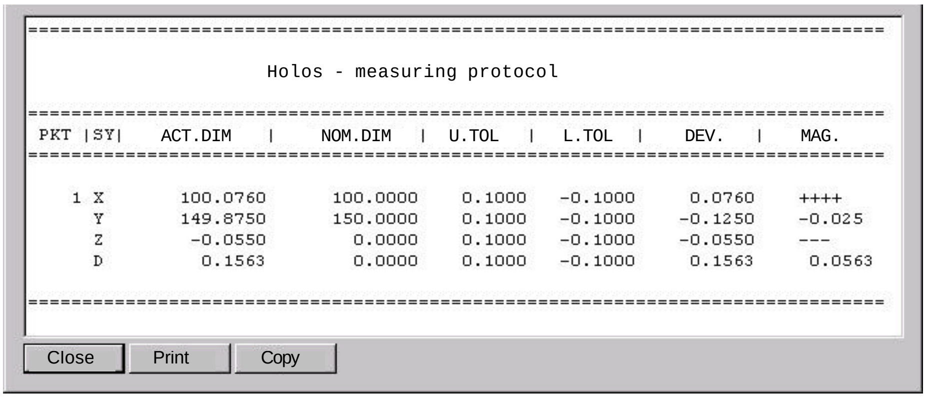
Pt.	X-act. X-nom.	Y-act. Y-nom.	Z-act. Z-nom.	Fx	Fy	Fz	Distance
1	100.0150 100.0000	149.8750 150.0000	-0.0250 0.0000	0.0150	-0.1250	-0.0250	0.1284

Figure 13-29

Protocol 2

For this record type, you can define the values to be output in the measuring record yourself:

Output X-coordinate	The values for the X-coordinates.
Output Y-coordinate	The values for the Y-coordinates.
Output Z-coordinate	The values for the Z-coordinates.
Output deviations	The values for the deviations.



Holos - measuring protocol

PKT	SY	ACT.DIM	NOM.DIM	U.TOL	L.TOL	DEV.	MAG.
1	X	100.0760	100.0000	0.1000	-0.1000	0.0760	++++
	Y	149.8750	150.0000	0.1000	-0.1000	-0.1250	-0.025
	Z	-0.0550	0.0000	0.1000	-0.1000	-0.0550	---
	D	0.1563	0.0000	0.1000	-0.1000	0.1563	0.0563

Figure 13-30

Protocol 3 (short)

Only the point number, nominal point and deviation are output:

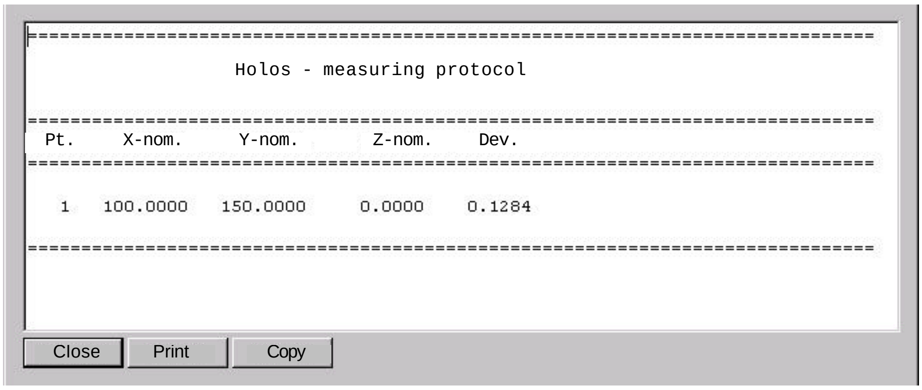


Figure 13-31

13.12 Create record head

With this function you define the entries for the record head of your measuring records. You can choose between a standard record head and a self-defined record head.
The record head will be used
- for the measuring records
- for output of the graphic window (<Screen> and <Plotter> functions on the graphic menu bar).

The record head is linked to a model and to subgroups of that model. When you load a model or a group the associated record head will be selected. If you have not yet defined a record head for a model or group, the last one that was defined will still be active.

13.12.1 Standard record head (UX)

With this function you define the entries for the standard record head.

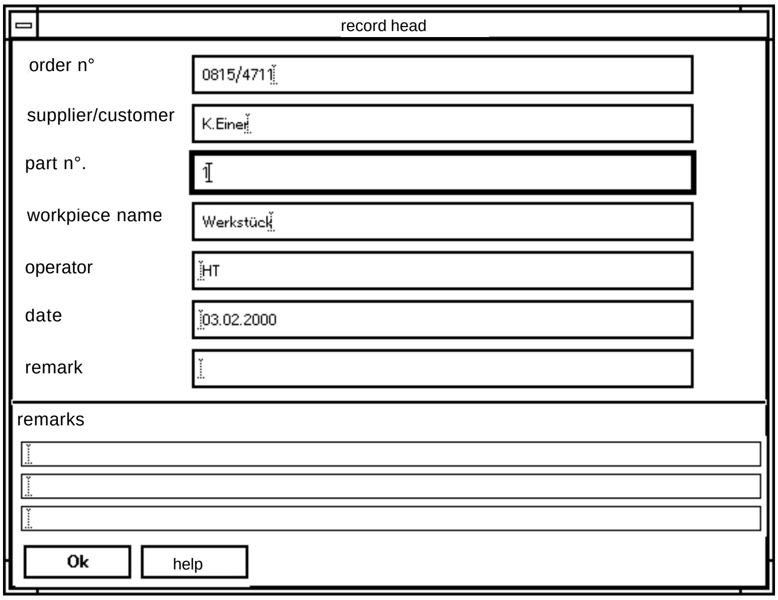


Figure 13-32

Enter the relevant entries for the record head into the text fields and click on <OK> in order to adopt the entered values.

13.12.2 Standard record head (NT)

Use the "Standard" option and the "Record variables" key to define the entries for the standard record head.

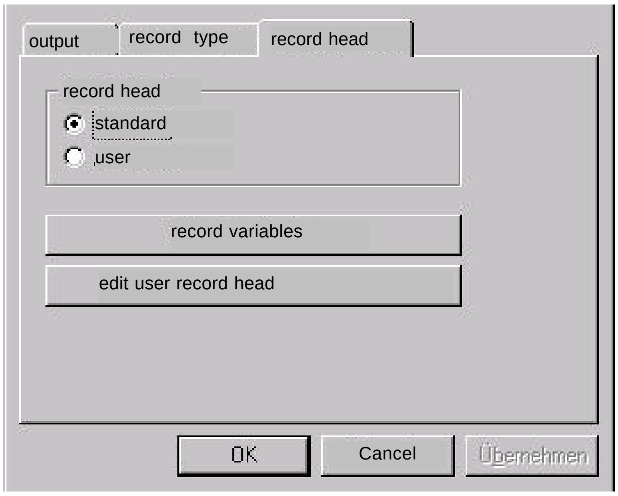


Figure 13-33

The record variables are displayed

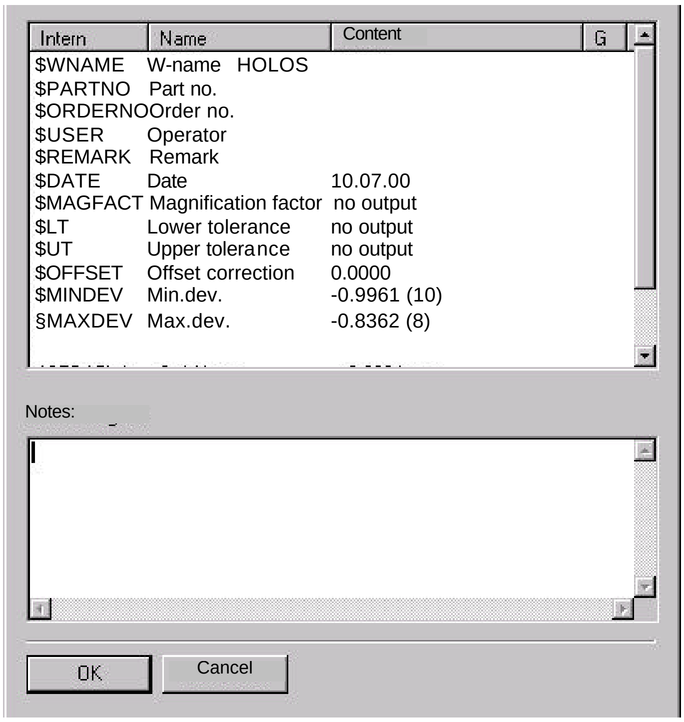


Figure 13-34

Except for the last eight variables, you can edit all protocol variables:
Double-click in the "Name" and "Content" columns to overwrite the entries. You can define which variables are output in the record head.

In column G you define whether a variable is global by clicking on it. It is then marked with *. Global variables are not overwritten by macros. For example, you can define the operator as the global variable here, so that he will be entered everywhere.

In the "Remarks" column, you can enter up to 256 text characters.

NOTES

The variables can be integrated into the graphic record output as text elements, and can be applied to the user record head.
Additional variables for the record head can be defined:
see Appendix B.2

13.12.3 User record head

After activation of this function a dialog editor appears for definition of a self-defined record head. This record head is used for the output of measuring records, if you have activated the output of a user-defined record head in the output parameter page.

Click on the <Record head><User record head> function.

A dialog editor appears for entering the record head.

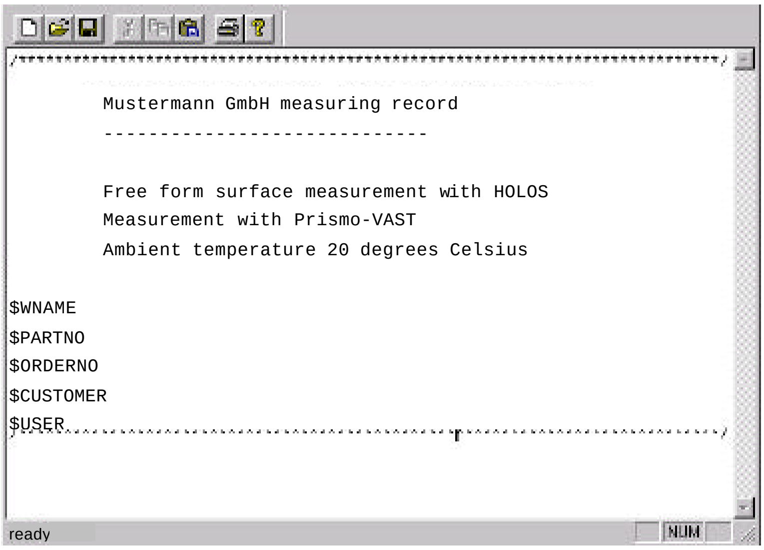


Figure 13-35

Enter the required entries in the editor and confirm with <File>-<Save>.

If you want to apply the entries from the standard record head to your self-defined record head, you can define this by giving various keywords.

The keywords must always begin at the first column in a record head line and be given in their exact spelling, since otherwise they will not be recognized by the system. The first character is always a $.

Keywords

$ORDERNO	accepts the entry for the order number
$CUSTOMER	accepts the entry for the supplier/customer
$PARTNO	accepts the entry for the part number
$WNAME	accepts the entry for the workpiece name
$USER	accepts the entry for the operator
$DATE	accepts the entry for the date
$REMARK	accepts the entry for the remarks

Additionally, in the NT version:

$MAGFACT	accepts the entry for the magnification factor
$LT	accepts the entry for the lower tolerance

Definition of parameters

$UT	accepts the entry for the upper tolerance
$OFFSET	accepts the entry for the offset correction
$MINDEV	accepts the entry for the min. deviation
$MAXDEV	accepts the entry for the max. deviation
$STDDEV	accepts the entry for the std. deviation
$FORMERR	accepts the entry for the form error

The keywords are displayed in the record head editor with their respective spelling. The relevant entry then appears in the record itself:

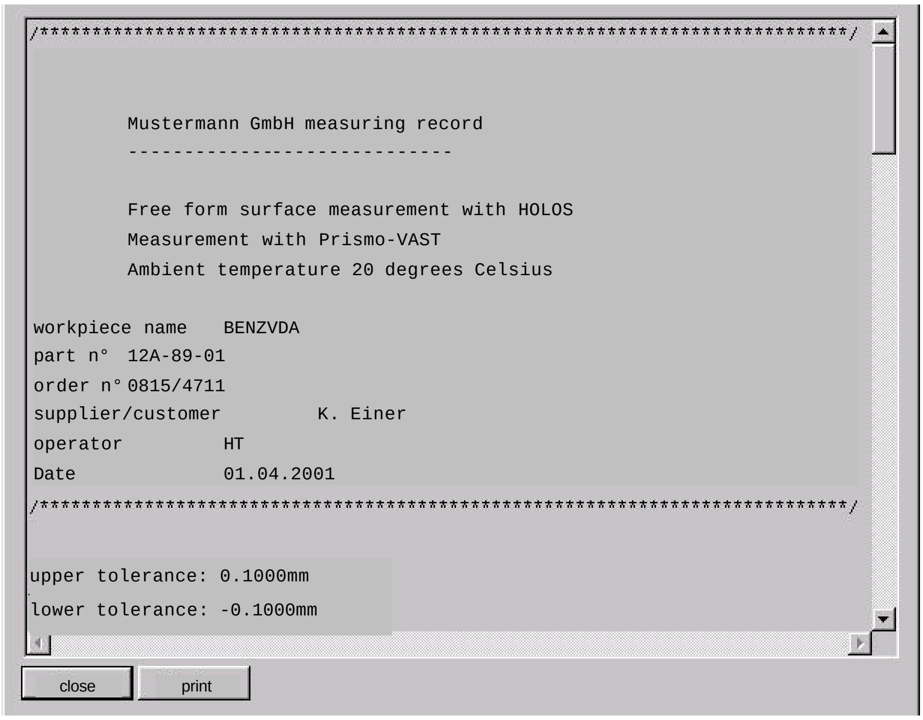


Figure 13-36

13.13 Clearance planes

Define the parameters for the clearance planes.

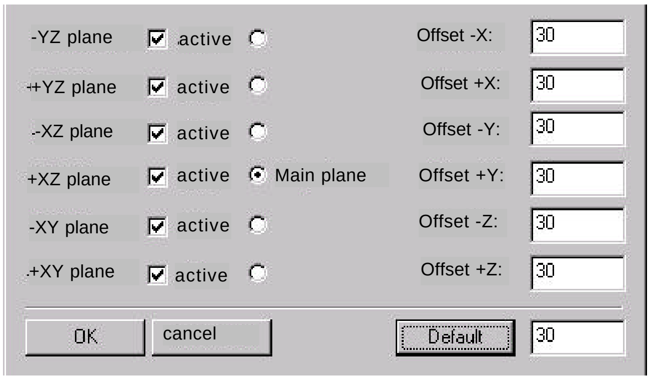


Figure 13-37

Offset

The value for the offset of the clearance planes defines the distance of the respective clearance planes to the extreme values of the workpiece surface.

You can also enter a standard value and click on the "Default" key, in order to configure an entire column with this value.

Activate/Deactivate clearance planes

If you wish to prevent unintentional selection of a clearance plane which cannot be reached in a measuring run (e.g. the clamping surface of a workpiece), deactivate this clearance plane.

The options have the following meaning:

-YZ-plane	YZ-plane in direction of the negat. X-axis	(left)
+YZ-plane	YZ-plane in direction of the posit. X-axis	(right)
-XZ-plane	XZ-plane in direction of the negat. Y-axis	(front)
+XZ-plane	XZ-plane in direction of the posit. Y-axis	(back)
-XY-plane	XY-plane in direction of the negat. Z-axis	(bottom)
+XY-plane	XY-plane in direction of the posit. Z-axis	(top)

> ***NOTE***
>
> An inactive clearance plane will no longer be displayed on the screen and can therefore no longer be selected. It will then no longer be taken into consideration when generating diversions.

Main plane

A main plane always defines the clearance plane which is used to generate the diversion, in cases where parallel clearance planes are defined for two consecutive measuring runs.

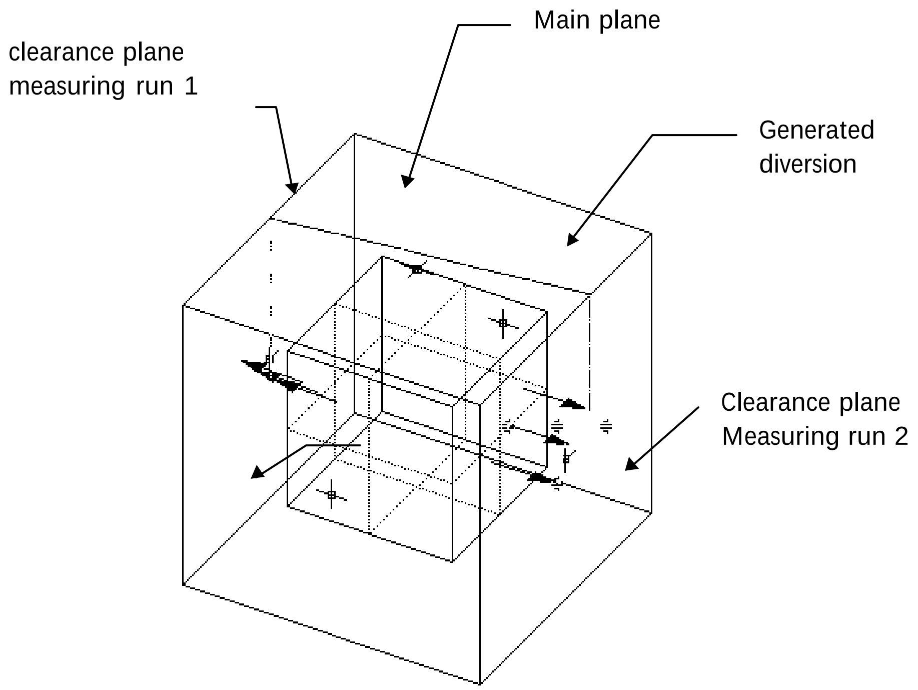


Figure 13-38

You define a main plane by pressing the switch behind the relevant clearance plane.

The diversion route between two parallel clearance planes is **always** generated via the main plane.
Main planes are marked by yellow crosses.

CAUTION

If no main plane is defined (option "no main plane") and the clearance planes are parallel, then there is a danger of collision!

In the case of clearance planes which are located at right angles to each other, the by-pass route extends over the edge.

⇘ Click on the relevant option to define which clearance planes are active. If, for example, a workpiece is clamped with the underside on the work plate of the coordinate measuring device, you should deactivate the -XY-plane.

⇘ Enter the values for the offset of the clearance planes. The offset defines the distance of the clearance planes from the workpiece surface in the respective direction.

⇘ Click on <OK> in order to accept the entered values

Clearance planes can be graphically displayed on the screen. By clicking on a clearance plane, the values are adopted into the measuring run parameters.

13.14 Digitizing parameters

The <Digitize> function allows you to set the parameters for the digitizing functions. These parameter settings are only relevant if you have installed the Digitization option on your system.

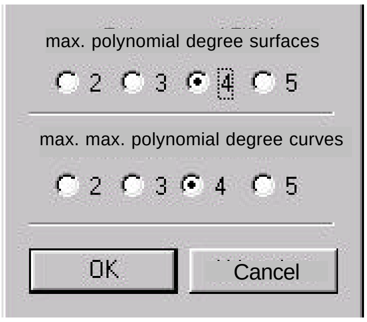


Figure 13-39

Polynomial degree

- Define the polynomial degree for surfaces and curves using the options shown above.
- Click on <OK> in order to accept the settings.

The polynomial degree of curves and surfaces is first of all dependent on the number of probed points:
Number of points - 1 = Polynomial degree

The polynomial degree does however have an upper limit. This upper limit must be selected for the max. polynomial degree (the term polynomial degree is explained in the glossary).

Scanning - line distance (UX)

When scanning areas with the LTP60 laser triangulation probe, lines are scanned in a meandering fashion. The “line distance” parameter defines the distance between these lines in mm.

13.15 Printing (NT)

Define the parameters for printing graphics and measuring records.

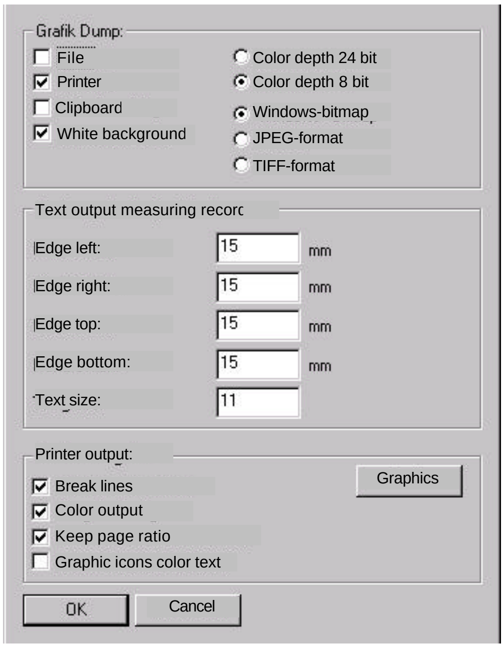


Figure 13-40

Graphics dump

Here you define the parameters for graphics output. You can output graphics by clicking on <Graphics dump> in the graphics menu bar with the left mouse button.

File The graphic is output in the defined format.

Printer The graphics are output to an installed printer.

Clipboard The graphics are output to the clipboard. From the clipboard, the graphics can be inserted into any application under Windows-NT that supports graphics (Paint, Word etc.).

White background The background is not printed in the background color. Activate this option in order to save printer color.

Color depth This option defines the color depth for the graphics to be output: For 8-bit output in 256, for 24-bit in 65536 colors.

Measuring record text output

You can use these parameters to set the margins and font size for outputting numeric measuring records, if the measuring records are output directly to the printer.

Printer output

Here you define the parameters for outputting the graphic record.

break line Print a broken line for ISO and raster lines.

color output The text of the graphic symbol is output in color.

keep aspect ratio The screen display is exactly applied during printing. If this option is not selected, automatic optimization takes place to fully utilize the page area.

graphic icons text color The text in the graphic icons can be displayed in color or black-white.

"Graphics" key

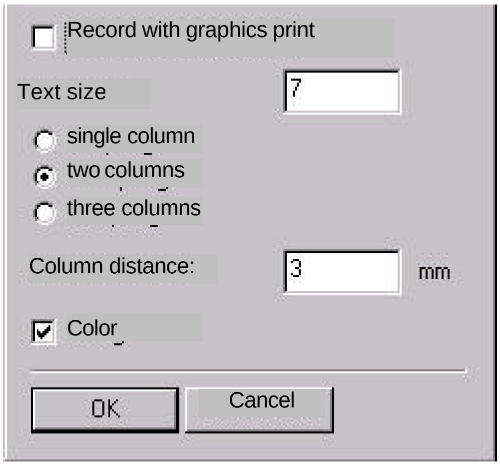


Figure 13-41

You can output the text record as the following page, together with the graphics. You must define the font size, number of columns and distance between columns.
With the "color" option, values within tolerance are e.g. printed in green, values above in red and values below tolerance in blue.

13.16 Printer management (UX)

Printers and plotters which HOLOS-UX uses for outputting measuring records are always set up via the INSTALL installation tool.
If other printers are to be used locally, which are connected to computers in your company or company area network, you can control these via the printer setup in the <Parameters> menu.

Prerequisites

The respective printer must first be installed in the operating system.

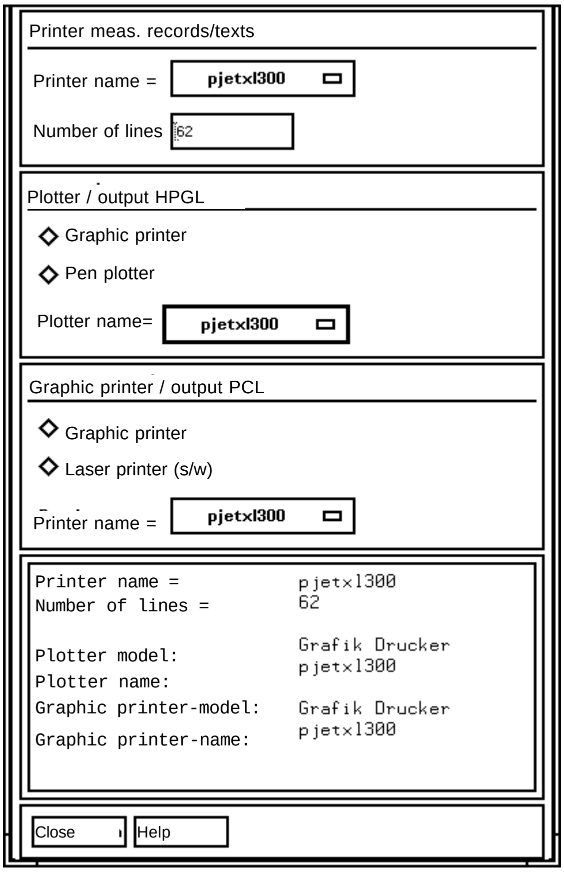


Figure 13-42

Click on the box in the "Printer name" or "Plotter name" field and select a printer from the list.

If the printer is entered in the bottom text window, it can be used for the following tasks.

If you define a printer that is unknown to the system, you will receive an error message.

13.17 System parameter

With this function you define various parameters which the system uses when generating file names, calculating the distance between points and generating measuring points.

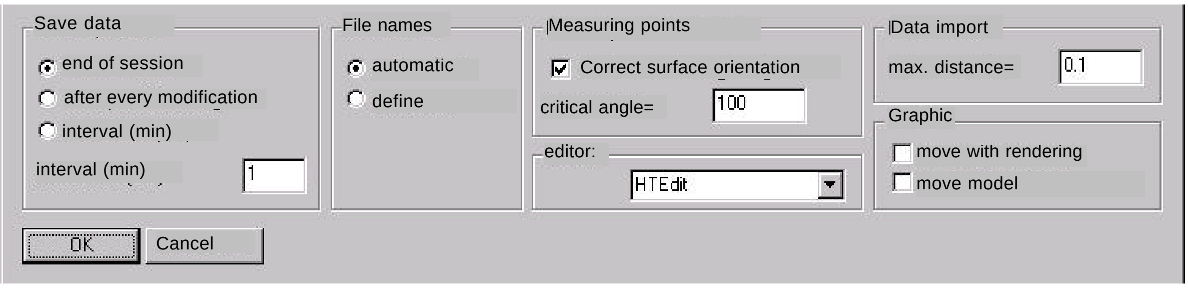


Figure 13-43

Save data

With the <Save data> function you specify when the modified data structure is to be saved. The data structure is modified when, for example, new objects are generated, the orientation of objects is rotated or parameters are changed.

You can save the data as follows:

- on exiting HOLOS ("End of session")
- after every modification
- after a preset time interval in minutes, but only if the data really have been modified.

Automatic file name

If this option is selected, data that are saved on the hard disk are given an automatically generated file name.

When saving nominal values, the measurement strategy utilized for the generation of file names will be taken into account.

Definition of parameters

Depending on the respective measurement strategy, the file name will automatically be prefixed with the following designation:

P_ Points (individual points or raster and grid points, where points outside of FACE boundaries have been removed.)
D_ Grid points for digitization (Digitize surface)
L_ Line points
G_ Grid points
R_ Raster points
C_ Nominal points on parallel curves
S_ Scanning runs

File name defined

You also have the option of freely defining the file name.
If you select the <Define filename> option, the system will ask for a file name each time you save data.

The file name generated by the system is always proposed as the default setting.

Correct surface orientation

During the generation of measuring points, the normal vector of a measuring point will point into the workpiece if the orientation of the respective surface is not correctly defined. This is normally always the case for some surfaces in surface models, since it is not imperative to pay attention to uniform surface orientation during construction in the CAD system.
Since the normal vector of a measuring point is used for defining the probing direction, the surface under a measuring point must always present the correct orientation.
If you have activated the mode for correcting the surface orientation, the system itself can, in most cases, carry out a correction of the respective surface orientation.
You can use these parameters to set the margins and font size for outputting numeric measuring records, if the measuring records are output directly to the printer.

1. Generation of measuring points for individual points and CAD points.

If you define an incorrect surface orientation when generating measuring points (normal vector pointing into the workpiece):

Click on the surface with the mouse.

Activate the <Rotate orientation> function in the <Objects> menu.

The orientation of the surface will then be rotated. All measured points located on this surface are also automatically rotated.

2. Generation of measuring points on a line or a grid.

If you have activated the mode for correcting the surface orientation, the orientation of the normal vectors for measuring points and the associated surfaces is automatically carried out when generating measuring points on a line and a grid.

For this, the angle between the normal vectors of two consecutive measuring points is compared. The angle must not exceed a maximum defined value.

NOTE

The first defined measuring point is always used as a reference for defining the orientation. If the orientation of the surface does not agree with this measuring point, a correct result cannot be obtained.

Automatic correction is not feasible in various cases (for example at exact 90 degree corners). In this case, you have to correct the surface orientation manually.

Definition of parameters

Critical angle

The critical angle value gives the maximum value for the angle between the normal vectors of two consecutive measuring points, up to which no rotation of the orientation must take place.

Max. distance during data import

When importing various data, the points to be imported must have a minimum distance to a surface existing in the system in order to be accepted as part of the saved data.

Enter the desired minimum distance here.

Editor (NT)

HOLOS uses an editor to display various files (records, errors, warnings, etc.).

Define here whether you wish to work with the HOLOS editor, **HTEdit**, or with the Windows editor, Notepad.

Graphic (NT))

With the "move rendered" option, the rendered representation is moved during rotation, translation and zooming without the wire model (quicker image construction).

With the "Translate model" option, you can move the model freely on the screen without a translation vector.

13.18 Parameters for section representations

You can define various parameters for the graphic representation of sections in <Parameters>-<Sections> menu.
After selection of the function a dialog window is opened for definition of these parameters.

Operation:

Select the <Parameters>-<Sections> function.

A dialog window is opened to define the parameters.

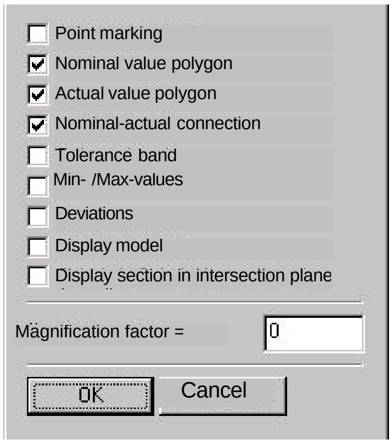


Figure 13-44

If you have not defined any of these parameters, the representation of sections is carried out as the superimposition of polygons of the nominal / actual values. The polygon of the nominal values will be shown in green, the polygon of the actual values in red.

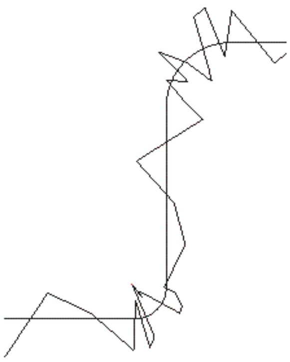


Figure 13-45

Definition of parameters

Point marking

If you have selected the parameter for point marking, the nominal points will be indicated by a yellow circle.

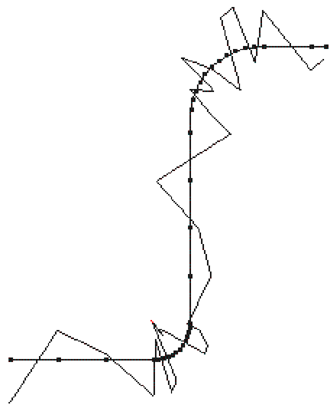


Figure 13-46

Nominal value polygon / Actual value polygon

Activate and deactivate the representation of the polygons.

Nominal/actual link

With this parameter you link together the nominal and actual values of the respective polygons with a red line.

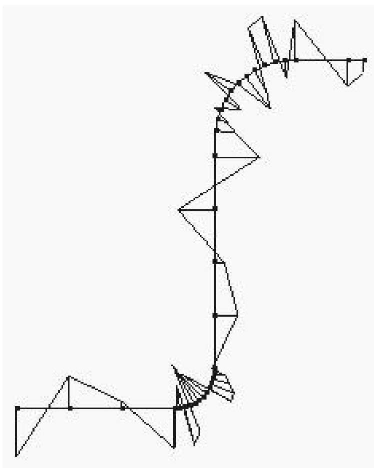


Figure 13-47

Tolerance range

Here you represent the lower and upper tolerances as a tolerance range around the polygon of nominal values. Tolerance ranges are drawn in blue.

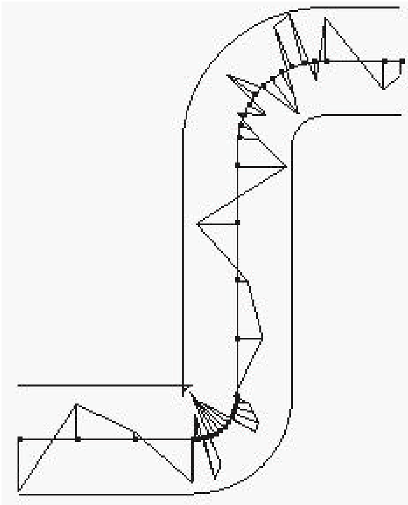


Figure 13-48

Min/Max values

The absolute value of the smallest and largest deviations of the actual values is represented. The extreme values are indicated by means of a white square (black on the plotter).

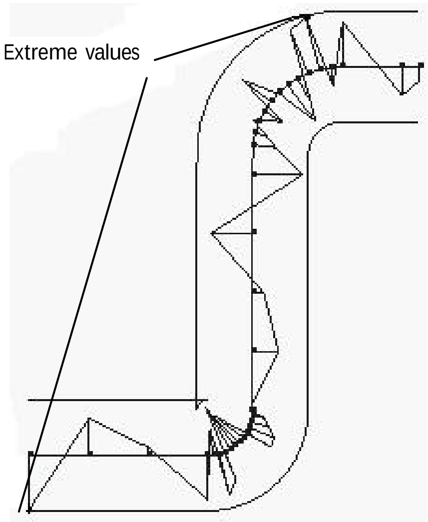


Figure 13-49

Definition of parameters

The representation of the polygon of the actual values as well as of the tolerance ranges can be carried out with a magnification factor, since in the case of very small deviations nominal and actual polygons can hardly be differentiated.

Deviations

With this function you can display the deviations in the corresponding points (see below).

Model representation

Sections are displayed separately, as standard. If you select this option, the sectional representation will be superimposed on the graphic representation of the workpiece.

Display section in intersection plane

The section is displayed in the plane in which it was defined.

Magnification factor

With this function you can define a magnification factor for displaying the polygon of the actual values as well as of the tolerance ranges.

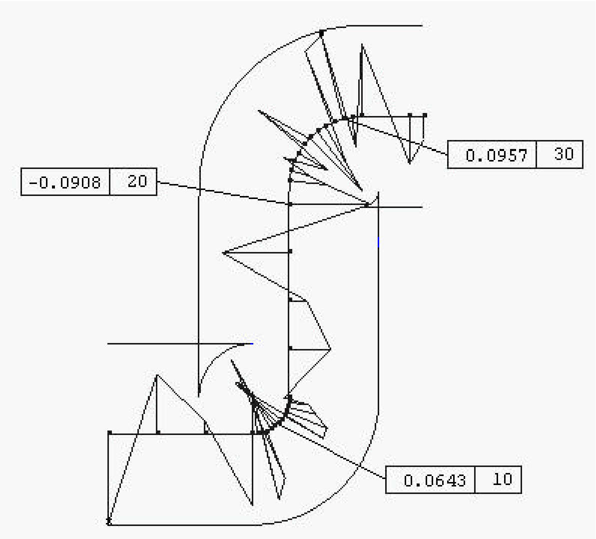


Figure 13-50

13.19 Output

With this function you define the parameters for output of the record.

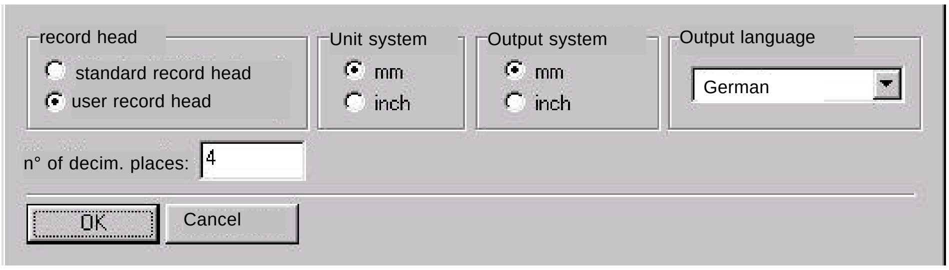


Figure 13-51

Record head

This function enables you to define whether the measuring record output is to be processed with the standard record head or with a user-defined record head.

Unit system

You can select mm or inches as the unit of measurement. This unit determines in which unit of measurement your data model is available.

Output system

For output system you can select measurement units in mm or inches. This unit will then be used for displaying measurement results. If the output measurement system differs from the defined measuring system for your data model, the measurement results are converted.

Output language

With this parameter you define the language in which the measuring results will be output.

Definition of parameters

No. of decimal places

The number of decimal places for the measuring record output and the graphic output of measuring results can be set as desired. The number of decimal places is limited to a maximum of six.

With a value of six decimal places, it is no longer possible to output all of the components in the measuring record; the point number is therefore not displayed.

13.20 Selection of the measuring software

Here you select the measuring software that you wish to work with.

You can also select another measuring software during a HOLOS session or end the communication with a measuring software by selecting "none". In this way, you can alternately access the machine with HOLOS and "Dimension", without ending HOLOS.

📖 If you are working in multiple column mode, please refer to Chapter 1.8.

13.20.1 Serial interface parameters

With the configuration of the serial interface you define the parameters for linking to the UMESS 300 or UMESS 1000 measuring software or CADLINK on PC 2000 or MZ 1050. MZ 1050.

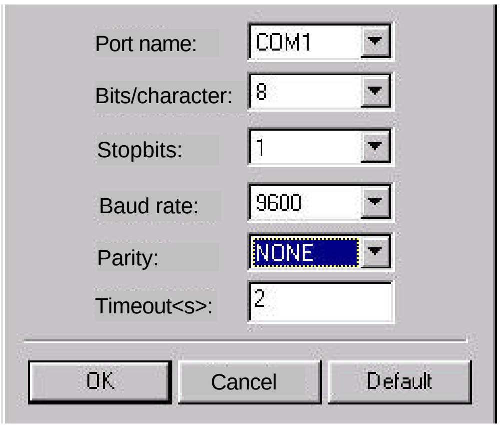


Figure 13-52

<OK>	initializes the serial interface with the set values.
<Close>	cancels the initialization of the interface.
<Default>	invokes a default setting.

Parameter	Meaning	Default
Port name (NT)	Name of the serial port	COM1
Bits/character	Number of bits per character transmitted	8
Stopbits	Number of stop bits per character transmitted	1
Baud rate	Transmission speed in bit/s	19200
Parity(NT)	Parity bit	NONE
Timeout	Various according to DIN 66019 defined response time in sec	2

The settings for the parameters must be identical in HOLOS and CADLINK!

13.20.2 ScanMax parameters (NT)

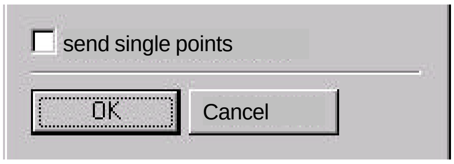


Figure 13-53

If single points are transmitted with ScanMax this option must be activated, so that single point transmission is recognized by HOLOS.

13.20.3 Communication CADLINK (UX)

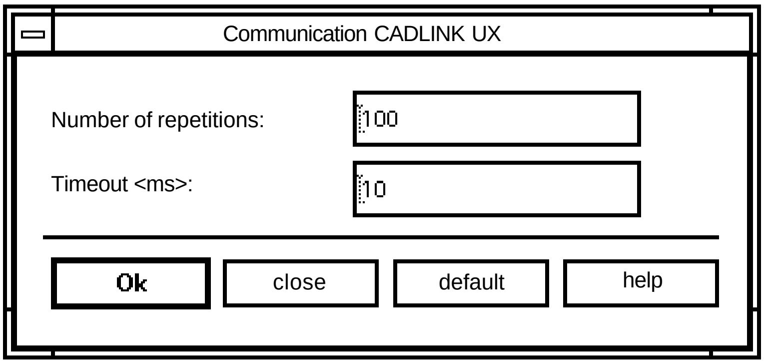


Figure 13-54

Enter the number of selection repetitions and the waiting time in between.

14 Macro programming

This chapter describes programming management for macro runs.

Principles

Via macros which you have programmed, you can automatically perform various actions, e.g.

- Generate measuring runs,
- Perform 3D best fits with measured data,
- Evaluate measured data,
- Output graphic and numeric measuring records etc.

Macros can be started directly via the user interface.

In conjunction with the UMESS-UNIX measuring software, HOLOS macros can be adopted into CNC runs during CNC programming and be started directly from a CNC run.

The <Macro> function in the menu bar branches into the following subfunctions:

- Macro recording
- Macro run
- Display macro
- Delete macro

14.1 Programming macros

During the programming of macros, particular actions which you perform via the user interface are combined in macro runs. However, not all actions are recorded, but only those which are relevant for the processing and documentation of a measuring run. These are, in detail:

- Start measuring runs
- Execute 3D best fit
- Generate evaluations
- Output measuring records to printer or in record file
- Output graphic measuring records to plotter or as graphics dump
- Delete graphic content

You can program new macros or extend existing macros. The option to edit macros (Modification, Correction, Insertion and Deletion of programmed functions) is not included in this version.

Macros are programmed via the functions of the <Macro recording> submenu:

- Program macro
- Program and execute macro
- Extend macro
- Extend and execute macro
- End macro recording

The functions <Program macro> and <Program and execute macro> or <Extend macro> and <Extend and execute macro> are differentiated by the fact that measuring runs which are started during macro recording are not sent to the measuring software, but are only applied to the macro run.

Please note that **before** programming a function, all parameters relevant for its execution must be defined, as they are stored with the function.
The individual parameters concerned are described for the respective function in this chapter.

If you have activated a function for programming macros, the macro function bar appears above the status line. This bar is used for programming different functions. The keys are:

"Program evaluation"
"Program 3D best fit"
"Program graphics dump"
"Program plotter output"

In addition, a message appears, that you have activated the macro programming:
"Macro programming active"

If these two elements are visible on the user interface, the recording of macro functions is activated.
When macro recording is finished, both elements disappear.

NOTE

Various functions can only be executed if macro recording is deactivated (Start macro run, Program macro).

14.1.1 Program new macro

Function: <Program macro> or <Program and execute macro>.

NOTE

If you perform the programming at a workstation which does not have a coordinate measuring device connected to it, always select the <Program macro> function, as measuring runs cannot be sent to the UMESS measuring software.

Click on the <Program macro> function.

A dialog window appears for entering a file name under which your macro will be saved on the hard disk.

Enter the desired macro name in the text field and confirm the entry with <OK>.

If a macro file already exists under the entered name, your attention will be drawn to this.

If you confirm the query with <Yes>, the existing macro file will be overwritten. The content of the old file is consequently deleted.
If you do not wish to overwrite the existing macro file, cancel the action with <No>. You will then be asked to enter a new file name.

When you have entered the file name for the macro to be programmed, you will be asked to enter a comment.

Enter the desired text and confirm with <OK>.

Comments are stored together with the programmed macro data and serve for the documentation of a macro. You can enter text of any length as a comment.

You do not need to record the name of the macro in the comment entry, as it will be automatically inserted, together with the date and time of creation, when the macro file is output.

If you do not wish to enter a comment, confirm the comment entry with <OK> if the comment field is empty.

HOLOS is now ready to record a macro. All actions which are relevant for processing and recording a measuring run are applied to the macro.

14.1.2 Programming measuring runs

Measuring runs are programmed by starting the corresponding measuring run.

Procedure

1. Define a measuring run.
2. Save the measuring run if it is not saved automatically.
3. Start the measuring run by selecting it or by calling up the <Start measuring run> function in the <Define measuring run> submenu.

If you have started recording via the <Program and execute macro> function, the measuring run will be recorded and processed immediately.

14.1.3 Measuring run parameters

For each programmed measuring run the parameters which are defined for the measuring run are applied. Therefore, the parameters **must** be defined **before** starting a measuring run.

The following parameters are stored:

- Output device for measuring records
- Backaway path before probing
- Backaway path after probing
- Correction value for offset correction
- Clearance plane
- Height of clearance plane
- Clearance plane after each point
- Probe
- Upper tolerance
- Lower tolerance

NOTE

Clearance planes for automatic diversion generation cannot be defined via the dialog page for the measuring run parameters during macro programming!

If you define the clearance planes via this dialog page, only a position before and after a measuring run in the respective clearance plane will be preset.

If you wish to work without a clearance plane, you can activate the switch <No clearance plane> in the dialog page for the measuring run parameters.

🕮 see Chap. 13.5

14.1.4 Clearance planes

You can define clearance planes during macro programming. After clicking on the functions for recording a macro, the clearance planes defined for a workpiece will also be displayed on the screen:

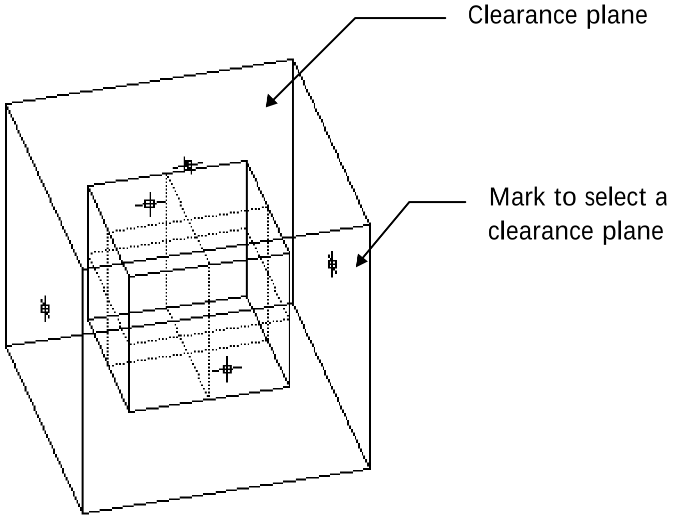


Fig. 14-1

Select a clearance plane by clicking on its marker.

The active clearance plane is shown in green, the other clearance planes are red.

NOTE

If you work with these clearance planes during macro programming, HOLOS will automatically generate the diversion routes from one measuring run to the next via the clearance planes. Therefore, please make sure that the corresponding clearance plane is defined for each measuring run.

If you have not defined a valid clearance plane for automatic diversion route generation (e.g., if you define a clearance plane manually via the dialog page for the measuring run parameters), you will be advised of this with the warning “No valid clearance plane defined”.

The clearance planes themselves are defined via a dialog page in the parameters menu (<Clearance planes> function).

see Chap. 13.5

The following procedure is recommended for programming measuring runs with automatic diversion route generation:

1. Generate measuring run 1
2. Define measuring run parameters
3. Define clearance plane
4. Define main plane
5. Start measuring run 1

6. Generate measuring run 2
7. Define measuring run parameters for measuring run 2
8. Define clearance plane for measuring run 2
9. Start measuring run 1

10. Generate measuring run 3
11. Define measuring run parameters for measuring run 3
12. Define clearance plane for measuring run 3
13. Start measuring run 3

etc....

If the parameters (measuring run parameters, clearance plane) do not change between consecutive measuring runs, the relevant step can be omitted.

Deactivate programmed diversion routes

Diversion routes which HOLOS has generated during macro programming can be deactivated during a macro run:

Before the start of a macro run, call up the <Macro run> function in the <Parameters> menu and then the dialog page for the macro parameters.
Deactivate the execution of the programmed diversion routes.

14.1.5 Program 3D best fit

A programmed 3D best fit can only be performed with data which have been generated during a macro run.
The execution of a best fit via the <Evaluation> menu is therefore **not recorded** during macro programming.

Operation

After calling up a function for macro programming, a function bar appears above the status line.

Select the function <Program 3D best fit>

Data for the best fit

The actual values generated after a programmed measurement are stored in an internal buffer during the macro run.

If you then program a 3D best fit, all previously measured data will be used in the macro run during the 3D best fit to calculate a correction matrix.
In the macro run, the correction matrix is immediately transmitted to the UMESS measuring software and modifies the previously defined coordinate system.

For this purpose it is necessary to program DAW 1713 in the UMESS-CNC run (create workpiece coordinate system from control coordinate system) after the 3D best fit.

However, the new workpiece position system is not saved in UMESS and is only active during the current UMESS session. If you wish to save the new workpiece position system, you must either program this in the CNC run or perform it manually.

> ***NOTE***
>
> Please note in this case too, that before programming the best fit, the parameters for the best fit must be defined.

> ***NOTE***
>
> Another best fit cannot be performed with the same data.

Parameters

The following parameters are stored for a programmed 3D best fit:

- Degrees of freedom for the 3D best fit
  Translation in X
  Translation in X
  Translation in Z
  Rotation around X
  Rotation around Y
  Rotation around Z

14.1.6 Evaluation programming

You can record evaluations during a measuring run in different ways.

Basically, a measuring run is evaluated immediately after calculation of the measuring task. I.e. the deviations are determined, shown on the screen with the preset parameters and, if defined, the measuring record is output to a printer or into a record file.

If you wish to evaluate and record your measuring runs individually, then the evaluation is not directly programmed during macro programming. In this case, define the output device for the measuring record for the respective measuring run:

- No output
- Output to printer
- Output to record file

The measuring record is then output accordingly during evaluation in the measuring run.

Graphic output

For the graphic output of the measuring run, program the graphic output via a macro programming function key (graphics dump or plotter output).

The graphic output should be programmed immediately after programming the measuring run.

The graphics are output in the view that was set on the screen when the graphic output was programmed.

Then program a “Graphics clear” by pressing the corresponding button in the graphic function bar on the right hand edge of the screen.

The content of the graphic representation generated during the evaluation is then deleted, and you have a “clean” screen for the next evaluation.

Via the <Program evaluation> macro function you can program an overall evaluation.

Data for the programmed evaluation

The actual values generated after a programmed measurement are stored in an internal buffer during the macro run. If you now program an evaluation, all previously measured data are evaluated together in the macro run, output in the measuring record and graphically represented.

Call up the <Program evaluation> function.

A dialog window appears in which you define how the measuring record is to be output:

No measuring record output No output of the measuring record during the macro run.

Output to printer Measuring record output to the entered printer during the macro run.

Output to record file The measuring record is stored in a record file during the macro run. In the macro run it is not possible to define the name of a record file yourself. The name is composed of the name of the respective macro and a serial number.

Select the required option and confirm with <OK>.

To output the graphic representation, proceed as described above: Program the respective output function immediately after programming the evaluation. In this case too, the graphic representation is shown in the view which is set at the time of programming.

After a programmed evaluation has been performed, the internal data buffer is emptied. An overall evaluation can therefore only be programmed once.

Parameters

The following parameters are stored for a programmed evaluation:

- Record head parameters
  Order n°
  Customer
  Operator
  Part n°
  Workpiece name
  Comments
- Parameters for the graphic representation
  Camera setting (zoom, rotation etc.)
  Rendering (on/off) **only for HOLOS-UX on HP series 700**
  Representation of deviations
    vectorial
    marking
    graphic icon
    numerical
  Magnification factor
- Lower tolerance
- Upper tolerance

14.1.7 Program graphic output

Graphic measuring records can be generated in HOLOS-UX in two ways:

- Graphics dump (PCL format) or
- Plotter output (HPGL format).

The output of a graphic measuring record is programmed either by clicking on the respective <Graphics dump> or <Plotter> function in the graphic menu bar on the right hand edge of the screen or via the macro function bar with “Program graphics dump“ or “Program plotter output”.

The output is only programmed, not executed, via the macro function bar.
Alternatively, the graphic output for the macro run is recorded and executed immediately.

Program graphics dump

> **_Prerequisite_**
>
> Before programming the graphic output, set the required view (zoom, rotation etc.), as these settings are stored in the macro file together with the output.

Click on the <Program graphics dump> function.

A window opens for defining the parameters for the output (background and image size).

Select the desired values and confirm with <OK>.

If a graphics dump is output during the macro run, normally no frame is generated with the record data.
If you also require this frame to be output during a macro run, you must execute the frame output before programming the graphic output (<Screen> function in the graphic functions).

Program plotter output

Click on the <Program plotter output> function.

A window opens for defining the parameters for the output (background and image size).

Select the desired values and confirm with <OK>.

The frame for the graphic measuring record is automatically generated during the plotter output and does not need to be explicitly displayed.

Parameters

The following parameters are stored for a programmed graphics output:

- Output type (graphics dump/plotter output)
- Display frames (in the case of graphics dump)
- Paper format (A4/A3)
- Inverse representation (for graphics dump)
- Record head parameters
  Order n°
  Customer
  Operator
  Part n°:
  Workpiece name
  Comments
- Parameters for graphic representation
  Camera setting (zoom, rotation etc.)
  Rendering (on/off) **only for HOLOS-UX on HP series 700**
  Representation of deviations
  vectorial
  marking
  graphic symbol
  numerical
  magnification factor
- lower tolerance
- upper tolerance

14.1.8 End macro recording

When all functions for performing a particular measuring run have been programmed or recorded, end macro recording.

Click on the <End macro recording> function in the <Macro recording> menu.

The macro function bar and the message “Macro programming active” in the bottom function bar disappear.

> **NOTE**
>
> In order to start a macro or to program another macro, the macro recording must be ended in any case, as otherwise the relevant function cannot be performed.

14.1.9 Extend macro

An existing macro is extended after macro recording is finished.

Click on the function for extending a macro.

A list of all macros which are defined for the model or for a group appears.

Select the desired macro. After confirming with <OK>, you can start extending the macro.

14.1.10 Parameters for macro programming

When programming macro functions, the parameters which are necessary to execute a particular function are stored together with each function. You **must** therefore define these parameters **before** programming the respective function.

The following parameters are stored with the respective functions:

14.2 Start macros

Select the <Start macro> function in the <Macro run> menu.

A list of all available macros for the model or for a group appears.

Select the macro and start the run with <OK>.

During a macro run, the HOLOS user interface is blocked. You cannot perform any further actions until a macro has been completely processed.

14.3 Program macros in UMESS CNC runs

HOLOS macros can be integrated into CNC runs of the UMESS-UX measuring software. The prerequisite is a UMESS revision from UMESS 7.70.

Procedure

1. Start the PROG mode for UMESS CNC programming.
   see operating instructions for CNC programming in UMESS
2. Program your CNC run as usual.
3. Program a HOLOS macro:
4. Call up the CADLINK option in the PROG mode (DAW 2000).
5. When the CADLINK screen page appears, activate the <Program macro (CNC PROG->CADLINK)> function in the <Macro run> menu in the HOLOS user interface.

A list of all macros which exist for the model or for a group appears.

Select a macro and confirm with <OK>.

The macro run is transmitted to the UMESS CADLINK option and is programmed for the CNC run.

End CADLINK (<Back>) and continue in CNC programming.

If during CNC programming in UMESS you wish to execute the respective macro as well as program it, you must execute the <Start macro> function in the <Macro run> menu (see above).

The corresponding macro is then programmed for the CNC run in UMESS and processed immediately.

When programming a macro in a UMESS CNC run, in addition to the macro name, the name of the corresponding model and the name of any active group are also saved.

Thus you can program macros of different groups of a model as well as macros of different models in a UMESS CNC run.

When you start a HOLOS macro in a CNC run, HOLOS automatically loads the corresponding model as well as any defined group. HOLOS must therefore already be started when you start a UMESS CNC run.

14.4 Macro runs with UMESS 300 / UMESS 1000

The CNC programming of HOLOS macros in UMESS measuring software is only possible with UMESS-UX.

CNC programming of HOLOS macros is not possible with other Carl Zeiss measuring software packages. However, macros can be started and executed via the user interface of HOLOS-UX.

14.5 Display macro run

With this function, you display the content of the programmed macros.

Click on the <Display macro> function in the <Macro> menu.

A list of all macros which exist for the model or for a group appears.

Select a macro and confirm with <OK>.

The steps to be executed for the macro run are displayed.

Example:

Here you will find information on the macro name and date of creation

The comment which you have made is located at this point

Measurement of a measuring run; no measuring record output occurs during evaluation

Output of graphic recod to plotter

Graphics clear

Intermediate position on clearance plane

Evaluation of all previously measured runs:
P_KUBUS_17.mess
P_KUBUS_18.mess
P_KUBUS_19.mess

```
macro: macro4
==================================

Macro created on: 02.05.96 14:25:03
-------------------------------------------------------------------
Example of macro programming

Measure outer edges of a cube
-------------------------------------------------------------------

001: Measurement:
               P_KUBUS_17.mess
               no measuring record output
-------------------------------------------------------------------
002: Record output:
               Plotter A4
-------------------------------------------------------------------
003: Graphics clear
-------------------------------------------------------------------
004: Travel to clearance plane
               SPOS: X= -20.0000, Y= 20.0000, Z= 20.0000
               last clearance plane: YZ-plane in X-MIN
-------------------------------------------------------------------
005: Measurement:
               P_KUBUS_18.mess
               no measuring record output
-------------------------------------------------------------------
006: Record output:
               Plotter A4
-------------------------------------------------------------------
007: Graphics clear
-------------------------------------------------------------------
008: Travel to clearance plane
               SPOS: X= -20.0000, Y= -20.0000, Z= -30.0000
               last clearance plane: XZ-plane in Y-MIN
-------------------------------------------------------------------
009: Measurement:
               P_KUBUS_19.mess
               no measuring record output
-------------------------------------------------------------------
010: Record output:
               Plotter A4
-------------------------------------------------------------------
011: Graphics clear
-------------------------------------------------------------------
012: Evaluation of measured files
               measuring record output
to record file
-------------------------------------------------------------------
```

⇘ In order to print a macro, click on the <Printer output> function at the bottom edge of the window.

⇘ Close the window with <OK>.

If you display a macro whose programming is not yet complete, you will receive the warning:

```
----------------------------------------------------
macro4: Macro recording is still active
----------------------------------------------------
```

14.6 Delete macro run

This function deletes the programmed macros .

Select the <Delete macro> function in the <Macro> menu.

A list appears of all macros which exist for the model or the group.

Select a macro and confirm with <OK>.

A safety query appears, asking whether you really wish to delete the macro.

If you click on <YES> the macro file will be deleted, if you click on <NO> the procedure will be cancelled.

15 Editors

15.1 Macro editor

The macro editor allows you to display and modify recorded macros.

Macros can be programmed under UNIX as before, or also via the macro editor.

15.1.1 Start editor

Select the <Macro editor> function in the "Macro" menu and then the macro which is to be processed, by double clicking or by pressing "Open":

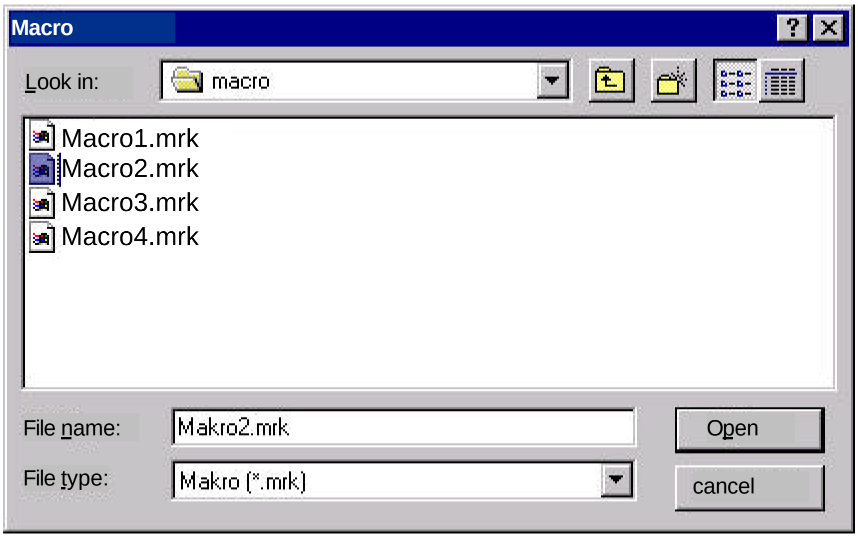


Figure 15-1

In the UNIX version you can also start the macro editor from the task definition window:

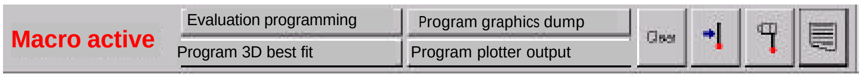


Figure 15-2

Click here to start the macro editor

The macro editor dialog window then opens:

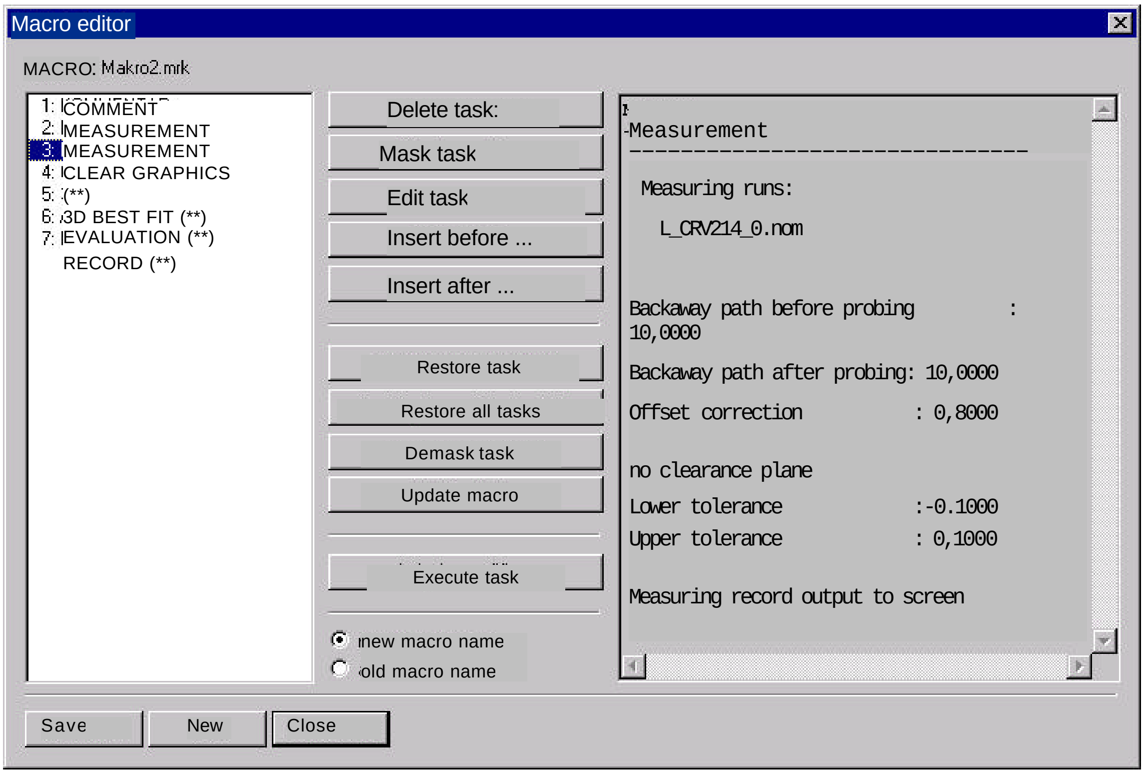


Figure 15-3

The left column contains the tasks contained in the macro.
The center keys execute the functions of the macro editor.
The right column contains the parameters which are defined for a particular task.

15.1.2 Process macros and their tasks

Delete task

Delete one of the macro's tasks.

First of all the task is marked as deleted with (DEL). The task is only actually deleted when the macro is saved or the task "Update macro" is performed.

Mask task

Mask a task.

Masked tasks are marked with (M). They are not performed in the macro run, but can be reactivated at a later point in time.

Edit task

Modify the parameters of a task.

Select the task whose parameters you wish to modify.
To do this, click on <Edit task> or select the task with a double click.

The parameters can be modified for the following tasks:

Comment

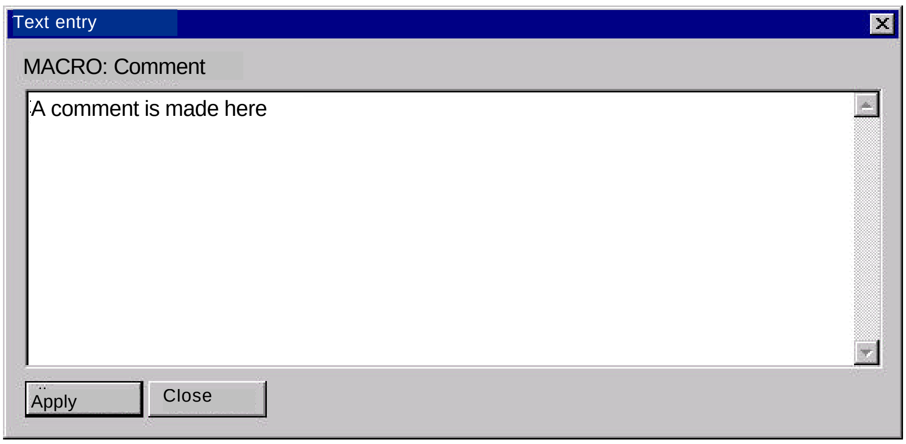


Figure 15-4

Modify the comment entry and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

Measurement

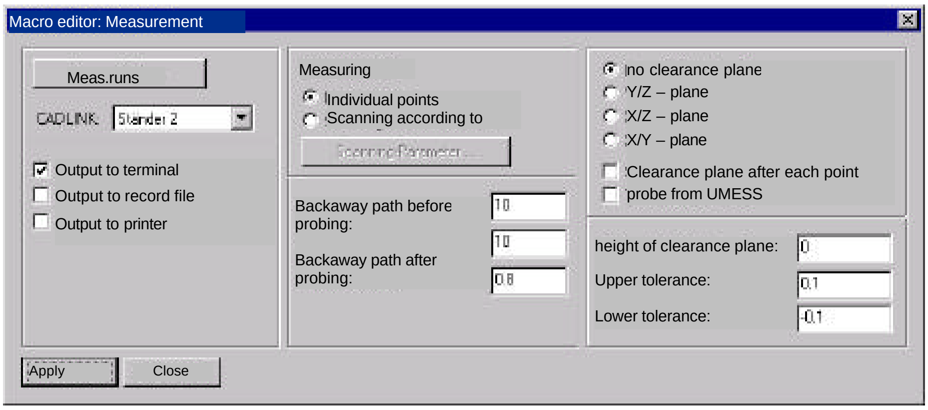


Figure 15-5

Modify the parameters for the measuring run and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

In order to insert new measuring runs into the task, or to remove measuring runs, click on <Measuring runs>.

All existing measuring runs are located on the left, on the right are the measuring runs that are defined for the MEASURE macro task:

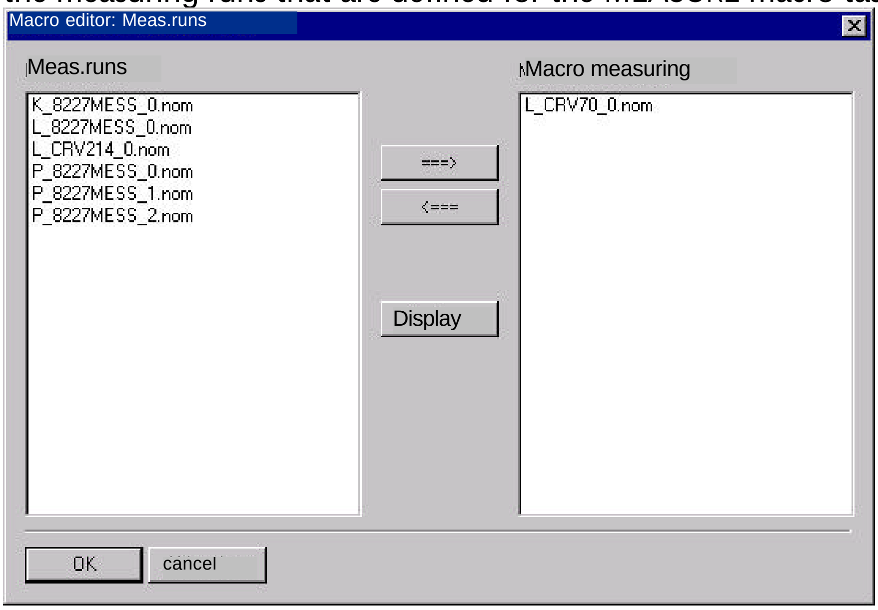


Figure 15-6

Select a measuring run.
= = > applies the measuring run to the macro.
< = = removes the measuring run from the macro.
<Display> displays the measuring run on the screen.

Regular geometry measurement (NT)

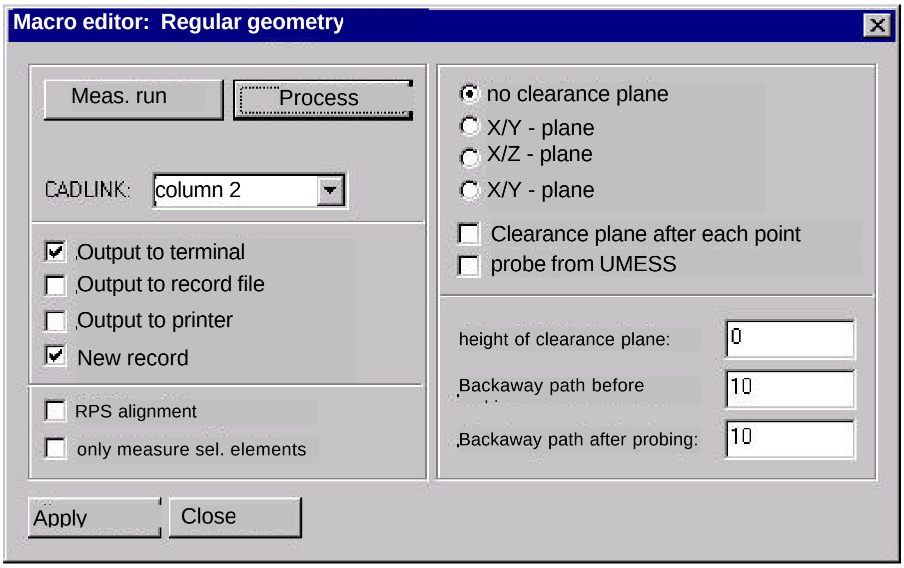


Figure 15-7

Modify the parameters for the measuring run for regular geometries and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

<New record>
A new measuring record is generated with this option. If this option is not selected, the record for the regular geometry measurement is attached to an existing measuring record.

<RPS alignment>
You must activate this option if you wish to perform an RPS alignment after the measurement.

<Only measure selected elements>
This option specifies that you only wish to measure selected regular geometry elements from the list.

With <Process> you can process the selected measuring run. (see "Regular geometry" chapter).

evaluation

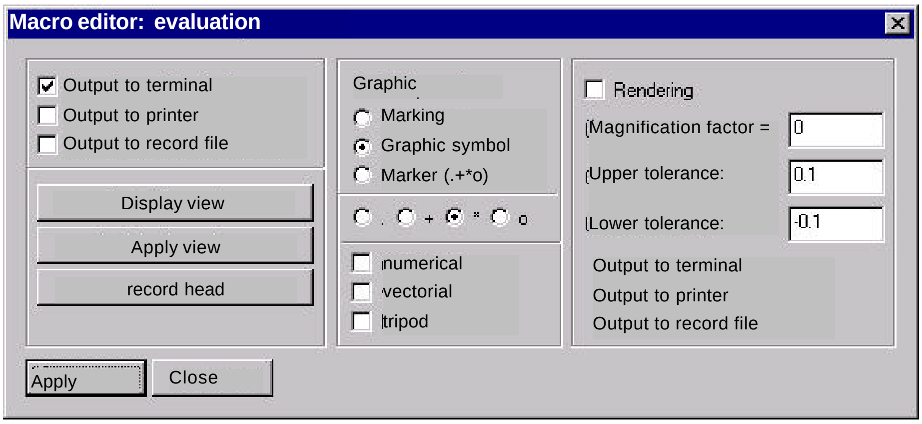


Figure 15-8

Modify the parameters for the evaluation and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

When programming an evaluation in the macro, the graphic representation set at the time will be saved in its position. You can display this view via <Display view>. If you wish to set another view, perform the setting and click on the <Apply view> key. The set view is saved.

3D best fit

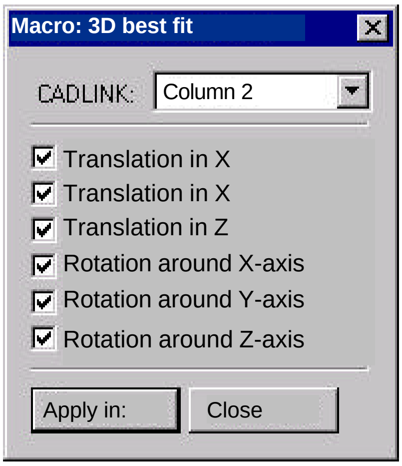


Figure 15-9

Modify the parameters for the best fit and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

Intermediate position 1

(generated by the system)

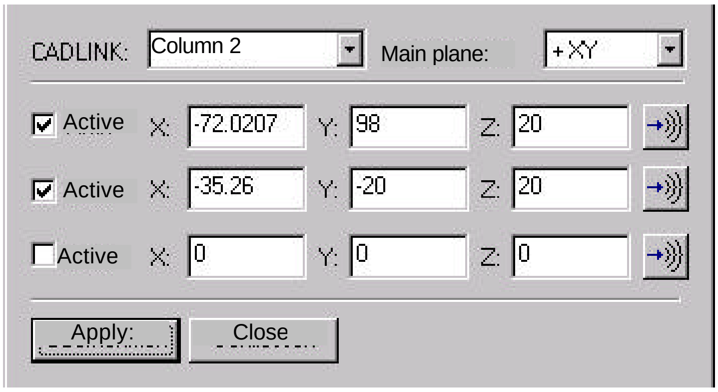


Figure 15-10

Intermediate position 2

(defined by the user)

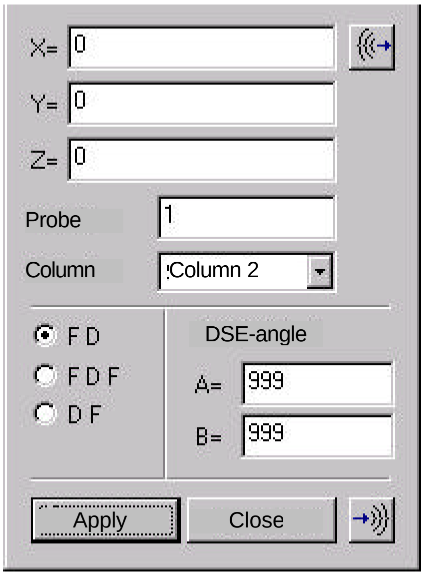


Figure 15-11

Defined intermediate positions are not overwritten by the system. I.e. if an intermediate position 2 exists, the system will no longer generate an intermediate position 1.

Modify the parameters for the DSE position and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

The key with the "radio transmission" graphic allows you to apply an intermediate position directly from your coordinate measuring machine.

DSE position

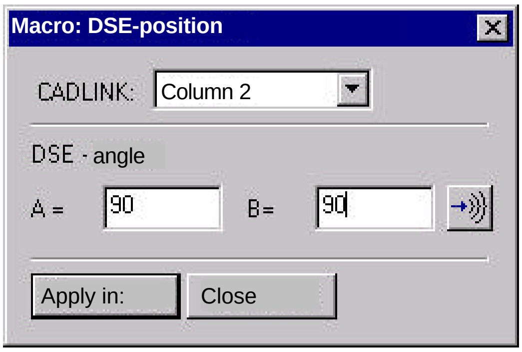


Figure 15-12

This task is only available for coordinate measuring machines with DSE.

Modify the parameters for the DSE position and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

The key with the "radio signal" allows you to apply a DSE position directly from your coordinate measuring machine.

Output (record)

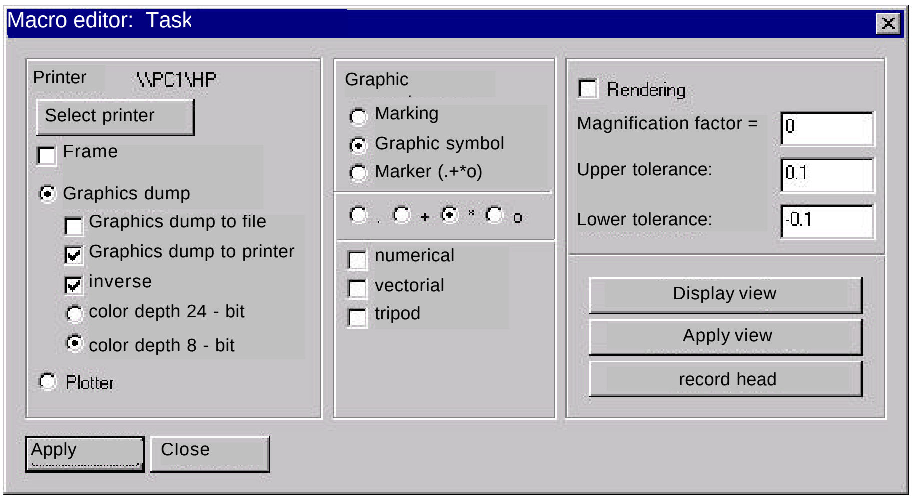


Figure 15-13

Modify the parameters for the record output and confirm the modification with <Apply>.
If you click on <Close> the modification will not be applied.

When programming an output in the macro, the graphic representation set at the time is saved in its position. You can display this view using the <Display view> key. If you wish to set another view, perform the setting and click on <Apply view>. The set view is saved.

Script (NT)

You can use this function to integrate a programmed script into a macro run.

For script programming, see 15.2“.

Insert before/after ...

You can insert new tasks before or after an existing task. To do this, select a task and click on <Insert before> or <Insert after>.

A dialog window appears for selecting a new task:

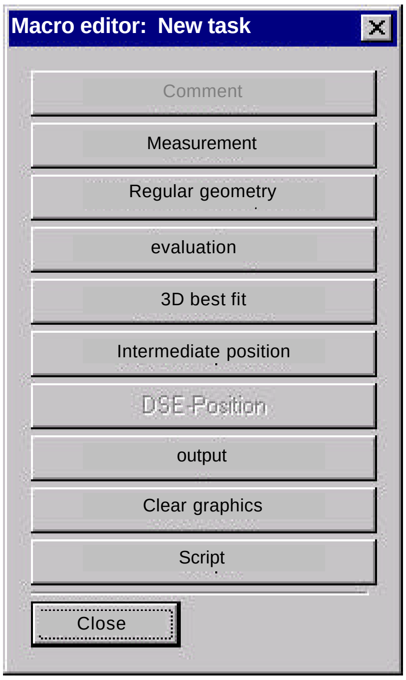


Figure 15-14

Click on the key for the required task.

The dialog window for defining the relevant task appears.

⇘ Define the new task as described for the "Edit task" field.

Restore all tasks

Restore all tasks marked DEL (deleted).

Demask task

Demask a selected masked task (marked M).

The task will be executed again during the next run.

Update macro

Perform an update of all tasks.

All tasks marked as deleted are removed from the list. They cannot subsequently be restored, but must be reprogrammed if they are required again.

Execute task

Execute a selected task from the macro editor immediately.

New macro name

Save a modified macro under a new name.

Old macro name

Save a modified macro under its old name.

Save

Save a modified macro on the hard disk.

New

Create a new macro run.

If a macro has previously been processed and not yet saved, you will be asked whether you wish to save this macro.

Figure 15-15

Close

Close macro editor.

If a macro has previously been processed and not yet saved, you will be asked whether you wish to save this macro.

15.2 Script programming (NT)

You will find script programming in the <Macro> - <Script programming> menu.
It is based on the VBScript language from Microsoft®. A detailed description of this language can be found in the help provided.
You can use script programming to extend the macrofunctionality of HOLOS if you have the necessary programming skills.

An existing script is started via the "Macro" - "Start run" functions and by selecting the "script" file type.
Similarly, you delete a script using the functions "Macro" - "Delete".

15.2.1 The script editor

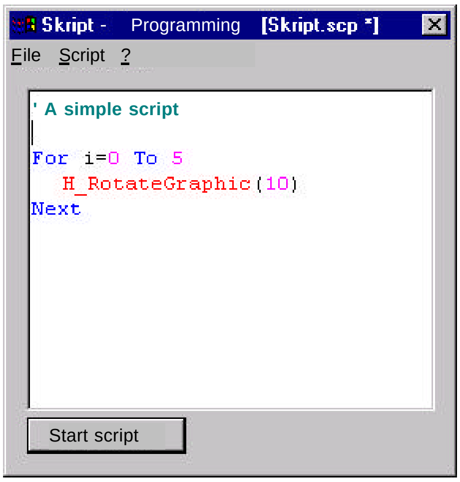


Figure 15-16

The script editor is a simple text editor for generating scripts.

For the sake of better legibility, the VBScript basic functions are displayed in blue, the data in magenta, comments in green and HOLOS commands in red

You can execute the current script via the <Start script> key.

File

The following functions are contained under the file menu item:

- New: create a new script
- Open: open an existing script file
- Save: save the current script file
- Save as: save the current script file under a new name
- Printing
- Close: close the script editor

Edit

Here you will find the standard Windows editing functions, such as "Undo", "Cut", "Copy" etc.

Script

The <Execute> function executes the currently loaded script.

For the <Debug> function, you need the Microsoft Script Debugger. Download under http.//msdn.microsoft.com/scripting/

?-Help

The <VBScript help> function starts the help for the VBScript.

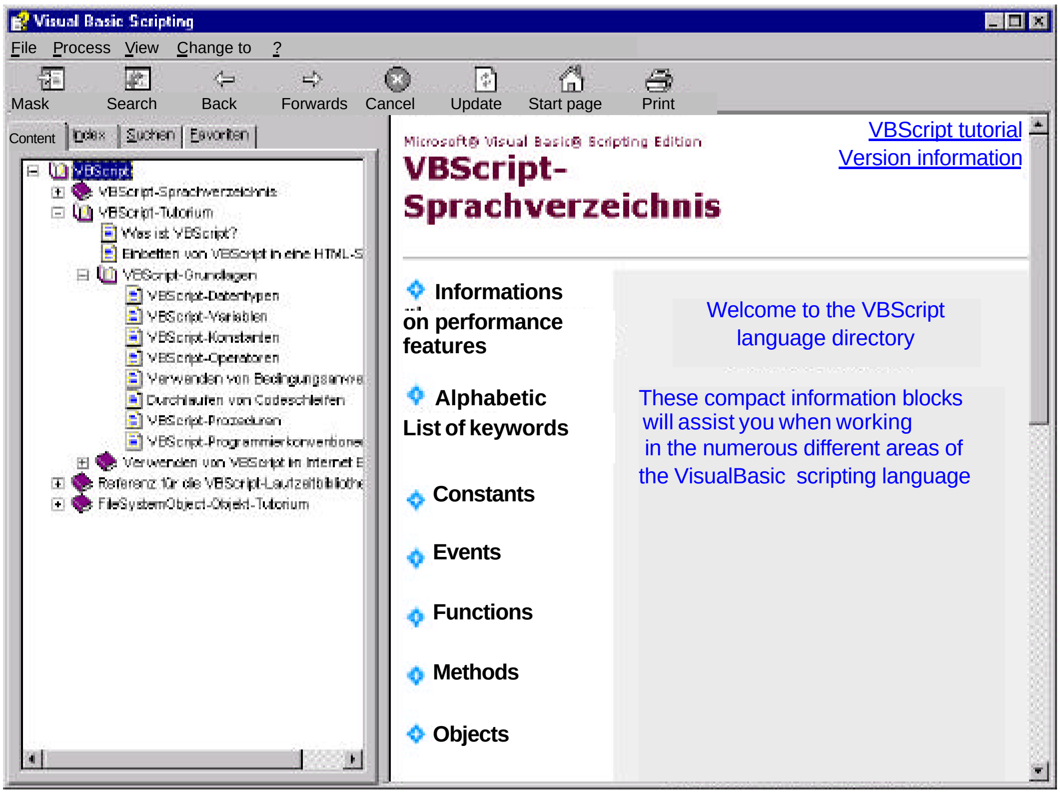


Figure 15-17

15.2.2 Overview of commands

List of all functions which you can use in scripts.

The overview of commands is constructed as follows:

feedback type

Function name(*Parameter type* <parameter name>)
Short description of the function
Description of parameters
Feedback values

If <File> is stated as the parameter name, then a file without path specification and with file name extension is requested as transfer parameter.

The parameter types are given as VBScript data types.

integer **H_LoadModel(***string* **<**model**>)**

Loads the model specified in the string(specify complete path)
Return value:
0:Model has been loaded
1:Error when loading model
2:The model must not be changed while a macro is running

integer **H_CopyModel(***string* **<**path**>,** *string* **<**file**>)**

Copies the current model according to <Path>\<File>
Return value:
0:Model has been copied
1:Error during copying

integer **H_LoadCamera(***string* **<**file**>)**

Load camera values from file
No return value

integer **H_LoadCameraDefault(***int* **<**number**>)**

Load default camera values (numbers: 1-6)
No return value

integer **H_LoadActuals(***string* <file>**)**

Load evaluation
Return value:
0:Evaluation has been loaded
1:Error when loading the evaluation

H_LoadActualChroma(*string* <file>**)**

Load evaluation and display in chromatic coordinates
Return value:
0:Evaluation has been loaded
1:Error when loading the evaluation

H_GraphicDump

Generate a graphics dump with the set parameters
No return value

H_Measure(*string* <file>, *Boolean* <llag>**)**

Start measuring run
Flag:
False:Measuring run on measuring machine
True: Simuluate measuring run
Return value:
0:Measurement OK
1:Measurement was terminated by measuring software

long **H_GetModelInfo(***integer* <type>**)**

Fetch model information (number of elements)
Type:
0:Surfaces
1:Masked surfaces
2:Rational surfaces
3:Patches
4:Masked patches
5:Curves
6:Masked curves
7:Rational curves
8.Faces

9:Masked faces
10:Scanning lines
11:Scanning points
12:Digitization points
13:CAD points
Return value:
Number of elements

H_ResetGraphic

Return graphic to home position
No return value

H_RefreshGraphic

Refresh graphic
No return value

H_ClearGraphic

Clear graphic
No return value

H_MakePreview

Create preview
No return value

integer **H_LoadSection(***string* **<**file**>)**

Load section
Return value:
0:Load OK
1:Section could not be loaded

H_Set3DParameter(boolean <tx>, boolean <ty>, boolean <tz>, boolean <rx>, boolean <ry>, boolean <rz>)

Degrees of freedom for alignment - translational, rotational
tx:Release translation in X
ty:Release translation in Y
tz:Release translation in Z
rx:Release rotation around X
ry:Release rotation around Y

rz:Release rotation around Z
No return value

integer **H_BestFit(***string* **<**file**>))**

3D best fit with nominal values
Return value:
0:all OK
1:Error in best fit

H_SetGraphicParameter1(*boolean* <surf>, *boolean* <patch>, *boolean* <curve>, *boolean* <face**>)**

surf: Display surfaces
patch: Display patches
curve: Display curves
face: Display faces
No return value

H_SetGraphicParameter2(*boolean* **<**scan_lin**>,** *boolean* **<**scan_pkt**>)**

scan_lin: Display scanning lines
scan_pkt: Display scanning points
No return value

H_SetGraphicParameter3(*boolean* <digit>, *boolean* <cad>, *boolean* <wpos**>)**

digit: Display digitization points
cad: Display CAD points
wpos: Display alignment points
No return value

H_SetGraphicParameter4(*boolean* <raster>, *boolean* <splan**>)**

raster: Display raster
splan: Display clearance planes
No return value

H_SetGraphicParameter5(*boolean* <render>**)**

render: Activate/Deactivate rendering
No return value

H_SetGraphicParameter6(*boolean* <o_name>, *boolean* <coord_null>, *boolean* <coord_rot>, *boolean* <isopa**>)**

o_name: Display object name
coord_null: Display coordinate zero point
coord_rot: Display center of rotation
isopa: Display isoparametric lines
No return value

integer **H_SetRasterParameter(***boolean* <YZ>, *boolean* <XZ>, *boolean* <XY>, *boolean* <3D>, *long* <x>, *long* <y>, *long* <z>**)**
YZ:Raster in YZ-view
XZ:Raster in XZ-view
XY:Raster in XY-view
3D:Raster in 3D-view
x:Distance in X
y:Distance in Y
z:Distance in Z
No return value

H_SetDevGraphParameter(*boolean* <num>, *boolean* <vec>, *boolean* <symb>, *boolean* <klecks>, *boolean* *<marker>, boolean* <tri>, *single* <*fa*ct>**)**
Deviation representation:
num:numeric
vec:vector
symb:deviation flag
klecks:marking
marker:marker
tri:tripod
fact:magnification factor
No return value

H_SetDumpParameter(*boolean* <file>, *boolean* <printer>, *boolean* <inverse>, *boolean* <clipboard>**)**
Graphics dump parameters
file: Output to file
printer: Output to standard printer
inverse: Background becomes white
clipboard: Output to buffer
No return value

H_SetChromaParameter(*integer* <depth>**)**

Depth for chromatic evaluation
No return value

integer **H_SetToleranceParameter**(*single* <ut>, *single* <lt**>)**
Tolerance parameters
lt:lower tolerance
ut:upper tolerance
No return value

H_LoadFrame(*boolean* <flag**>)**
Display frame on / off
No return value

H_RunMacro(*string* <file**>)**
Start macro

Warning: If this macro contains scripts, these will not be executed,

integer **H_RegMeas(***string* <name>, integer <select>, integer <align>**)**
Measure regular geometries
select:
0:all
1:only measure selected elements
align:
0:no alignment
1:RPS-alignment (after measurement)
2:3D-alignment (after measurement)
Return value:
0:all OK
>0:Error in alignment
-1:Measurement terminated by CADLINK
-2:New workpiece position not obtained
-3:Error when loading regular geometries

integer **RegMeasManual(***string* <name>*, integer* <select>*, integer* <align>**)**
Measure regular geometries manually
select:0:all
1:only measure selected elements
align:

0:no alignment
1:RPS-alignment (after measurement)
integer 2:3D-alignment (after measurement)
Return value:
0:all OK
>0:Error in alignment
-1:Measurement terminated by CADLINK
-2:New workpiece position not obtained
-3:Error when loading regular geometries

H_RotateGraphic(*single* <degree>**)**
Rotate graphic around the current axis by <degree> angle
No return value

H_StartStop
Activate stop button
No return value

H_EndStop
Deactivate stop button
No return value**H_GetBreak**
Fetch stop button status
Return value:
0:Stop button was not pressed
1:Stop button was pressed

single **H_GetBestFit(***integer* <type>**)**
Fetch result of 3D best fit
Type:
0:Translation X
1:Translation Y
2:Translation Z
3:Rotation around X
4:Rotation around Y
5:Rotation around Z
Return value:Result

H_EndMacro
End macro
No return value

H_WindowSet
Adjust window
No return value

single **String H_GetModelPath**
Fetch model path
Return value:
Model path

String H_GetModelName
Fetch model name
Return value:
Model name

H_SetSecPlane(integer plane)
Set clearance plane
Plane:
0: YZ minus
1: YZ plus
2: XZ minus
3: XZ plus
4: XY minus
5: XY plus

H_SetPlaneOffset(integer plane, single offset)
Set height of clearance plane
Plane:
0: YZ minus
1: YZ plus
2: XZ minus
3: XZ plus
4: XY minus
5: XY plus
Offset:
Value by which the plane is to be moved.
No return value

string **H_GetActualData(integer type, integer no)**
Fetch actual values
Type:
107: Actual values
215: Regular geometry points
104: Patch-ident points
150: Edge points
86: Contour points
72: Corner points
No: Point number

Return value:
String mit X Y Z NX NY NZ D NAME UT OT
X, Y, Z: Nominal points
NX, NY, NZ: Deviation vector
D: Total deviation
NAME: Point designation
LT, UT: lower and upper tolerance
String = TYPE OR NO NOT VALID
Type or number was not valid

string **H_GetNominalData(integer type, integer no)**
Fetch nominal values
Type:
108: Measuring points
No:
Point number
Return value:
String mit X Y Z NX NY NZ NAME
X, Y, Z: Measuring point
NX, NY, NZ: Direction vector
NAME: Point designation
String = TYPE OR NO NOT VALID
Type or number was not valid

integer **H_GetPtCount(integer type)**
Fetch number of points
Type:
107: Actual points
215: Regular geometry points
104: Patch-ident points
150: Edge points
86: Contour points
72: Corner points
108: Measuring points
Return value:
No. of points

integer **H_PatchIdent (double x, double y, double z, double nx, double ny, double nz, integer probe, double probe_radius)**

Patch identification
x,y,z: Probing point
nx, ny, nz: Probing direction
probe: Probe
probe_radius: Probe radius
No return value

H_EdgeMeasure (double x, double y, double z, double nx, double ny, double nz, integer probe, double probe_radius)

Edge measurement
x,y,z: Probing point
nx, ny, nz: Probing direction
probe: Probe
probe_radius: Probe radius

No return value

integer **H_GetRegCount(integer typ)**

Fetch number of regular geometries
Type:
1: Point
3: Plane
4: Circle
5: Slot
6: Rectangular hole
7: Sphere
8: Cylinder
9: Cone
12: Hemisphere
101: Point
102: Straight line
103: distance
104: Angle
105: Circle
106: Plane

Return value:
Number of elements

string **H_GetRegNominalData(integer typ, integer no)**

```
Fetch actual values of regular geometries
Type:
1:   Point
3:   Plane
4:   Circle
5:   Slot
6:   Rectangular hole
7:   Sphere
8:   Cylinder
9:   Cone
12:  Hemisphere
101: Point
102: Straight line
103: distance
104: Angle
105: Circle
106: Plane

Return value:
String
1:   X Y Z U V W NAME
3:   X Y Z U V W NAME
4:   X Y Z U V W R GEO NAME
5:   X1 Y1 Z1 X2 Y2 Z2 U V W B GEO NAME
6:   X Y Z U V W NX NY NZ D1 D2 GEO NAME
7:   X Y Z R GEO NAME
8:   X Y Z U V W R H GEO NAME
9:   X Y Z U V W A R R1 R2 H GEO NAME
12:  X Y Z U V W D R GEO NAME
101: X Y Z NAME
102: X Y Z U V W NAME
103: DX DY DZ D NAME
104: WX WY WZ W NAME
105: X Y Z R NAME
106: X Y Z U V W NAME

String = TYPE OR NO NOT VALID
Type or number was not valid
```

string **H_GetRegActualData(integer typ, integer no)**

Fetch actual values of regular geometries
Type:
1: Point
3: Plane
4: Circle
5: Slot
6: Rectangular hole
7: Sphere
8: Cylinder
9: Cone
12: Hemisphere
101: Point
102: Straight line
103: distance
104: Angle
105: Circle
106: Plane

Return value:
String
1: X Y Z U V W
3: X Y Z U V W
4: X Y Z U V W R
5: X1 Y1 Z1 X2 Y2 Z2 U V W R1 R2
6: X Y Z U V W NX NY NZ D1 D2
7: X Y Z R
8: X Y Z U V W R
9: X Y Z U V W A R
12: X Y Z U V W
101: X Y Z
102: X Y Z U V W
103: DX DY DZ D
104: WX WY WZ W
105: X Y Z R
106: X Y Z U V W

String = TYPE OR NO NOT VALID
Type or number was not valid

string **H_GetRegTolData(integer typ, integer no)**

Fetch actual values of regular geometries
Type:
1:   Point
3:   Plane
4:   Circle
5:   Slot
6:   Rectangular hole
7:   Sphere
8:   Cylinder
9:   Cone
12: Hemisphere
101: Point
102: Straight line
103: distance
104: Angle
105: Circle
106: Plane

Return value:
String
1:   X Y Z
3:   (OT UT)
4:   X Y Z R ISO
5:   X Y Z L B
6:   X Y Z L B
7:   X Y Z R ISO
8:   R (OT UT)
9:   (OT UT)
12: X Y Z
101: X Y Z
102: NO TOLERANCE
103: DX DY DZ D
104: WX WY WZ W
105: X Y Z R
106: NO TOLERANCE

String = TYPE OR NO NOT VALID
Type or number was not valid

string

H_LoadGraphic(string <File>)
Load graphic elements from file
No return value

H_LoadFrameByName(string <File>)
Load frames by names and allocate to the model
No return value

H_LoadTolClass(string <File>)
Load new tolerance class by file
No return value

string H_GetVersion
Query HOLOS version
Return value:
HOLOS Version

H_ClearGraphicElement
Delete graphic elements
No return value

H_DigitPoint(single x, single y, single z, single r, integer probe)
Generate digitization point
X: X-coordinate
Y: Y-coordinate
Z: Z-coordinate
r: Probe radius
probe: Probe number
No return value

16 Window (NT)

This chapter describes the functions of the <Window> menu via which you can change the appearance of the graphics window .

16.1 Divide window.

The <Divide> function allows you to divide the graphics window into several windows, with the projection representations being displayed in three windows and the perspective representation in one window.

see Chap. 3.12

In order to display one of these representations in the full window, click on the relevant window with the left mouse button (double click).
You can define the partitioning of the window as desired by clicking in the center of the frame separators with the left mouse button and moving it to the required position.

This means that it is also possible, for example, only to view two of the windows shown above on the screen.

16.2 Adjustment

The output on the screen is automatically adjusted so that it agrees with the printer output.

16.3 View 3D / YZ / XZ / XY

Change directly into the respective view, without partitioning the window.

16.4 Frames

The <Frame> function allows you to select a frame for the graphic measuring record.

With HOLOS you have the option of generating your own frame for display in the graphic measuring record. You can save the frame under the defined name and use it for a model on request.

(See Chapter 17.4, Drawing functions)

Click on the <Frame> function in the <Window> menu.

A frame selection window appears.

Select a frame and click on <Open>.

The frame is adopted and displayed on the screen.

17 Extras(NT)

This chapter describes the functions of the <Extras> menu.

The main function is subdivided into the following functions:

- Options for setting the working area
- Definition of key combinations
- Definition of colors
- Symbol functions for designing the graphic display
- Graphic icons
    - Move graphic icons
    - Arrange graphic icons
    - Reset
- Save graphic icons
- Create preview
- Release rendering
- Menu editor

17.1 Options for configuring the working area

You can select a number of different options for displaying the working area on the screen.

Activated options are visible on the screen; deactivated components are not visible.

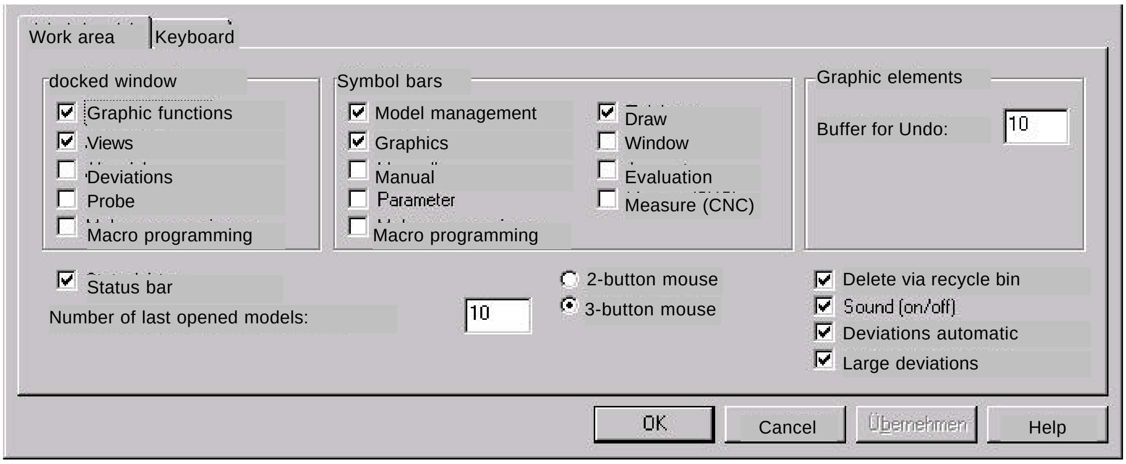


Figure 17-1

All selected display windows and symbol bars can be moved within the working area of HOLOS. The relevant position is saved, so that your working area always retains the same layout each time you use the program.

"Button for undo"	Here you specify how many actions can be undone during drawing
"2-/3-button-mouse"	If the mouse is not automatically correctly recognized, specify here whether you are working with a 2- or 3-button mouse.
"Recycle bin"	With this option, the Windows recycle bin is used for deletion. Without this option, final deletion occurs immediately.
"Sound"	Switch sound on and off.
"Deviations automatic"	Switch display on and off.
"large dev."	complete or reduced deviation display.

17.1.1 Graphic functions

The graphic functions window contains all functions for manipulating the graphic display.

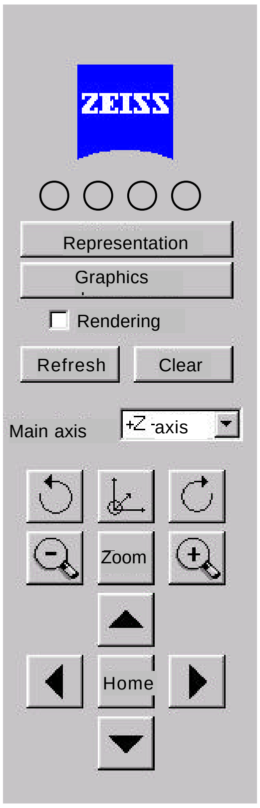


Figure 17-2

For the meaning of the individual functions, please see Chapter 3.

You can arrange the graphic functions window either on the left or the right of the graphics area. Move it as follows:

Click on the window background with the left mouse button.

Whilst keeping the mouse button depressed move the window to the new position. Release the mouse button.

The window is placed in the new position.

17.1.2 Views

The view window contains the functions for saving and calling up defined graphic views.

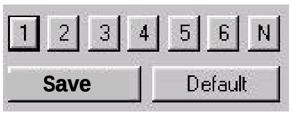


Figure 17-3

📖 For saving and calling up defined views, see Chapter 3.11.

17.1.3 Errors

The results of manual measurements are displayed in the deviation window. This window is activated with the <Deviation> option.

complete display

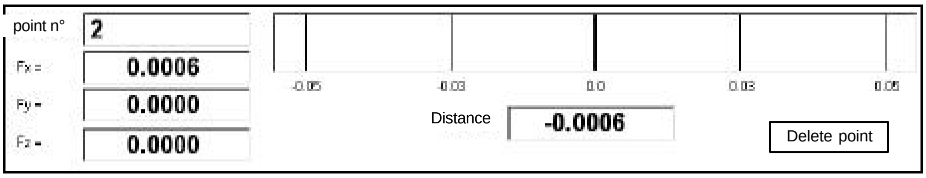


Figure 17-4

reduced display

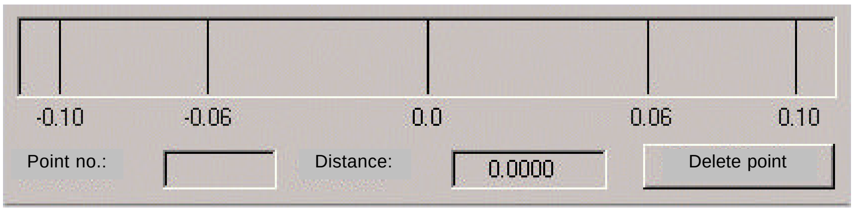


Figure 17-5

During manual measurement the deviation window is opened automatically as soon as a probing point has been adopted by the coordinate measuring device.
In order to close the deviation window after measurement, deactivate the <Deviation window> option via the <Extras> menu.

17.1.4 Probe

The probe window contains the star probe representation for quick definition of a probe.

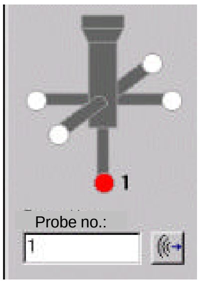


Figure 17-6

17.1.5 Macro programming

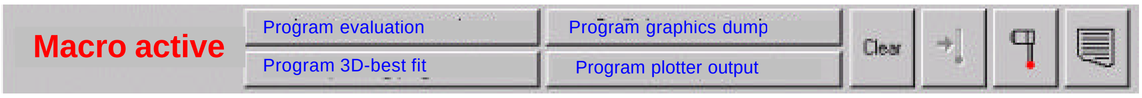


Figure 17-7

The window contains the functions for macro programming:

Output evaluations to the printer or to the record file, graphics dump and plotter output and 3D best fit.

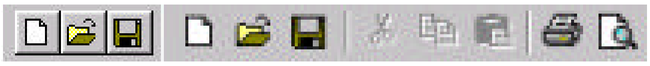
📖 Macro programming, see Chap. 14

17.1.6 Symbol bars

Symbol bars contain graphic keys to enable rapid activation of different functions. The function of each key is displayed when the mouse pointer is held over the key without pressing a mouse button.
You can place symbol bars in any position on the screen. The following symbol bars are available:

Model management

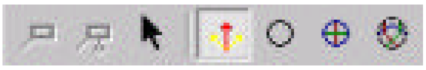
Symbol bar with functions for model management.



- Generate new model
  A new model is generated
- Open model
  An existing model is loaded into the main memory.
- Save model
  The currently loaded model is saved
- Cut, copy and insert model.
- Print model
- Page view

Graphics

Symbol bar with functions for working in the graphics area.



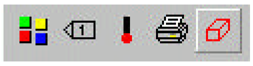
- Move graphic icons
  Click on this key in order to move graphic icons for the representation of deviations on the screen.
  See Chap. 17.5. Graphic icons.
- Arrange graphic icons automatically
  Click on this key in order to arrange graphic icons for the representation of deviations at the edge of the screen.
  See Chap. 17.5. Graphic icons.
- Multiple selection
  Click on this key in order to select several displayed elements simultaneously Drag a window around the elements to be selected in the graphics area. All elements that are located within the window are selected and adopted into a group.
- Four rotations: Normal rotation (specify center of rotation and rotational angle), free rotation, rotation around camera axes, rotation around model axes.

Cadlink

This key is only available if you connect HOLOS to the UMESS-UX measuring software. Via the <CADLINK> key the UMESS option, CADLINK , is requested to transmit intermediate positions during travel. You can thus follow the movement of the coordinate measuring machine on the screen.

Parameter

Symbol bar with keys for the activation of different parameter settings.



- Definition of the colors for the rendered model representation
- Definition of the parameters for graphic icons
- Definition of probe parameters
- Definition of printer parameters
- Definition of colors for clearance planes

Macro programming

Execute, record and end macro.

Drawing

Symbol bar with keys for designing the graphic measuring record and for the definition of graphic frames.

These keys provide functions for generating lines, rectangles, circles and text or for displaying bitmap graphics in the graphics area.

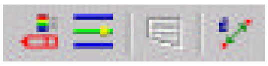
The operation of these functions is dealt with in Chapter 18.

Window

Partition window or change directly into a defined view.

See Chapter 16.3

Evaluation

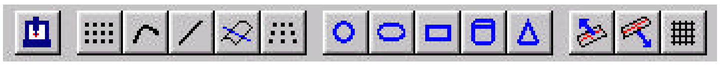


For a description of evaluations, see Chapter 9

- evaluation
- sections
- measuring record
- distance

Measurement (CNC)

Symbol bar with keys for activating functions for generating measuring runs. These functions are only available if a version with CNC functionality is present on your computer. These functions have no relevance for manual coordinate measuring machines.



- Start last measuring run
  Click on this key in order to start the last defined measuring run.

Click on one of the keys shown above in order to define measuring runs:

- Grid
- Curve
- Line with constant point distance
- Plane/workpiece intersection with constant point distance
- Raster with constant point distance
- Circle
- Slot
- Rectangular hole
- Cylinder

- Cone
- Select individual point
- Select edge point
- Net point / net section

Status bar

The status bar at the bottom edge of the screen provides you with constant information about which actions HOLOS is executing, which objects are selected or which selection mode you are in. Where possible, the status bar should therefore not be deactivated.

Number of last opened models

Enter the number of models that are to be displayed as the recently opened models in the <File> menu.

You can load one of these models into the main memory by clicking on the relevant entry with the mouse.

17.2 Key combinations

These functions allow you to define key combinations for rapid activation of the selected functions.

Click on the <Options> function and then on the <Keyboard> panel in the dialog window.

The window for defining key combinations appears:

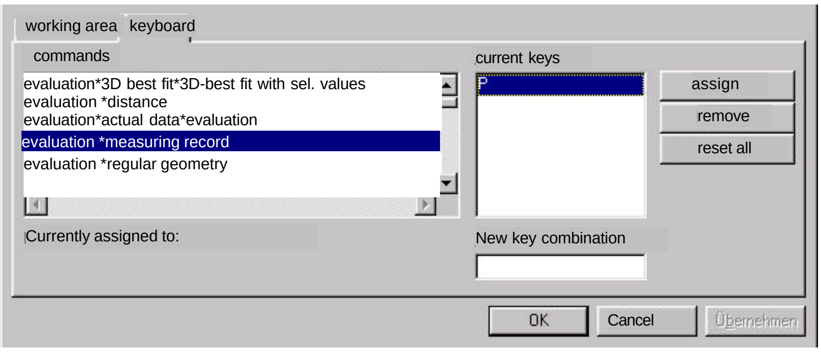


Figure 17-8

Operation

The <Commands> window contains all function entries from the menu bar.

Select a function for which you wish to define a key combination by clicking on it.

Click with the left mouse button in the <New key combination> window in order to activate it.

Then click on the desired key combination (e. g. Ctrl+D).

Click on the <Assign> function in order to assign the key combination to the function.

The assigned key combination is displayed in the <Current keys> window.
If you select a combination that is already configured with another function, you will receive a message.

To remove an assigned key combination, proceed as follows:

Select the function from the list, for which you wish to remove the key combination.

The combination is displayed in the <Current keys> window.

Click with the mouse on the displayed key combination and then on <Remove>.

The key combination is removed.

Example

The key combination Ctrl + D is assigned to the <Rotate orientation> function in the <Objects> menu. You can now rotate the orientation of a surface by simply performing the following actions:

Select a surface.

Press the <Ctrl> and <D> keys simultaneously.

The orientation of the selected surface is rotated.

Reset all

All defined key combinations are removed via this function.

Configuration of key combinations

When HOLOS is delivered and installed, various keys are already pre-configured. The configuration of these keys is given in the appendix.

17.3 Definition of colors

You can freely define the colors for different elements via these functions.

Click on the <Colors> function in the <Extras> menu.

The dialog window for defining the element colors appears on the screen.

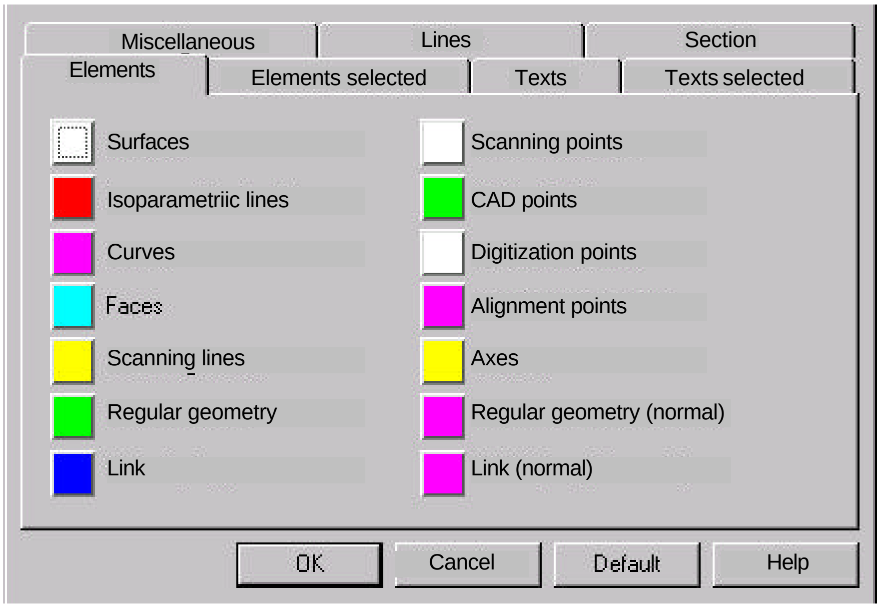


Figure 17-9

Click on the colored key of the element whose color you wish to change.

The dialog window for color definition appears.

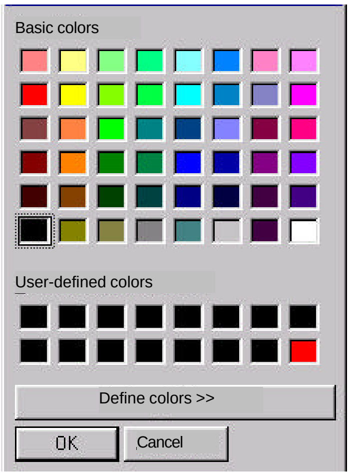


Figure 17-10

Click on the desired color and confirm with <OK>.

Click on the <Define colors> function if you wish to select a color that is not available in the displayed color range.

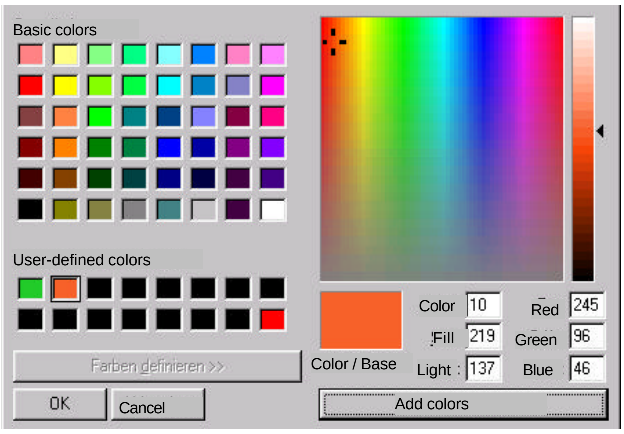


Figure 17-11

Define a new color.

Click on <Add colors> and then on <OK>.

The new color is adopted for the selected element.

Elements

This page contains all geometric elements that can be displayed on the screen. You can define a color for each of these elements.

Elements selected

This page allows you to define the color in which elements will be displayed when they are selected.

Texts

This page allows you to define colors for text that is output on the screen.

Texts selected

Here you can define the color of text that is output for selected elements.

Lines

On this card you define the line types and thickness.

Section

On this card you define the line type and thickness and the color of the sections.

Miscellaneous

On this card you will find other elements for which you can define a color.

Default

The Default function is used to set the preset color values for all elements. All colors defined by you will thus be overwritten with the default values.

17.4 Drawing functions

This provides functions for designing the graphic measuring record, as well as functions for generating the frame.

📖 For further information on these functions, please see Chapter 18.

17.5 Menu editor

You can define a menu that contains frequently used functions via the menu editor.

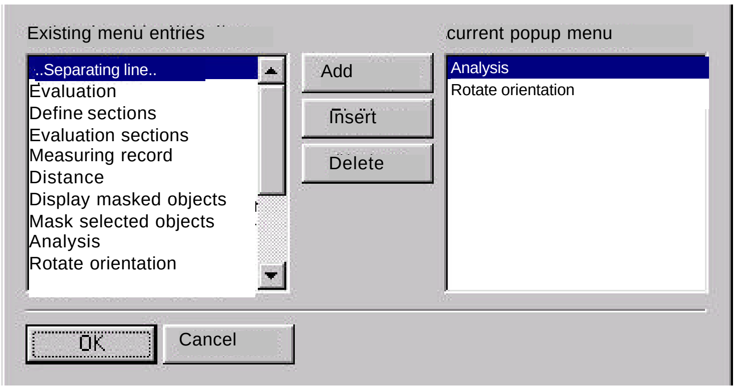


Figure 17-12

📖 The operation of the menu editor is explained in more detail in Chapter 1.9.

17.6 Counter

If data are transferred from CALYPSO or UMESS and the intermediate positions are simultaneously activated in HOLOS (under: manual / point entry / manual / intermediate positions), then the counter is displayed in a large style.

17.7 DSE position (with DSE machines)

With "DSE status" the current angle is automatically entered. If you enter new values into the fields and confirm with OK, these will be applied and the DSE will swivel.

17.8 Graphic icons

These functions allow you to process the positions of the graphic icons for the representation of deviations.

The <Graphic icons> function is subdivided into the following subfunctions:

Move

This function allows you to move graphic symbols.

Start an evaluation of measured values.

The deviations are displayed on the screen.

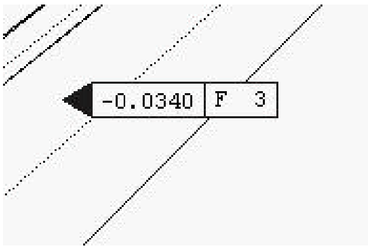


Figure 17-13

Click with the left mouse button on the graphic icon that you wish to move, and drag it to the new position, keeping the mouse button pressed down.

Release the mouse button.

The graphic icon is placed in the new position.

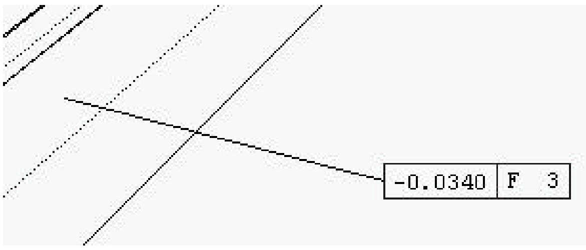


Figure 17-14

Arrange

You can use the <Arrange> function to arrange graphic icons for the representation of deviations at the screen edge.

Start an evaluation of measured values.

The deviations are displayed on the screen.

Start an evaluation of measured values.

The graphic icons are arranged at the screen edge.

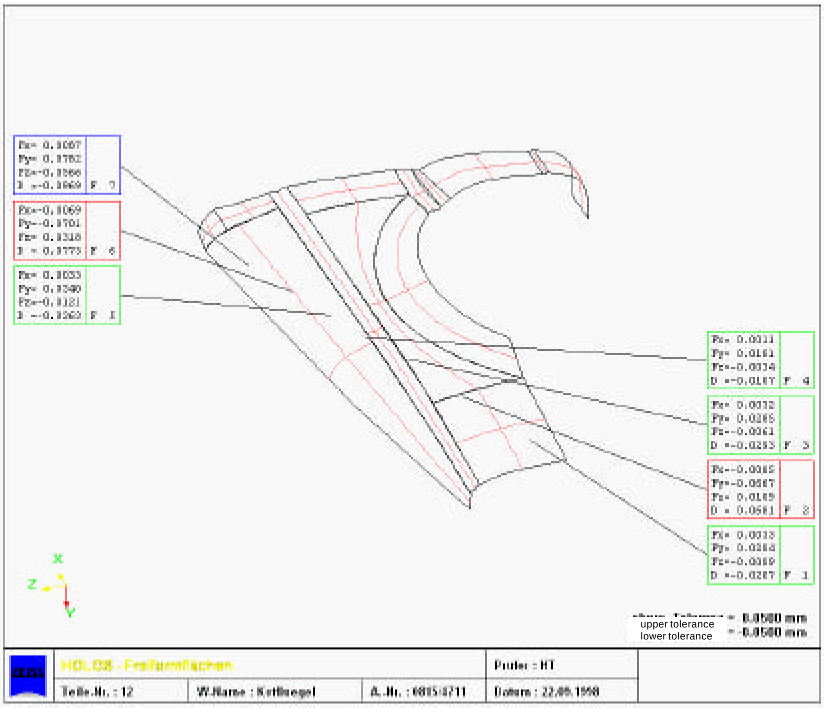


Figure 17-15

Reset

The <Reset> function is used to replace graphic icons that have been moved in their original position.

Save

You can use the <Save> function to save the positions of graphic icons. The positions are assigned to the relevant measuring run.

If a measuring run for which the positions of the graphic icons have been saved is re-evaluated, all graphic icons will be displayed in their respective defined positions, if the <Read position> option is activated in the graphic icon parameter window.

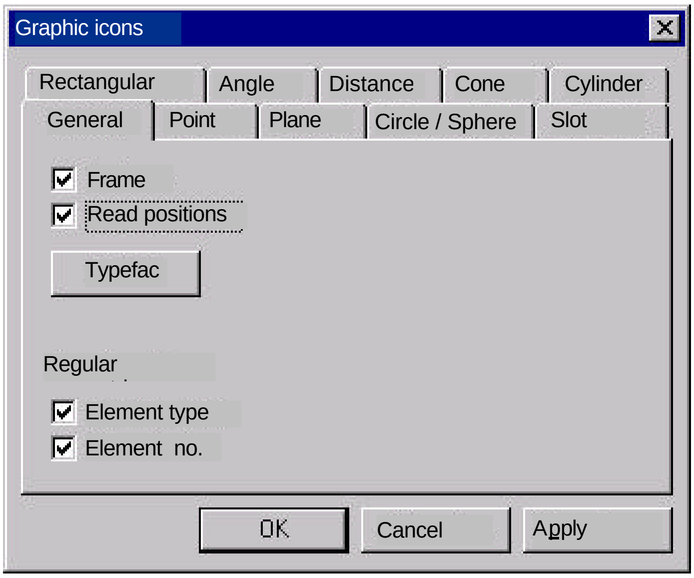


Figure 17-16

17.9 Create preview

You can save a preview of the graphic representation with the <Save preview> function.

A reduced graphic representation of the selected model is then displayed in the model selection catalog. This enables you to quickly identify a particular model.

17.10 Release rendering

A relatively large amount of memory is required in the rendered representation of models, particularly in the case of larger data descriptions.

If you are no longer working in the rendered representation, you can release the memory that has been reserved for representation in this mode via the <Release rendering> function.

17.11 Graphics tool

Here you will find all commands which you can also execute via the graphic icon bar.

17.12 Individual selection

Change into the standard mode for individual selection, e.g. from analysis mode.

17.13 Rotation

normal	An axis must be selected around which rotation then occurs with the right mouse button.
no axis	Free rotation.
camera axes	Three axes are displayed. Rotation always occurs around the axis to which the mouse pointer is closest. It is displayed in yellow.
model axes	The three main axes are displayed. Rotation always occurs around the axis to which the mouse pointer is closest. It is displayed in yellow.

17.14 Hidden line

Only the lines in the foreground are shown, hidden lines are not visible.

Many graphics cards do not support this mode.

17.15 Coordinates

If you move a mouse pointer over the drawing, the coordinates are shown at the top left.

17.16 Surface points

HOLOS calculates a defined number of surface points for the graphic representation of elements. This calculation occurs when a model is first loaded. The points are filtered depending on curvature and saved.

The number of points for the respective elements and the angle for the curvature-dependent filtering are defined in the file .../holos/sys/pt_default.

As a rule, the set values are sufficient for representation of the elements. However, in exceptional cases, which depend on the respective design, it can occur that the graphic representation is too inaccurate. In this case, you can change the number of points for individual elements and recalculate them.

Operation

In the "Extras" menu select the <Surface points> function (NT) or press F3 (UX).

The following input menu is displayed:

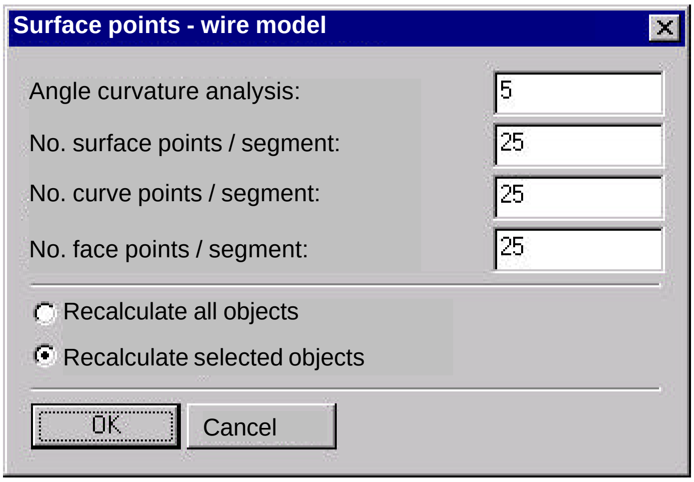


Figure 17-17, Surface points - wire model

If the "rendered representation" option is selected, the dialog window for changing the setting for the rendered representation is displayed:

Dialog window for changing the setting for the rendered representation:

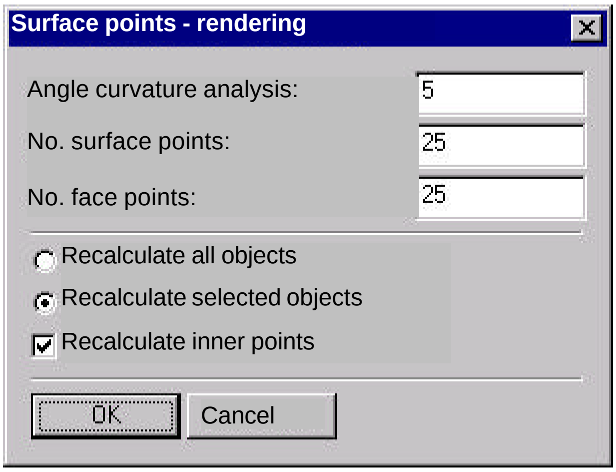


Figure 17-18, Surface points - rendering

Enter the new number of points and an angle for curvature-dependent filtering.

Recalculate all objects

All elements are recalculated.

Recalculate selected objects

Only individual elements which you select are recalculated.

18 Drawing functions (NT)

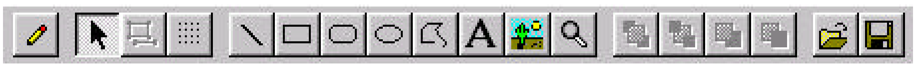
This chapter describes the functions of the <Drawing> menu. These serve to design the graphic measuring record, providing it with text, lines and graphics.

You can use these graphic elements to generate a self-defined frame for the graphic measuring record. You can save a frame so that it is available for other models.

The main <Drawing> function is subdivided into the following functions:

- Elements
  The Elements submenu contains all elements that can be generated.
- Sequence
  Via these functions you can define the sequence of various elements (in the background/foreground).
- Delete
- Mark all
- Raster
- Properties
- Save
- Load
- Load frame
- Save frame

You can also activate all functions in this menu via the <Drawing> symbol bar.



Click on the <Drawing> function in the <Extras> menu.

You are now in drawing mode.

The functions for changing the graphic representation itself are not available in drawing mode. If you wish to change the graphic representation, you must first exit drawing mode.

18.1 Draw elements

The <Draw elements> function allows you to generate graphic elements on the screen.

Rectangle

- Click on the <Rectangle> function.
- Click with the left mouse button on the position at which the first corner of the rectangle is to be located.
- Pull up the rectangle and release the mouse button.

Slot

- Click on the <Slot> function to generate a rectangle with “round corners”.
- Click with the left mouse button on the position at which the first corner of the slot is to be located.
- Pull up the slot and release the mouse button.

Line

- Click on the < Line > function.
- Click with the left mouse button on the position at which the first point of the line is to be located.
- Pull up the line and release the mouse button.

  If you are using a line to point to a defined point on the model, you can connect the start point of the line to a point on the model.
- To do this, press the <Ctrl> key when clicking on the start point, and then click on a point on the workpiece.
- The start point is connected to the workpiece position and moved accordingly if the position of the graphic representation changes.

Ellipse/Circle

Click on the <Ellipse > function.

Click with the left mouse button on the position at which the ellipse is to be located.

Pull up the ellipse and release the mouse button.

To generate a circle, press the <Shift> key and the mouse button simultaneously.

Polygon

Click on the < Polygon > function.

Click with the left mouse button on the positions at which points are to be generated for the polygon.

Text

Click on the <Text> function.

Click with the left mouse button on the position at which the text is to be located.

The text entry window appears.

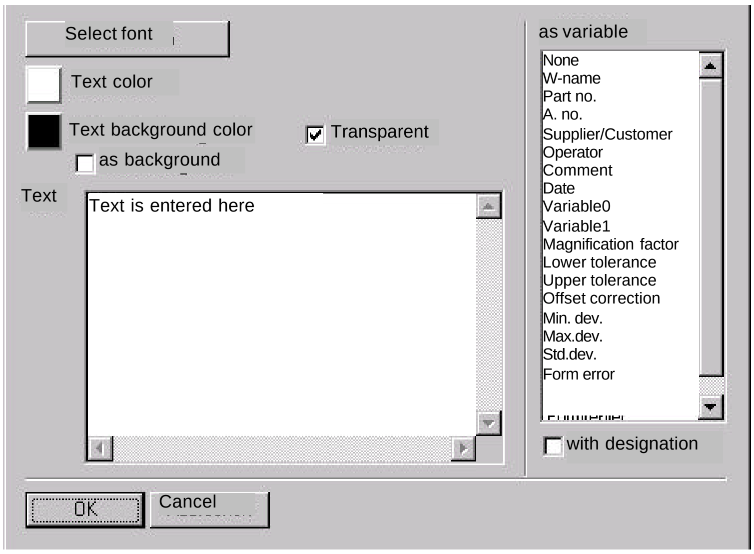


Figure 18-1

Drawing functions

Enter a text and if appropriate define the font type, size, color and background color for the text. - "As background" means the same background as on the screen.

The text is displayed on the screen.

You can define texts as variables. Variables are all texts which you can enter via the parameter page for the standard record head.

Example:

Click on <Date>, if you wish to apply the date from the record head as text. You will then obtain the output: **22.09.98**

Click on the <with designation> option if you also wish to apply the designator for the date as text. You will then obtain the output: **Date: 22.09.98**

Bitmap

Click on the <Bitmap > function in order to output a bitmap graphic on the screen.

Click with the left mouse button on the position at which the graphic is to be located.

The window for selecting a bitmap file appears.

Select a bitmap file and click on <Open>.

The graphic is displayed on the screen.

Detail view

Via the <Detail view> function you can isolate details from the graphic representation and position them at random on the screen.

Click on the <Detail> function in order to capture an area of the graphic representation in a window.

Draw a rectangle around the desired area and release the mouse button.

Click on the <Select> function and move the window with the selected area to a new position.

The selected area is displayed in a separate window on the screen.

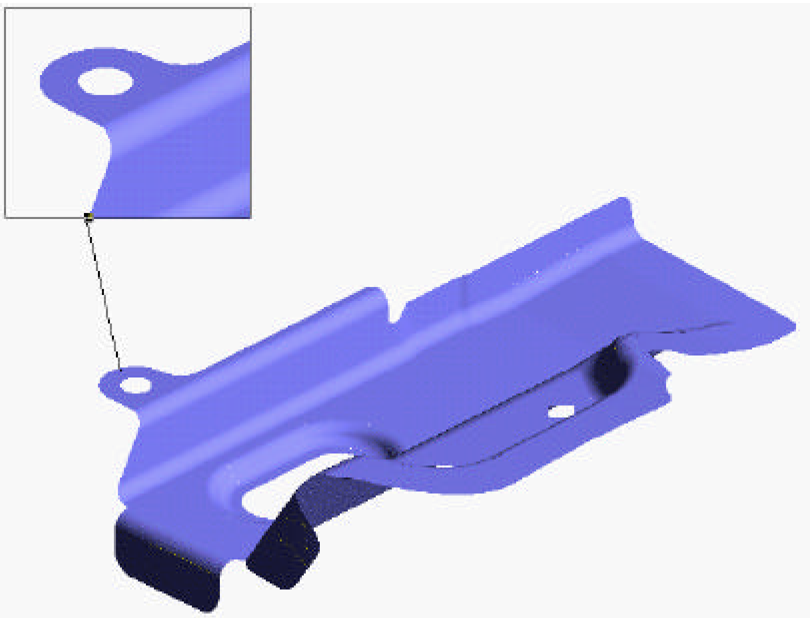


Figure 18-2

Representation of sections

In conjunction with the representation of sections, the function for representation of details has a special significance:
If the rectangle for detail representation is completely drawn over a section, the detail view will contain all properties for representing the selected section. For each window containing a section, the parameters for the representation of the section can be individually defined.
You can display a number of evaluated sections on the screen at the same time.

Evaluate the sections using the <Evaluation> function.

Click on the <Detail> function to capture a section in the graphic representation in a window.

Pull a window over the section.

Click on the <Select> function and move the window with the selected section to a new position.

The selected area is displayed in a separate window on the screen.

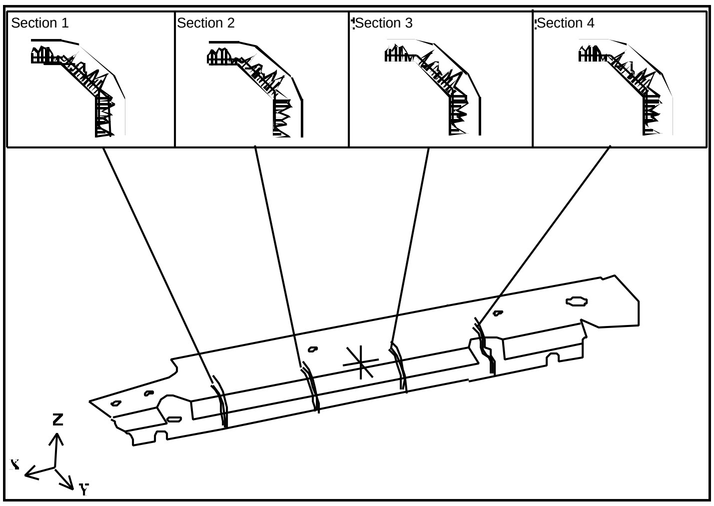


Figure 18-3

The windows that contain a section are marked with the name under which the respective section has been saved.

18.2 Process elements

The generated graphic elements can be processed with different functions.

Sequence

You can define the sequence of graphic elements that are superimposed on each other, by moving the elements into the background or foreground, or transferring elements one plane forward or backward.

Elements whose position is to be defined must be selected first of all.

Align

A number of elements can be selected and aligned to the right, left, above, below, to the same width or height in relation to the last selected element.

Undo

Undoes the last step.

Cut, copy, insert, delete

the selected graphic element.

Mark all

This function marks all graphic elements displayed on the screen.

Raster

A points raster is displayed on the screen. The raster points serve as snap points. All elements which you generate with the raster activated are pulled onto the snap points.

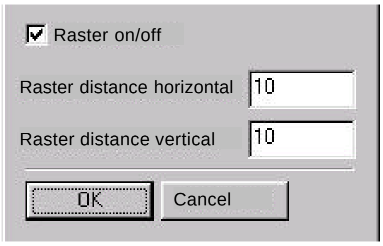


Figure 18-4

This enables you to generate elements with identical points.

Enter the distance for the raster points and confirm with <OK>.

The points raster is displayed on the screen.

Properties

All represented graphic elements have properties such as color or size. You can redefine and change the properties of the elements at will.

Select an element and click on <Properties>

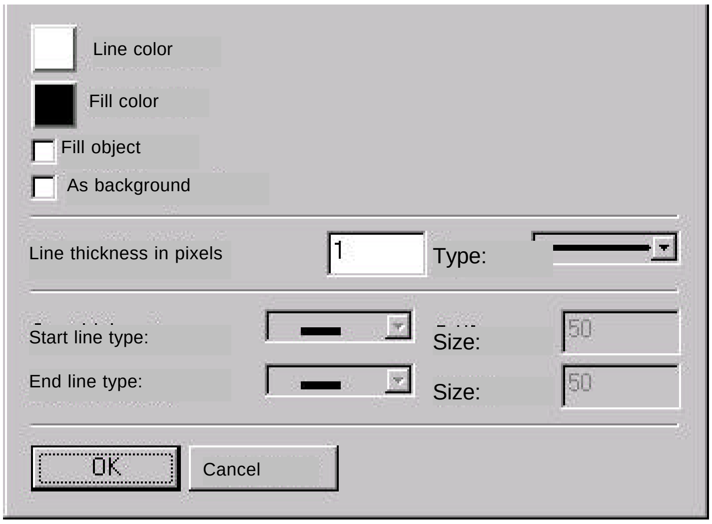


Figure 18-5

Define the new properties and confirm with <OK>.
In order to change the size or position of an element, you must select the relevant element. Then click on one of the capture points and change the size or position.

Save

You can save generated elements in a file. These will then be available again for subsequent HOLOS sessions.

Click on the <Save> function.

The dialog window opens for selecting a directory and entering a file name.

Enter a name and confirm with <Save>.

Load

You can load saved elements from a file.

Click on the <Load> function.

The dialog window for file selection opens.

Select a file and click on <Open>.

The saved graphic elements are displayed on the screen.

18.3 Generate frame

With the functions for generating graphic elements, you can e.g. generate a frame for the graphic measuring record.

A main constituent of the frame is an area for graphic representation. This must not overlay other areas outside its own.

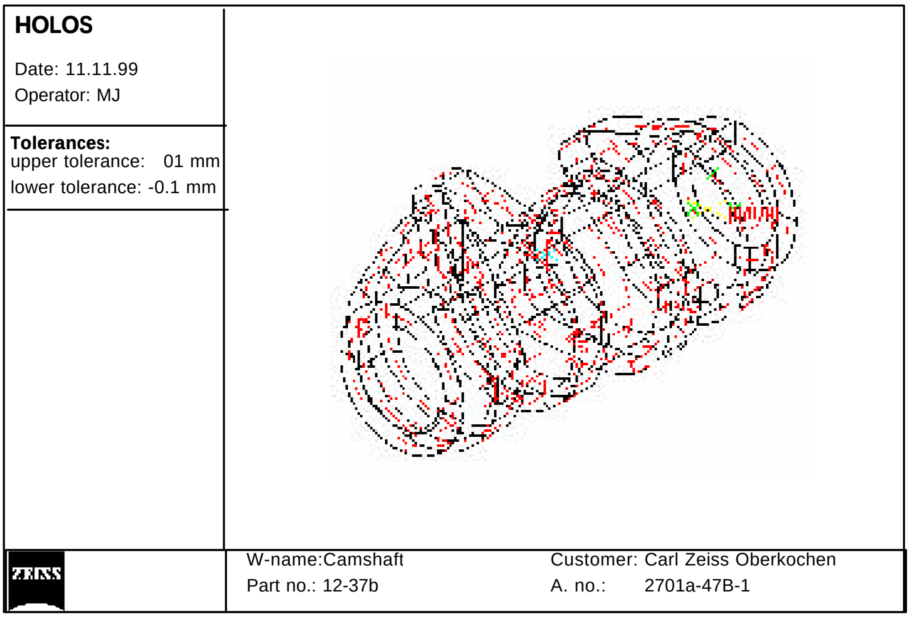


Figure 18-6

Click on the <Rectangle> function to generate a rectangle for the graphic area.

Press the <Ctrl> key and pull up a rectangle. Rectangles for graphic representations are marked in red.

The graphic representation appears in the rectangle.

Now generate the other graphic elements which you require to display the graphic frame.

Click on the <Save frame> function in the <Drawing> menu.

The dialog window for directory selection and entry of the file name opens.

Enter a name and click on <Save>.

The graphic frame is saved.

You can then specify whether the frame is to be used as a standard frame for the current model.

In order to use a saved frame for a new model, select it in the <Window> menu via the <Frame> function.

If you wish to process an already saved graphic frame, read it into the <Drawing> menu via the <Load frame> function. Changes can only be made to the frame elements via this function.

19 Administration (UX)

The “Administration” menu provides various options for designing the operator interface and for data management. The following functions are available:

- Administration level
- Toolbox
- Change model directory
- Save preview
- Start editor
- Menu editor

19.1 Administration level

The administration levels go from zero to four. Zero is standard. You may only activate another administration level if you are asked to do so by your customer care representative from Support.

19.2 Toolbox

Via the toolbox function you can activate various symbol bars beneath the menu bar:

- Model functions
- Editor
- Graphic functions
- Measuring procedure
- Parameters
- Macro
- Views
- Help

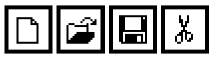
Click on the options for which you wish to activate a symbol bar. Then click on <OK>.

The symbol bars for the selected options are displayed.

You can now call up the respective function by clicking on a symbol.

Model functions

Opens the symbol bar for model functions:



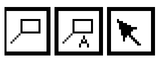
- Generate new model
- Load model
- Save model
- Delete model

Editor

Opens the symbol bar for the HOLOS editor, with which e.g. measuring records are displayed:

Graphic functions

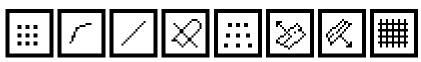
Opens the symbol bar for graphic functions:



- Move graphic symbols
- Automatically arrange graphic symbols
- Select elements by pulling up a window: all visible elements within the window are selected.

Measuring procedure

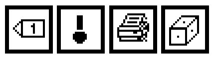
Opens the symbol bar for measuring procedures:



- Measure grid
- Measure curve
- Measure line
- Plane/workpiece intersection
- Measure raster
- Select measuring points
- Select edge points
- Net point/net section

Parameters

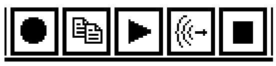
Opens the symbol bar for parameter functions:



- Define parameters for graphic symbols
- Define parameters for probes
- Printer set up
- Define parameters for clearance planes

Macro

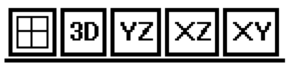
Opens the symbol bar for macro functions:



- Record macro
- Macro editor
- Start macro
- Macro → CADLINK
- Exit macro programming

Views

Change view:



- Four windows
- 3D
- YZ-plane
- XZ-plane
- XY-plane

19.3 Change model directory

This function allows you to manage your data in a project-related manner in different directories.

Click on the <Change model directory> function.

The directory selection is displayed.

Select a directory:

Select a model directory and click on <Ok>.
The standard directory is activated with <Default>.

Set up a new directory:

Enter the name of the new directory in the <Model directory=> input field.
Click on <Ok>.

The model directory is set up.

19.4 Save preview

This function allows you to save a preview of the graphic representation. It is displayed in the model catalog, so that you can access saved models in a more targeted manner.

Click on the <Save preview> function.

A window opens for displaying the preview.

Change the graphic until you obtain the desired representation.
Click on <Save> to save the preview.

The preview is applied and can be displayed the next time the model catalog is opened.

19.5 Editor

The Editor with which you can generate and modify files at the operating system level is started.

19.6 Menu editor

You can use the menu editor to define a menu which contains frequently used functions.

📖 The operation of the menu editor has been explained in more detail in Chapter 1.9.6.

20 Importing and exporting CAD data

This chapter describes the functions for converting CAD data into the internal format of HOLOS-NT and for converting HOLOS models into CAD data.

NT

The functions for importing and exporting CAD data are contained in the <File> menu.

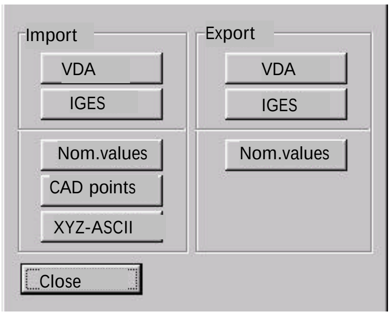


Figure 20-1

UX

Under the <VDA> main function you will find the subfunctions <VDA post processor>, <VDA preprocessor>, and <VDA header>.

Under the <IGES> main function you will find the subfunctions <IGES post processor> and <IGES preprocessor>.

20.1 VDA post processor

The VDA post processor is used to convert data in VDA format into the internal format of HOLOS.

20.1.1 Data conversion

NT

Start the VDA post processor in the “File” menu via <Import/Export> and <Import VDA>.

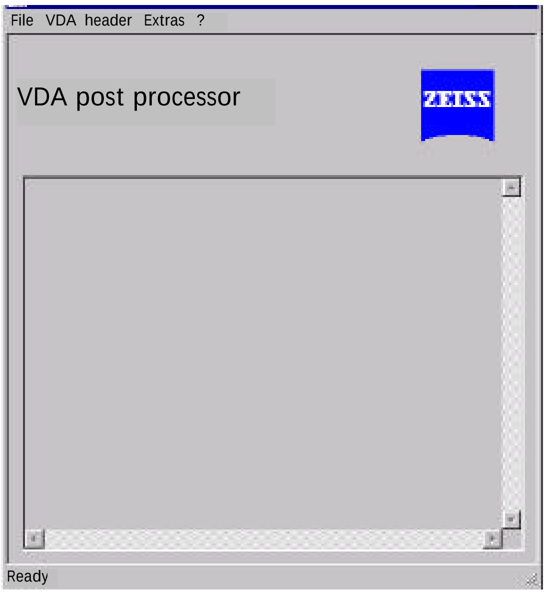


Figure 20-2

Click on the <Open> function in the <File> menu.

A list appears for selecting the VDA files.

Select a file and click on <Open>.
If a HOLOS model already exists under the relevant name, you can define whether you wish to generate a new model or if you wish to overwrite the old model.

Click on <Yes> to overwrite the model.

Click on <No> to generate a new model.

The VDA file is interpreted and saved as a HOLOS model. Information will be provided on the screen about the converted VDA data and the generated elements.

You can obtain information about the converted VDA files via the <Display record> function in the <File> menu.

You can delete the record file via the <Delete record> function in the <File> menu.

VDA header

Information is taken from the VDA header which can be used to preconfigure any HOLOS record variables, if you enter them here:

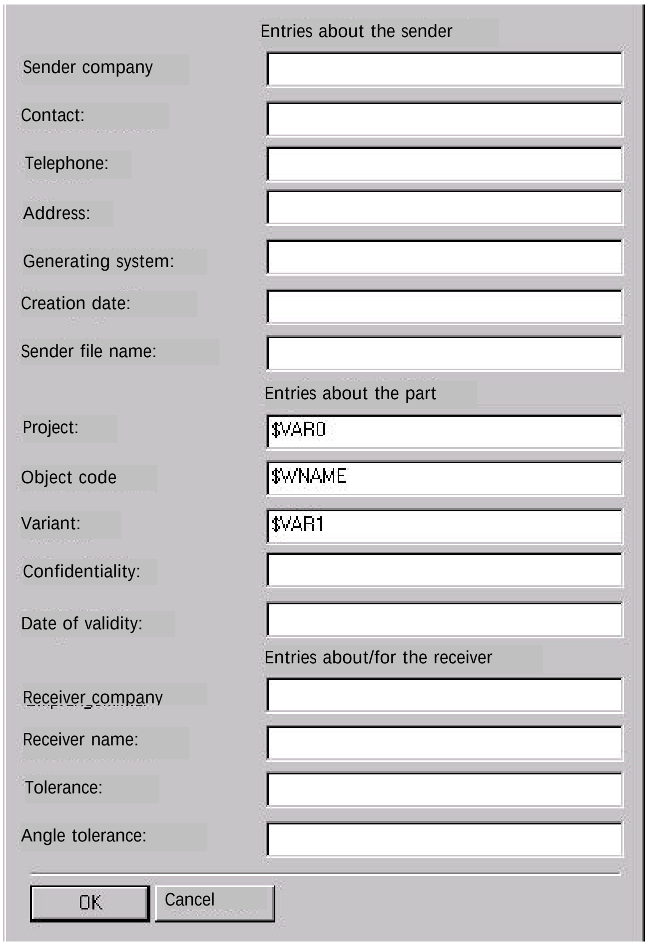


Figure 20-3

Importing and exporting CAD data

Extras

Via the <Extras> menu you can specify the directories for storing HOLOS models, VDA-files and CAD-points.
CAD-points can be found when interpreting VDA- or IGES-files.

The standard directories are:
...\HOLOS-NT\daten\model
...\HOLOS-NT\daten\vda
...\HOLOS-NT\daten\pt_data

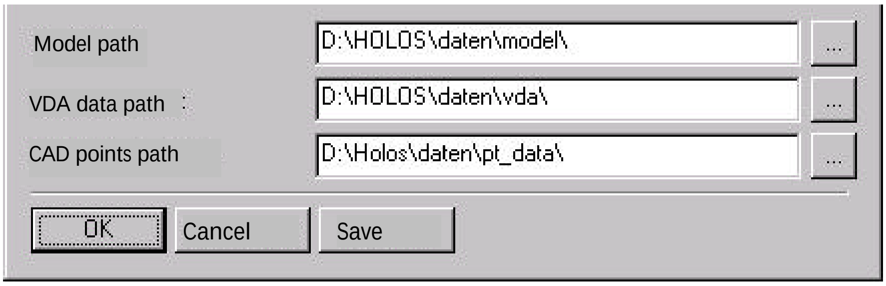


Figure 20-4

You can overwrite the standard directories.

With <Save> the modified directories are permanently saved.

Click on <OK> if the settings are only to apply once and the standard settings are to be used again when the VDA post processor is next started.

UX

Call up the <VDA post processor> function and select the desired directory in the file selection window:

If the path is known

Enter the search path and the file extension *.vda in the “Directory search path” field. The search path default setting is “/users/holos/daten/vda/*.vda”. You should therefore copy all VDA files into this directory.
Update the settings with <Update>. This is necessary if for example you are keeping the selection window open and in the meantime are executing other actions.

If the path is not known

Select the search path in the “Directories” field step-by-step with a double click. Click on the required file in the “Files” field: /.../.../**.** lists the content of the directory in the “Files” field.
/.../.../**..**jumps into the higher-level directory and lists its subdirectories.
Update the settings with <Update>. This is necessary if e.g. you are keeping the selection window open and in the meantime are executing other actions.

If the file name is known

Enter the file name in the “File name” field (you can accept or overwrite the proposed path).

The file lists of the selected directory are displayed.

Select the required file, and start the conversion with <OK>.

The file is converted; an information window displays the status, e.g.:
xxx.vda: 1082 lines read.

During the conversion of VDA files, various information is transferred concerning conversion into an info file:

- the VDA header
- comment lines
- errors that have occurred and information about the number of existing and the number of converted elements.

The info file is located in the respective model directory and can be displayed via the <Edit file> menu.

20.2 VDA preprocessor

The VDA preprocessor is used to convert data in HOLOS format into VDA format for transfer to a CAD system.

20.2.1 Data conversion

Start the VDA preprocessor from the “File” menu via <Import/Export> and <Export VDA> (NT) or with <VDA> - <VDA-preprocessor> (UX).

NT:

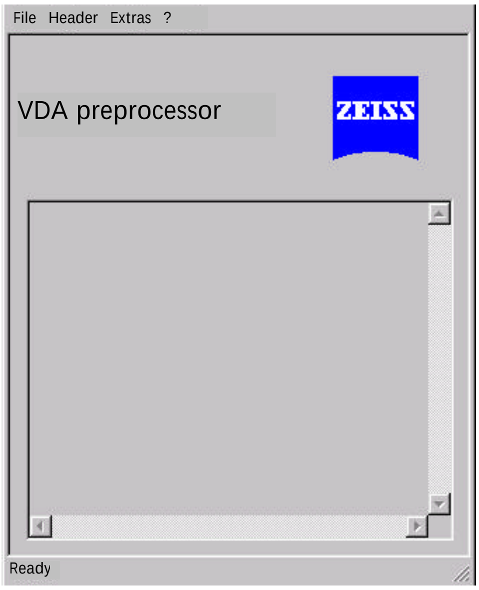


Figure 20-5

UX:
Display window for model selection.

NT: Click on <Open> in the <File> menu in order to select a HOLOS model which you wish to convert into VDA format.

Select the relevant model and click on <Open>.

NT: A dialog window appears in which you specify which elements are to be converted.
UX: Call up the relevant <VDA preprocessor> subfunction in order to define which values are being converted.

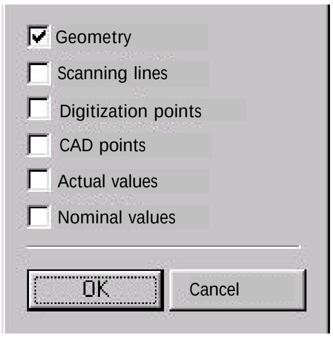


Figure 20-6

Geometry for applying geometrical data to the VDA file. Geometrical data include surfaces, curves and FACEs.

Scanning lines for applying scanning lines to the VDA file.

UX:
Further subfunctions enable the following selection:
<Save lines altogether > → All scanning lines are saved in a VDA file.
<Save lines individually> → Each scanning line is saved in its own VDA file.
<Save selected lines> → Only the selected scanning lines are saved in a VDA file.

Digitization points for applying digitization points to the VDA file.

CAD points (NT) for applying CAD points to the VDA file.

Actual values for applying actual values to the VDA file.

Nominal values for applying nominal values to the VDA file.

With the options for actual and nominal values, a window opens for selecting the relevant model files. Select the files and click on <Open>.

Select the required options and click on <OK>.

You obtain a display of the elements which are being converted.

Importing and exporting CAD data

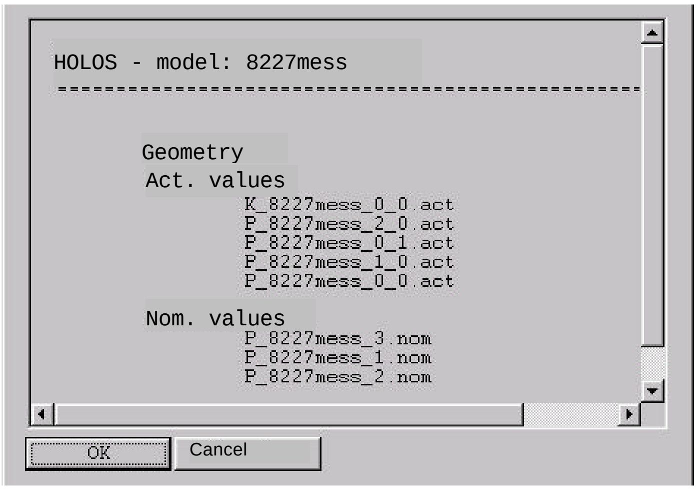


Figure 20-7

Click on <OK> in order to start the conversion.

If a VDA file already exists under the relevant name, you can decide whether you wish to create a new file or overwrite the old one.

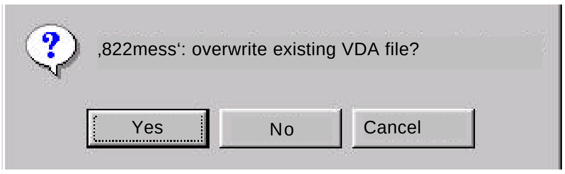


Figure 20-8

Click on <Yes> to overwrite the file, or <No> to create a new file.

The HOLOS model is converted into a VDA file. On the screen you will obtain information about the VDA data or about the converted elements.

20.2.2 Record and info file

Two files are generated during data conversion:

Record file

The record file contains information about converted HOLOS models.

Display and delete NT:
In the “File”menu of the VDA processor.

Display and process UX:
In the “Administration” menu, <Start Editor> function.

Info file

The info file contains information about the last generated VDA file: the header, comment lines, errors that have occurred and information on the number of existing and converted elements.

Display NT:
In the “File” menu of the VDA processor.

Display UX:
In the “Administration” menu, <Start editor> function.

20.2.3 Header

You define entries for the header of the VDA file via the <Header> (NT) / <VDA header> (UX) function.

The header is an element of the VDAfile. According to VDA convention, it must not contain any special symbols (before ASCII 32 and after ASCII 126).
The header must only contain upper-case letters in accordance with VDAVS 2.0. However, you can also enter lower-case letters – the program will automatically convert them when they are entered into the VDA file.

Select the <Header> (NT) / <VDA header> (UX) function.

The input window is displayed:

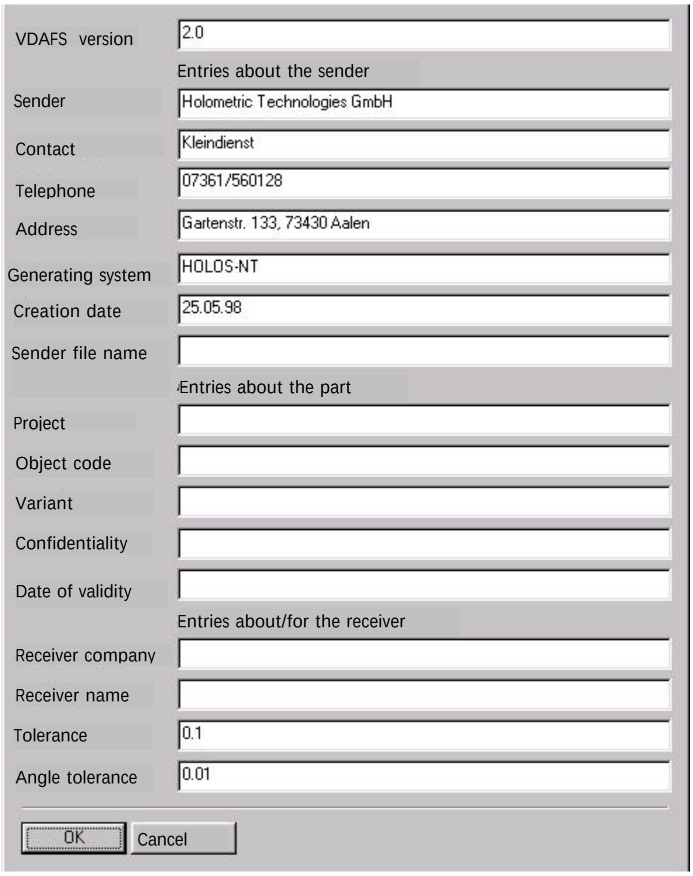


Figure 20-9

Fill in the required input fields and click on <OK> to apply your entries.

20.2.4 Extras (NT)

Via the <Extras> menu, you can specify the directories for storing VDA files or HOLOS models.

The standard directories are:
...\HOLOS-NT\daten\vda
...\HOLOS-NT\daten\model

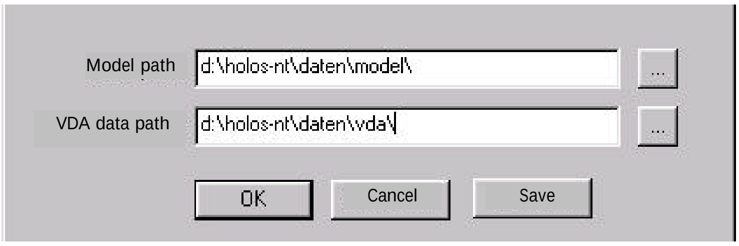


Figure 20-10

Enter new directories for storing your data.

Click on <Save>, if you wish to save the settings.
Click on <OK>, if the setting only applies once. When the VDA preprocessor is next started, the standard settings will then reapply.

20.3 VDA analyzer (NT)

The VDA analyzer allows you to analyze VDA files.

You will receive information on the screen about which VDA elements are contained in the VDA file.
A HOLOS model is not generated.

The analyzer is operated like the VDA post processor.

20.4 Check VDA file(UX)

With the <Check VDA file> function, you check VDA files for conformity with the interface format VDAFS 2.0. However, in the existing progam version, only the line length of 80 characters is checked. The result is given in the status line.

A dialog window opens for selecting the VDA file.

20.5 IGES post processor

The IGES post processor is used to convert data in IGES format into the internal format of HOLOS-NT.

Start the IGES post processor via <Import>-< IGES > (NT) or <IGES>-<IGES post processor> (UX)

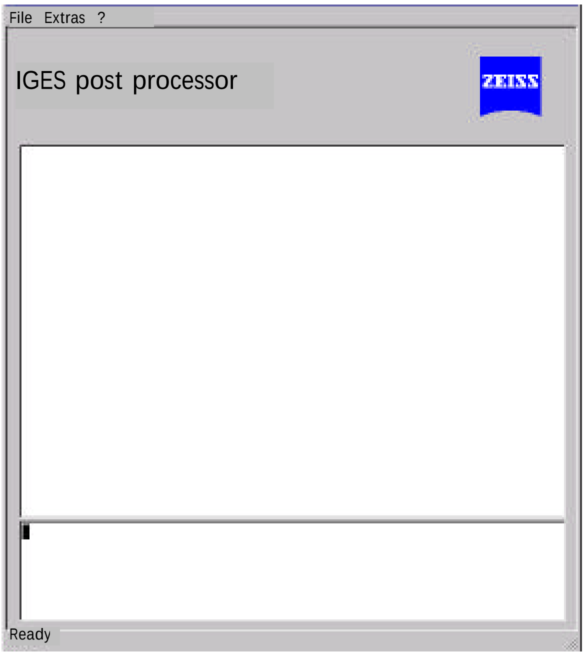


Figure 20-11

The bottom section serves for error display during data conversion.

NT: Click on the <Open> function in the <File> menu.

A list appears for selecting the IGES files.

Select a file and click on <Open>.

If a HOLOS model already exists under the relevant name, you can choose whether you wish to create a new model or overwrite the old one.

Click on <Yes> to overwrite the model, or <No> to generate a new model.

The names for new models are generated from the names of the corresponding IGES files.

Example: existing in the system HOLOS.igs
generated model HOLOS_IGS

if model exists HOLOS_IGS0
HOLOS_IGS1 .....

The IGES file is interpreted and saved as a HOLOS model. You will receive information on the screen about the converted IGES data and the generated elements.

You can obtain information about the converted IGES files via the <Display record> function in the <File> menu.

You can delete the record file via the <Delete record> function in the <File> menu.

Extras (NT)

Via the <Extras> menu you can specify the directories for storing HOLOS models, IGES-files and CAD-points.
CAD-points can be found when interpreting VDA- or IGES-files.

The standard directories are:
...\HOLOS-NT\daten\model
...\HOLOS-NT\daten\iges
...\HOLOS-NT\daten\pta_data

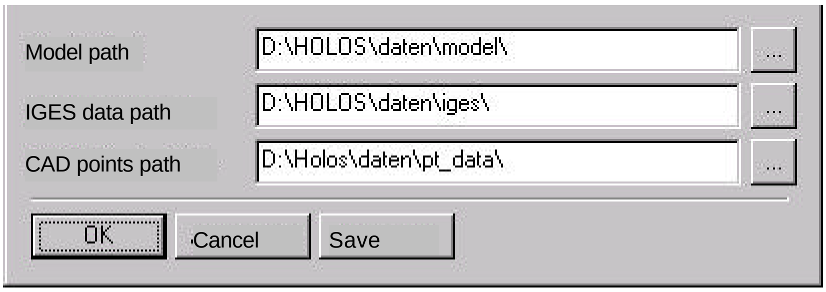


Figure 20-12

You can overwrite the directory entries in the text fields.

⇘ Click on <Save>, if you wish to permanently save the settings. With <OK> the settings will only apply once, and when the IGES post processor is next started, the standard settings will apply again.

Also under "Extras", you will find the command "additional output". With this command additional information and error messages are displayed during transformation.
Please note that this will slow the transformation down.

20.6 Configuration (UX)

You can restrict the interpretation of the entities contained in an IGESfile using options in the configuration.
This is helpful e.g. if you only wish to interpret curves or points and mask all other elements.

Operation

Select the <IGES>-<CONFIGURATION> function.

A window opens for selecting the options.

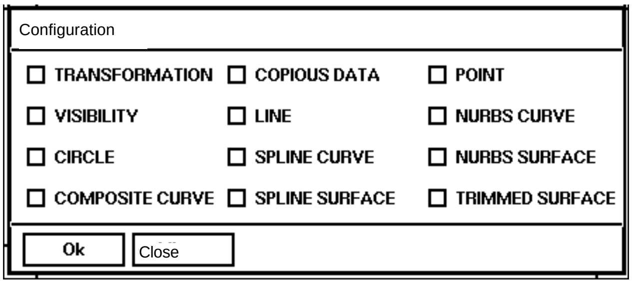


Figure 20-13

Option	Presetting	Meaning
TRANSFORMATION	on	TRANSFORMATION MATRIX is interpreted
VISIBILITY	on	BLANK STATUS is evaluated
CIRCLE	on	CIRCLE(100) is interpreted
COMPOSITE CURVE	on	COMPOSITE CURVE(102) is interpreted
COPIOUS DATA	on	COPIOUS DATA (106) is interpreted
LINE	on	LINE (110) is interpreted
SPLINE CURVE	on	SPLINE CURVE (112) is interpreted
SPLINE SURFACE	on	SPLINE SURFACE (114) is interpreted
POINT	on	POINT (116) is interpreted
NURBS CURVE	on	NURBS CURVE (126) is interpreted
NURBS SURFACE	on	NURBS SURFACE (128) is interpreted
TRIMMED SURFACE	on	TRIMMED SURFACE (144) is interpreted

If you switch the **TRIMMED SURFACE** option on, then the following options will automatically be switched on:

TRANSFORMATION	SPLINE CURVE
CIRCLE	SPLINE SURFACE
COMPOSITE CURVE	POINT
COPIOUS DATA	NURBS CURVE
LINE	NURBS SURFACE

If you switch the **COMPOSITE CURVE** option on, then the following options will automatically be switched on:

CIRCLE	SPLINE CURVE
COPIOUS DATA	POINT
LINE	NURBS CURVE

NOTE

In cases of doubt, work with the preset values.

Switch the required options on and confirm with <OK> or <Close>.

<OK> ends the configuration of options and saves these in a file. When the processor is next called up, you can work with them again.

<Close> ends the configuration of options, applies and processes the configurations, but does **not** save them in a file. When the processor is next called up, you will therefore be working with the originally defined values.

20.7 IGES preprocessor

With the IGES preprocessor, you convert data in HOLOS format into IGES format for transmission to a CAD system.

The following rules apply for converting HOLOS models:

- Models which have been read into HOLOS-UX in IGES format, can be saved in IGES format.
- Models which have been generated in HOLOS-UX can be saved in IGES format.
  Please note that the surface segments of segmented surfaces must not have different polynomial degrees in one parameter direction. The polynomial degrees in the parameter directions u and v may be different, but each segment in the parameter direction u and in v must have the same polynomial degree.
- Models which have been read into HOLOS-UX in VDA-FS format can **not** be saved in IGES format, as this is not a VDA-IGES converter.
- The VDA header produced with HOLOS is applied to the IGES file.

20.7.1 Data conversion

Start the IGES preprocessor via <Export>-< IGES > (NT) or <IGES>-<IGES preprocessor> (UX).

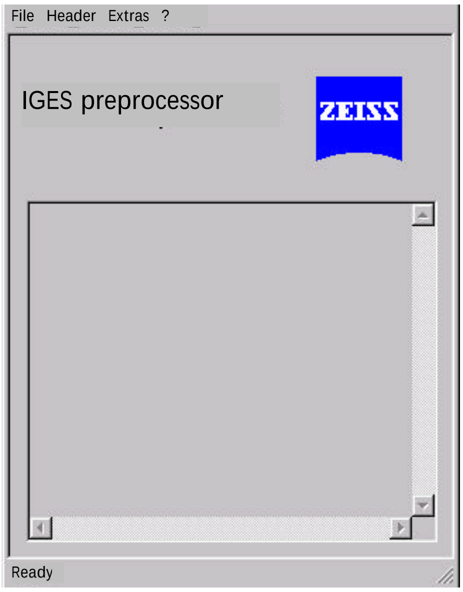


Figure 20-14

In the <Header> menu activate the <Enter header> option, if a header is to be entered into the IGES file. If the option is deactivated, no entry occurs.

Click on <Open> in the <File> menu, in order to select a HOLOS model for conversion.

The selection window is opened.

Select a model and click on <Open>.

The dialog window appears for defining the element to be converted.

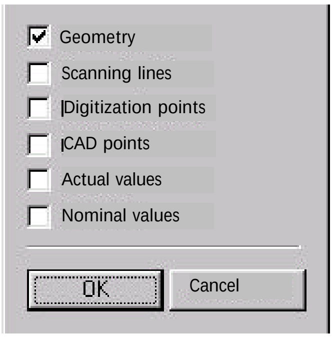


Figure 20-15

Geometry for applying geometrical data to the IGES file. Geometrical data are surfaces, curves and FACEs.

Scanning lines for applying scanning lines to the IGES file.

Digitization points for applying digitization points to the IGES file.

CAD points (NT) for applying CAD points to the IGES file.

Actual values for applying actual values to the VDA file.

Nominal values for applying nominal values to the VDA file.

In the options for actual or nominal values, a window opens for selecting the corresponding model files. Select the files and click on <Open>.

Activate the relevant options and click on <OK>.

You will obtain a display of the elements for conversion.

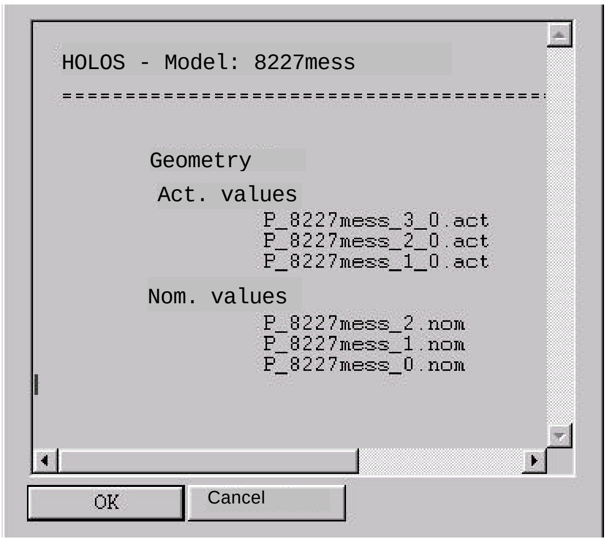


Figure 20-16

Click on <OK> to start the conversion.

If an IGES file with the same name already exists, you can choose whether you wish to generate a new file or overwrite the old one.

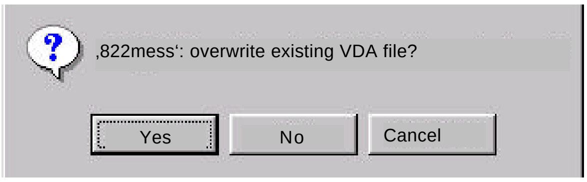


Figure 20-17

Click on <Yes> to overwrite the file, or <No> to generate a new file.

The HOLOS model is converted into an IGES file. You will receive information on the screen about the IGES data or about the converted elements.

20.7.2 Record and info file

Two files are generated during data conversion:

Record file

The record file contains information on converted HOLOS models.

Display:
In the “File” menu of the IGES processor

Info file

The info file contains information on the last generated IGES file.

Display:
In the “File” menu of the IGES processor

20.7.3 Header

Via the <Header> function you define entries for the header in accordance with VDAIS for the IGES file.

Select the <Header> function.

The input window is displayed.

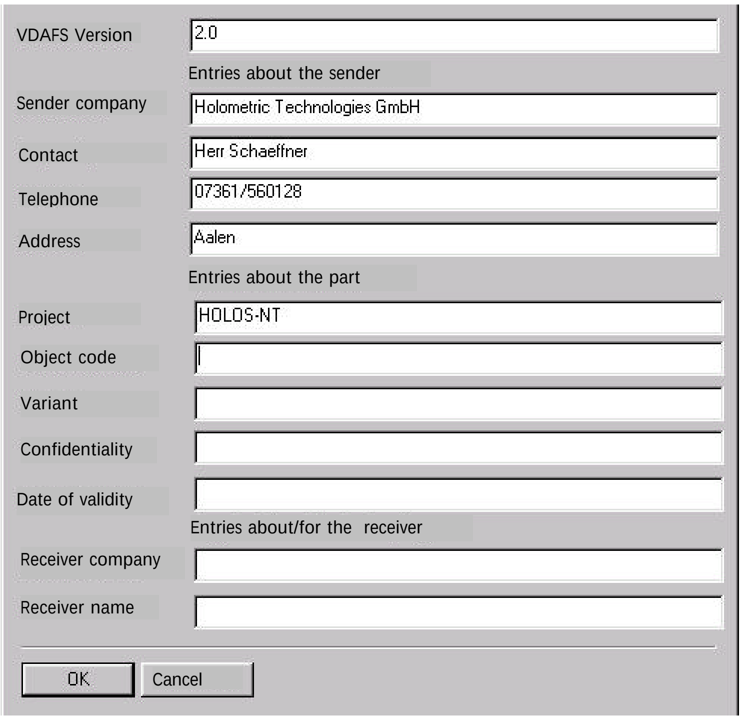


Figure 20-18

Fill in the required input fields and click on <OK> to accept your entries.

Activate the <Enter header> option in the <Header> menu if a header is to be entered into the IGES file. If the option is deactivated, no entry occurs.

20.7.4 Extras (NT)

Via the <Extras> menu you can specify the directories for storing IGES files or HOLOS models.

The standard directories are:
...\HOLOS-NT\daten\iges
...\HOLOS-NT\daten\model

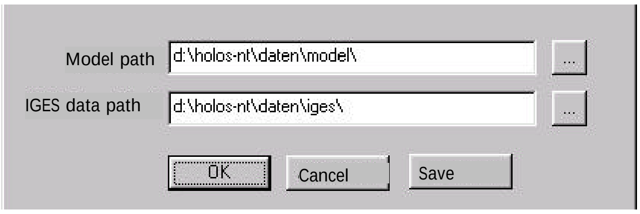


Figure 20-19

- Enter new directories for storing your data.
- Click on <Save> if you wish to permanently save the settings. Click on <OK> if the setting is only to apply once. When the IGES preprocessor is next started, the standard settings will then apply again.

20.8 IGES analyzer (NT)

You can analyze IGES files with the IGES analyzer.

You will receive information on the screen about which IGES elements are contained in the IGES file.
A HOLOS model is not generated.

The analyzer is operated like the IGES post processor.

A Installation

A.1 NT version

The software for HOLOS-NT is available either on CD-Rom or on floppy disks.

Insert the HOLOS data carrier into the drive.

Open Windows Explorer

Click on the drive letter of the data carrier.

Click on Setup.exe, in order to start installation of HOLOS-NT.

The welcome screen for installing HOLOS-NT appears.

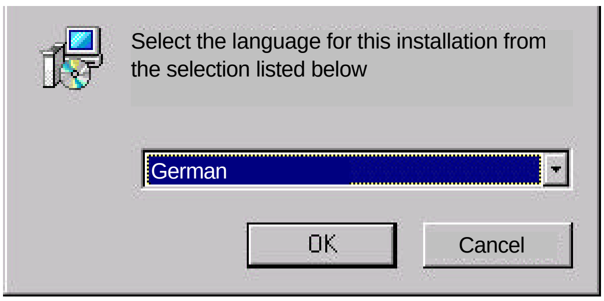


Figure A-1

Select a language and click on "OK".

The installation program is started.

A window appears for entering the installation directory.

Accept the proposed directory or click on "Search" to select another directory in the Data Manager.
Then click on <Continue>.

A window appears for entering the required installation:

Network
Languages
Standard

Network

Holos is installed on a server.

You must then access the server from each client and start the program setup.exe under \network in the installation directory.

Languages

HOLOS is automatically installed in German. You must additionally install all other languages with this function.

Standard

The complete program package is installed in the German language.

A window appears for entering the program group.

Accept the proposal or overwrite it with another name.
Click on <Continue> to continue the installation.

A window appears for selecting your measuring software.

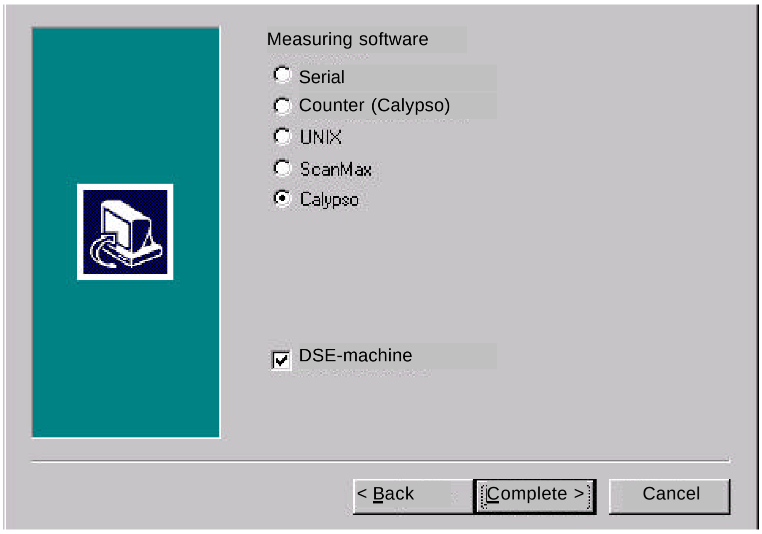


Figure A-2

Click on the measuring software you are using.

If you are using a horizontal arm measuring machine with a rotation-swivel device, the "DSE machine" option must be selected.

⇘ Click on <Complete> to conclude the installation.

A.2 UX version

A.2.1 General

As a rule, the software is preinstalled on the hardware supplied to you. If this is not the case, you will receive a DATtape which contains your software.

To install the software, perform the following steps:

1. End all current processes.
2. Log onto your system as a super user.
3. Insert the DAT-tape provided into your DAT-drive.
4. Your software is installed by entering the following command:

update -s /dev/dat “*“

/dev/dat denotes the device file for the DAT drive. If another system is defined in your system, you must use this here accordingly.

5. Log on as a super user.
6. Start your system.

Your HOLOS software is now installed.

Various parameters must be defined for a new installation. The relevant necessary steps are contained in this description.

When installing a software update, the steps described above must also be performed. However, the subsequent installation operations do not need to be repeated, as the defined parameters are not overwritten.

A.2.2 Prerequisites

When installing HOLOS-UX, a "/users/holos/" directory is set up. All files required for the HOLOS processes are located in this directory. The only exception is formed by the resource files for the user interface under X-Windows, as certain resources must also be available for other processes.

The drivers for the graphic software and for graphic output to external media (plotter, printer), are not permanently programmed, but are read into the system via environment variables when the respective program is started. This enables adaptations to different peripheral devices. However, to do this, various definitions are necessary, which as a rule are transferred from the CZ installation tools.

These environment variables are defined either in a ".kshrc" or ".profile" local file in the user's home directory.

Graphic screen

The driver for output on a graphic screen is defined via the environment variable SB_OUTDRIVER.

The sox11 driver is predefined upon delivery:

HOLOS_SB_OUTDRIVER=sox11
export HOLOS_SB_OUTDRIVER

Plotter output

The driver for output to a plotter or ink jet printer (Paintjet) is defined via the SB_PLTDRIVER environment variable. This variable can have the following values:
(1) SB_PLTDRIVER=hpgl
(2) SB_PLTDRIVER=CADplt
(3) SB_PLTDRIVER=CADplt2

All three drivers provide HPGL files as output, which can be output to a network-installed plotter or ink jet printer (Paintjet).

The table shown below shows which driver you must define for which plotter during system installation.

This table is an excerpt from the Hewlett-Packard literature; for this reason, no guarantee can be provided of its correctness. Should problems arise with different plotters, please consult a system administrator.

Driver	Plotter supported		
HP-GL	HP 7440A	HP 7470A	HP 7475A
hpgl	HP 7550A	HP 7570A	HP 7575A
	HP 7576A	HP 7580A	HP 7580B
	HP 7585B	HP 7586B	HP 7595A
	HP 7596A	HP C1600A	HP C1601A
HP-GL	HP 7510A	HP 7550A	HP 7570A
CADplt	HP 7575A	HP 7576A	HP 7580B
	HP 7585B	HP 7586B	HP 7595A
	HP 7596A,	HP C1600A	HP C1601A
HP-GL/2	HP 7595B	HP 7596B	HP 7599A
CADplt2	HP C1600A	HP C1601A	HP C1602A
	HP C1620A	HP C1625A	HP C1627A
	HP C1629A	HP C1631A	

When the system is delivered, the following driver is predefined for plotter output:

```
SB_PLTDRIVER=CADplt
export SB_PLTDRIVER
```

Plotter pens

Configure the pens for output to a plotter, as follows:

Pen 1: Black
Pen 2: Red
Pen 3: Yellow
Pen 4: Green
Pen 5: (any, as not used)
Pen 6: blue
Pen 7: (any, as not used)
Pen 8: (any, as not used)

X-Windows environment

To initialize X-Windows functionality of the user interface, the display variable DISPLAY must be defined. As a rule, this is given after installation of the software. The definition of these variables is located in the ".x11start" local file in the user's home directory.

In the case of installation on the system console:

: ${DISPLAY=`hostname`:0.0}; export DISPLAY

or

DISPLAY=`hostname`:0.0
export DISPLAY

In the case of installation for an X-terminal:

: ${DISPLAY=xterm:0.0}; export DISPLAY

or

DISPLAY=xterm:0.0
export DISPLAY

Definition of the directory for the user interface resources under X-Windows occurs via the XAPPLRESDIR environment variable. All resource files necessary for the Zeiss software are located in this directory. This directory is set up when the software is installed and contains all necessary entries.

Definition of the XAPPLRESDIR environment variables occurs in the ".x11start" local file in the user's home directory and as a rule is transferred from the CZ installation tools.

: ${XAPPLRESDIR:="/users/zeiss/app_defs/"}; export XAPPLRESDIR

or

XAPPLRESDIR="/users/zeiss/app_defs/"
export XAPPLRESDIR

The remaining installation operations, such as setting up all necessary directories and special entries in system files, are performed by the HOLOS installation tool.

A.2.3 INSTALL installation tool

The HOLOS installation tool is called up with the "INSTALL" command. The HOLOS installation tool is used to define various parameters, such as language, measuring software on the coordinate measuring machine computer, as well as the drivers for the peripheral equipment.

Various entries in system files or system directories are required for installing the data communication modules for communication with the measuring software. The INSTALL installation tool must therefore have super user rights.

As a rule, this is performed during installation of the software. If, however, problems arise after installation, first of all the rights of the INSTALL installation tool should be checked. This is done with the "ll /users/holos/bin/" command/“. In the list which appears, the entry for INSTALL should look like the following:

-rwsrwsr-x 1 root sys 1149101 Jan 11 15:58 INSTALL

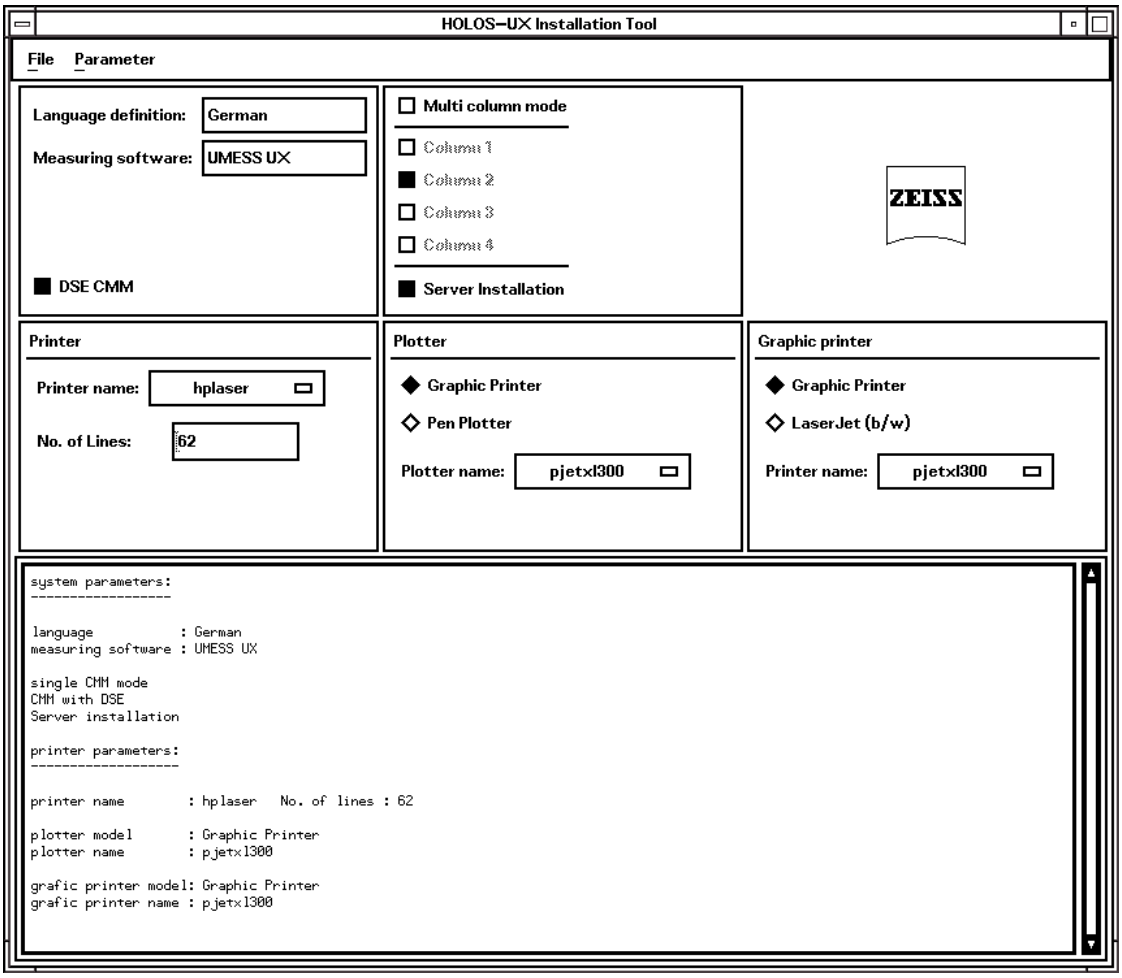


Figure A-3

In an input mask, you define the parameters in the INSTALL installation tool:

- language definition
- measuring software
- printer (printer settings)
- plotter (plotter settings)
- graphic printer (graphic printer settings)

- Language definition

Here you set the language for text outputs and for the texts in the user interface.

By clicking on the menu button for language definition with the left mouse button, a menu appears with the available languages. The respective language definition is adopted by clicking with the right mouse button.

The default language is German. If a language is set which is not present on the system, the system switches back to the default setting.

Measuring software

Here you define the measuring software on the coordinate measuring machine computer.

By clicking with the left mouse button on the menu button for definition of the measuring software, a menu appears with the systems available. The respective system is applied by clicking with the right mouse button.

- UMESS 300: Measuring system on CMS systems
- UMESS UX: Measuring system on UNIX systems
- UMESS 1000: Measuring system on HP 1000 systems
- SCANMAX

For communication with the relevant measuring software, various entries must be made in system files or system directories. The INSTALL installation tool must therefore have super user rights (see above).

Peripheral devices

Here you define the connections with the peripheral devices, such as printer, plotter and graphic printer. The peripheral devices can either be installed locally on the HOLOS computer or on another computer within the network.

When such devices are installed in the system, device names are allocated. These device names must be stated in the INSTALL installation tool.

Printer parameters

printer name	Name of a printer installed in the system. All text outputs (e.g. measuring records) are output to the printer defined here.
lines	Number of lines per output page.

Plotter parameters

plotter model	When installing a plotter (pen plotter or Paintjet) you must define the type and name of the plotter. By clicking with the left mouse button on the menu button for definition of the plotter type, a menu appears with the plotter systems available. Adopt the respective plotter type by clicking on it with the right mouse button.
plotter name	Name of a plotter installed in the system

Graphic printer parameters

printer model	When installing a graphic printer, you must define the type and name of the graphic printer. By clicking with the left mouse button on the menu button for definition of the printer type, a menu appears with the printer systems available. Adopt the respective printer type by clicking on it with the right mouse button.
printer name	Name of a graphic printer installed in the system.

Graphic printers serve to output graphics dumps. The entire content of the graphic window is output to the printer. This enables outputs which cannot be output to the plotter (e.g. the representation of deviations in chromatic coordinates) to also be put onto paper.

Color images can be output on Paintjet and Deskjet graphic printers. On the Laserjet, only black-white images can be output.

On principle, the output of measuring records is also possible on the graphic printer. However, as the Paintjet and Deskjet types in particular are relatively slow, this is not recommended when using one of these graphic printers.

Parameters

The set parameters can be saved in the menu bar in the <Parameters> function or displayed in the bottom text window.

Exit the INSTALL installation tool

The INSTALL installation tool is exited via the <Quit> function in the <File> menu.

If the set parameters have not been saved via the <Save parameters> function, the user will be asked to save his changes when exiting the INSTALL installation tool. A dialog window appears for this purpose:

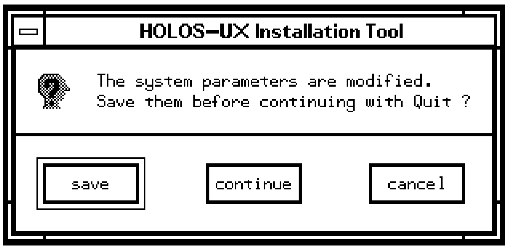


Figure A-4

Use the <Save> button to apply the set parameters and to continue the previously defined action (<Quit> or <Start HOLOS>).

The <Continue> key continues the previously defined action without applying the defined parameter values.

The <Cancel> key cancels the action, and the INSTALL installation tool remains active.

B HOLOS - Settings

The HOLOS Settings program is a separate program for HOLOS global settings. It is started via the <Start> menu in the task bar.

The settings only become effective when HOLOS is restarted.

B.1 Measuring software

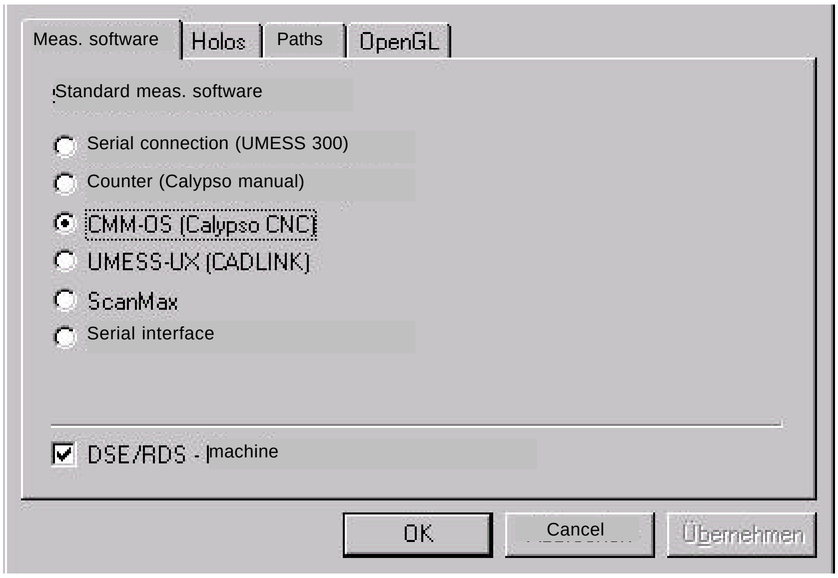


Figure B-1

As a rule, these settings are performed during installation, but can be modified here at any time.

NOTE

As a rule, you require administrator rights for this, because entries may need to be made in the system files.

Enter the relevant measuring software connection.

Click on <DSE machine>, if you are using a horizontal arm measuring machine with a rotational-swivel device.

B.2 Holos

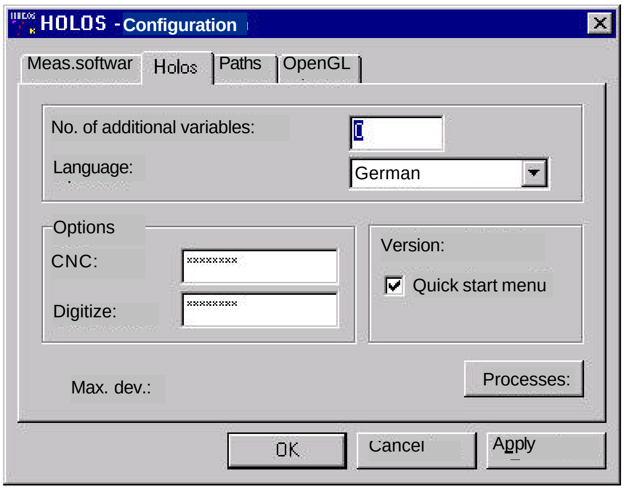


Figure B-2

No. of additional variables	State the number of variables which you wish to configure yourself. You can, e.g. also output them in the record header or in the frame.
Language	Language for the program operation. Make sure that the relevant language file is available, before changing the language. The languages German, English, French and Spanish are supplied as standard. If the file is not available for the entered language, then HOLOS will automatically be started in German.
Options	Enter the code for the program options, which you will have obtained after ordering the options. CNC: Option for measurement in CNC mode Digitization: Option for digitization and for CAD functions
Version	Number of the last installed HOLOS version.
Quick start menu	With this option you can activate the HOLOS symbol on the task bar, which provides you with quick access to all HOLOS programs.

Fig. B-3

When you click with the right mouse button on the HOLOS symbol, the context menu is displayed:

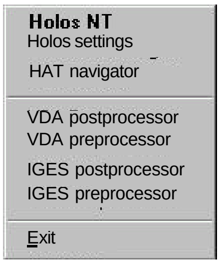


Figure B-4

Double-click on the symbol to start HOLOS-NT.

Processes

The running HOLOS processes are displayed.

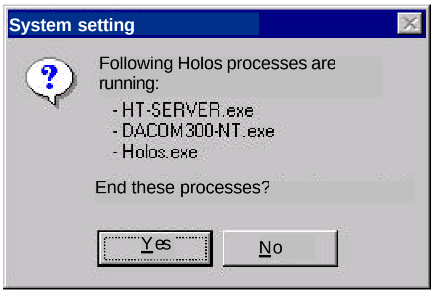


Figure B-5

HOLOS processes can be stopped by clicking on [Yes].

CAUTION

If possible, exit HOLOS-NT via the appropriate function provided. Exiting via the process display can in some cases lead to loss of data.

B.3 Paths

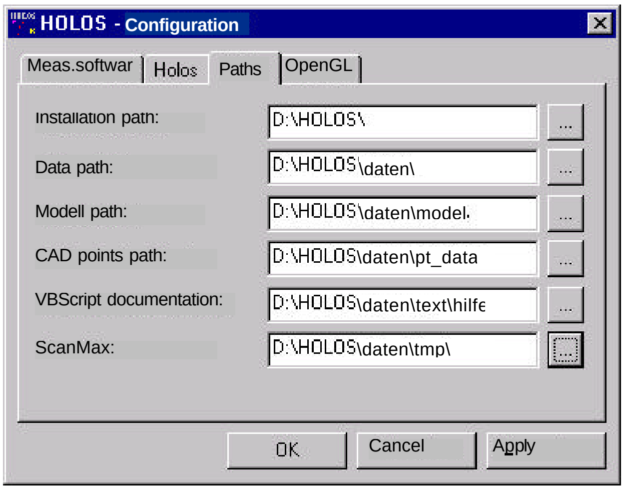


Figure B-6

Installation path	This is the directory in which HOLOS was installed. Only change the path if you have moved the complete HOLOS directory.
Data path	The "sys" and "tmp" directories of HOLOS are located here. Only change the path if you have moved the complete HOLOS directory.
Modell path	The HOLOS models are located here by default. You can change this path at any time.

NOTE

If you are working with UMESS measuring software, you can only start HOLOS macros which are located in this directory from here.

CAD points path	CAD points which are found during interpretation of a VDA or IGES file are stored here.
VBScript documentation	The documentation for the VBScript language is located here.
ScanMax	Data are exchanged with ScanMax via this path.

NOTE

VDA and IGES programs utilize their own path specifications.

B.4 OpenGL

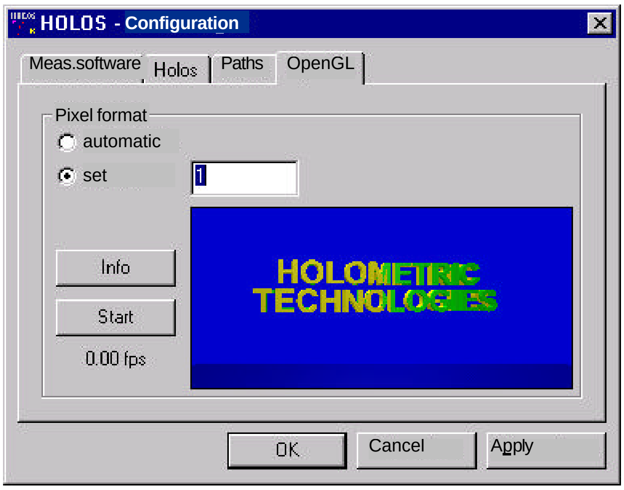


Figure B-7

The graphic output of HOLOS occurs with the help of OpenGL. With different graphic cards, you can accelerate the output by changing the pixel format.

Automatic — HOLOS itself searches for a suitable pixel format.

Set — You can define another pixel format by giving another number. As a rule, numbers between 1 and 7 are used.

Info — Information about the current pixel format.

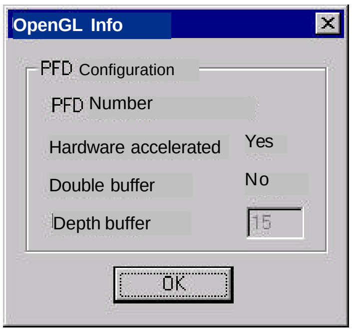


Figure B-8

Start	Test the set pixel format. If the "Holometric Technologies" logo turns, you have found a suitable pixel format. End the test with the same key. The fps value (windows per second) is then displayed. The pixel format with the highest "fps" value enables the quickest possible output.

C Key configuration (NT)

This part of the documentation contains the configurations for the keyboard and the mouse buttons, both for 3-button and for 2-button mice.

C.1 Three-button mouse

Mouse click without moving the mouse

Left mouse button	Selection
Middle mouse button	
Right mouse button	Popup menu

Mouse click with movement of the mouse

Left mouse button	Movement
Middle mouse button	Zoom
Right mouse button	Rotation

Double click

Middle mouse button	Input mode for individual selection
Middle mouse button + shift key	Switch drawing on and off

In design mode, the mouse buttons are configured as follows:

Mouse click without moving the mouse

Left mouse button	Selection
Middle mouse button	
Right mouse button	Popup menu

Mouse click with movement of the mouse

Left mouse button	Move selected objects
Middle mouse button	
Right mouse button	

Double click

Left mouse button	Change properties of clicked on graphic element
Middle mouse button	Selection input mode
Middle mouse button + shift key	Switch drawing on and off

C.2 Two button mouse

Mouse click without moving the mouse

Left mouse button	Selection
Right mouse button + control key	Popup menu

Mouse click with movement of the mouse

Left mouse button	Movement
Right mouse button	Rotation

Double click

Right mouse button	Input mode for individual selection
Right mouse button + shift key	Switch drawing on and off

In design mode, the mouse buttons are configured as follows:

Mouse click without moving the mouse

Left mouse button	Selection
Right mouse button + control key	Popup menu

Mouse click with movement of the mouse

Left mouse button	Move selected objects
Right mouse button	

Double click

Left mouse button	Change properties of clicked on element
Right mouse button	Selection input mode
Right mouse button + shift key	Switch drawing on and off

Draw line

Left mouse button + control key	Select start point on the model

Draw rectangle

Left mouse button + control key	Define representation area of the model

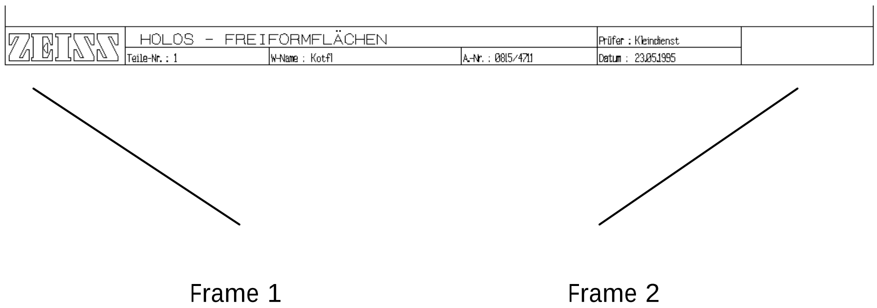
C.3 Keyboard configuration / hotkeys:

Key/key combination	Function
Control + D	Rotate orientation
G	Define group
Control + G	Extend group
Control + L	Lofting, curves>surface
M	Mask selected objects
Control + N	New model
Control + O	Open model
Control + S	Save model
Del	Delete selected objects
Number block +	Switch frame on/off
Number block -	Switch graphic elements on/off
Number block 0	Refresh
Number block 1	Surfaces on/off
Number block 2	Faces on/off
Number block 3	Scanning lines on/off
Number block 4	Patches on/off
Number block 5	Curves on/off
Number block 6	Scanning points on/off
Number block 7	Digitization points on/off
Number block 8	CAD points on/off
Number block 9	Alignment points on/off
Arrow key down	Zoom minus
Arrow key up	Zoom Plus
Arrow key right	Rotation right
Arrow key left	Rotation left
Arrow keys + Control	Movement

You can define further key combinations for activating functions yourself, via the options in the <Extras> menu.

D Logo variables in the graphic record (UX)

HOLOS-UX offers you the option of entering your own logos in the frame of the graphic record .



Frame positions for entering your own logos

Both frame positions are defined by their coordinates on the screen.

Frame 1:	Value range X: 0 ... 140	Y: 0 ... 40
Frame 2:	Value range X: 920 ... 1075	Y: 0 ... 40

The symbols for representing your own logos must be located within these ranges.
A maximum of 200 points can be specified per symbol.
The data must be saved in a file called logo.dat in the directory /users/holos/sys.
The file contains data according to the following format:

```
Number of symbols
Number of points symbol 1,  fill_flag, color, closed_flag
x1   y1
x2   y2
:       :
:       :
xn  yn
Number of points symbol 2, fill_flag, color, closed_flag
:      :
:      :
```

Appendix D, Logo variables

Number of symbols:	defines how many symbols are contained in the file
No. of symbols point n:	defines the number of points of which the symbol consists
fill_flag:	defines whether a symbol is displayed with filled color surface, or only as a boundary polygon 0 = not filled 1 = filled
color:	defines the color in which the symbol is t be displayed 0 = white 2 = red 3 = yellow 4 = green 5 = cyan 6 = blue 7 = magenta
closed_flag:	Defines whether a symbol is a closed polygon line 0 = not closed 1 = closed

Example

```
2
4  0  2  1
0        0
140            0
140            40
0              40
4  0  3  1
920            0
1075   0
1075   40
920            40
```

This example of the logo.dat file draws a rectangle at both frame positions.

The rectangles are not filled with color (fill_flag = 0) and are closed (closed_flag = 1).

Rectangle 1 is displayed in red (color = 2), rectangle 2 is displayed in yellow (color = 3).

NOTE

A logo.dat file in which the ZEISS logo is defined is supplied with the system. Before you generate your own file, you should copy this file as a backup.

Glossary

Bézier polygon

The Bézier polygon is an approximation to the actual surface. In other words: a curved surface is described approximately with flat polygons (point-to-point lines). With the help of Bézier polygons, irregularities in the curvature course of a generated surface are made visible.

Continuity

A function is constant at a particular point

- if the function value exists at this point and
- if the left and right limit value is identical to the function value at this point.

In other words: A function is constant at a particular point, if it has neither a gap nor a jump at this point.

HOLOS-UX differentiates three degrees of continuity:

C0-constant or constant:	the above mentioned condition only applies for the function.
C1-constant or constant tangent:	the above mentioned condition also applies for the first branch of the function.
C2-constant or constant curvature:	the above mentioned condition also applies for the second branch of the function.

Deviation

A deviation in the sense of free form surface measuring technology is the vertical distance of a measured spatial point from a free form surface (distance in the direction of the surface normals).

Digitization

Digitization denotes the collection of individual points on a surface. In HOLOS-UX the digitization function serves to mathematically describe unknown contours. Different functions are provided for the generation of individual points, curves and surfaces for this purpose. Individual points and curves can be further processed to form surfaces in a graphic interactive manner.

Digitization process

In HOLOS-UX the digitization process denotes the procedure which collects further surface points on the workpiece surface, to achieve a defined accuracy. For the most part, this procedure runs automatically, so that relatively few interactions are required by the operator.

FACE

A FACE is a free form surface limited by curves.

The surface with its surface segments can go beyond an actually existing surface geometry in its mathematical description. The actual surface is then also limited by curves on the mathematical surface in its mathematical description.

“Fixing in space”

The nominal/actual value pairs must be selected so that the contact points of a model are also the nominal points.
This prevents the model from moving in an undesired direction during transformation and incorrect values occurring as a result.

Free form surface, SURF

A free form surface is any smooth surface with a repeatedly curved contour. A free form surface can consist of several surface segments (patches), which are independently mathematically described in themselves.

Patch-Ident

Patch-Ident denotes the identification of a surface segment after any probing on the workpiece surface. The deviation on this surface segment is also determined.

Polynomial degree

Maximum exponent in the mathematical description of a curve or surface. The maximum polynomial degree which HOLOS can process from external data, is 20.

Reparameterization

In HOLOS-UX reparameterization denotes the amalgamation of patches to form a new surface with a modified number of surface segments, i.e. modified parameters S and T. Reparameterization serves to close and define surface transitions.

Segmentation

Division of a free form surface into patches (= surface segments). The parameters S and T define the division of patches into rows and columns.

Surface segment, patch

A patch is a segment of a free form surface which is completely mathematically described in itself. Patches have a local parameterization in the parameters u and v, where u and v can take on values between 0 and 1.

Transformation

Computing rule for converting values. The alignment of measured points to surface points provides a computing rule for converting machine coordinates to workpiece coordinates. This computing rule is called transformation.

Translation

Translate the coordinate system without rotation.

Index

D

E

F

G

H

I

Index

K

L

M

N

O

P

Q

R

Index

S

Index