NANOSUITE® Software

Software Reference Manual

Electron Beam Lithography
FIB Nanofabrication
Nanoengineering
Reverse Engineering

SOFTWARE AND MANUALS

RAITH
NANOFABRICATION

RAITH
NANOFABRICATION

Release 6.0

Product:	*NanoSuite* Software Reference
Document number	4 - 03 -7.0
Date of release:	March 2016

Declaration

All information in this manual has been carefully prepared and is considered to be accurate and complete. If there is any doubt about any detail or if you require additional information, please contact Raith GmbH or one of its representatives.

Copyright

Trademarks

Microsoft, Windows, Windows XP	are registered trademarks of Microsoft Inc.
Auto CAD, DXF	are registered trademarks of Autodesk Inc.
GDSII	is a registered trademark of Calma GE.

Support

Raith GmbH (head office)		
Konrad-Adenauer Allee 8	Phone	+49 231 95004 - 460
44263 Dortmund	Fax	+49 231 95004 - 460
Germany	E-mail	support@raith.com

Raith America Inc.		
1377 Long Island Motor Parkway	Phone	+1 631 738 9500
Suite 101	Fax	+1 631 738 2055
New York 11749	E-mail	support@raithusa.com
USA		

Raith Asia Ltd.

Two Chinachem Exchange Square	Phone	+852 2887 6828
No. 338 King’s Road	Fax	+852 2887 6122
Floor 7, Unit 05-06	E-mail	support@rraithasia.com
Hong Kong		

Contents

Chapter 1
Notes on this Manual

The NANOSUITE Reference Manual complements the basic documentation of a Raith lithography system. It describes:

- the operation, control and maintenance of the system via the NANOSUITE software,
- all dialogs, functions and control elements of the NANOSUITE software.

The NANOSUITE Reference Manual is intended for:

- all users of an Raith electron or ion optics based lithography systems,
- local software administrators,
- Raith trained and locally named system experts,
- authorized Raith service engineers.

In addition to this manual there is a tutorial available which introduces the first steps and basic tasks using the NANOSUITE software.

1 Notation and symbols

The following notation and symbols are used in this manual:

Reset A gray markup represents a key word which indicates the control element or software command to which the current description refers.

➡ This symbol specifies which menu entries or dialogs you must choose in succession. Example: Select **File** ➡ **Open Script**.

⇒ This symbol within the text indicates a link to further information within this manual or elsewhere in the basic documentation.

▶ This symbol indicates an operation that has to be performed by the user.

EO Indicates paragraphs with information specific to electron optics.

IO Indicates paragraphs with information specific to ion optics.

Loading samples Italics indicate the title of a section, table or figure referred to at a particular point in the manual.

Note: Keyword introducing a paragraph with more information about a function in a dialog.

Recomm-endation: Keyword introducing a paragraph with more entry recommendations for a function in a dialog.

F1 A black markup indicates a keyboard command.

Practical examples are highlighted like this.

Figure 1-1:

2 Safety Messages and Notes

DANGER

These notes refer to risks of personal injury. The alert symbol used depends on the potential risk.

The instructions for danger prevention must be obeyed.

NOTICE

These notes refer to risks of material damage.

The instructions are to be obeyed in order to avoid a faulty operation of the system, material damage or the destruction of the system.

Special notes and background information are identified by the light bulb symbol, as used for this paragraph.

These notes refer to activities which are only to be performed by the following persons:

– Raith trained and locally named system experts,
– authorized Raith service engineers.

These notes indicate how a dialog or function can be called up in the NANOSUITE software.

These kinds of notes indicate a link to further information
- within this manual,
- elsewhere in the basic documentation,
- elsewhere in additional manuals for optional hardware and software equipment.

Chapter 2
Getting Started

This chapter gives an overview of the installation, uninstallation and maintenance of the Raith NANOSUITE Software as well as the PC. It describes:

- The system and user rights
(⇒ *System requirements* on page 2-1),
- How to install and uninstall the software
(⇒ *Installing the* **NANOSUITE** *software* on page 2-2,
⇒ *Uninstalling the* **NANOSUITE** *software* on page 2-9),
- How to install and reset service packs
(⇒ *Installing service packs* on page 2-7,
⇒ *Resetting service packs* on page 2-11),
- The help tools and the Raith support centers
(⇒ *Help and support* on page 2-12),
- The Raith maintenance facilities and maintenance jobs to be performed by the user (⇒ *Debugging and maintenance* on page 2-13).

1 System requirements

The following operating systems and minimum hardware are required to run the NANOSUITE software reliably:

- Windows 2000 SP 4, XP, Vista, 7.0
- RAM: 512 MB
- Processor: Pentium 4, minimum 2.2 GHz
- Hard disk space: Minimum 1 GB

For installing, uninstalling and maintenance of the software and the computer, administrator rights are required. It is recommended to carry out these tasks via the local software administrator.

2 Setup

Generally, the NANOSUITE software will be set up during the initial startup of the lithography system by a Raith service technician. In the following cases, the software can be installed and updated by the user:

- To provide offline licenses for testing and user configuration purposes on workstations, which are independent from the lithography system workstation. User-specific data files can be taken over to the lithography system workstation via the company network or a portable memory medium. The files are to be copied into the corresponding folder in the local user directory of the software (⇒ *File administration* on page 3-8).

- To install new software releases as demo software.
- To install new service packs for online and offline licenses.
- To reset the NANOSUITE software to a former installed service pack.

2.1 Installing the NANOSUITE software

The installation will be explained in the following subsections.

2.1.1 Installation tools

For the installation of the software, the following tools are required:

- CD which contains the following files, to install the software:
  - **Setup <program name><release no.#.#.#>.EXE** ,
  - **customer.ldb** contains license and setup information.
- Hardware license key for enabling the software.

2.1.2 Components of the NANOSUITE software

The following software components will be installed during the installation process automatically and can be called up via the Windows **Start** directory.

- NANOSUITE application as the operating software of Raith electron beam or ion beam lithography systems.
- **Raith Protocol tool** to protocol, evaluate and administer procedural data,
- **Raith Scripting Help** offering predefined objects and classes for the creation of individual scripts for procedural automation functions, (⇒ *Automation* on page 15-1)

2.1.3 Installing the software

The installation will be performed via an installation wizard leading the user through the installation process. Install the software as follows:

▶ Close any Raith applications open on your computer.

▶ Have the hardware license key ready to hand.

▶ Insert the installation CD and double click on the **Setup <program name><release no.#.#.#>.EXE** file.

The software data files will be extracted and locally stored in the **<product name>** directory.

If the extracting process is finished, the **Raith Installation Wizard** dialog will be opened:

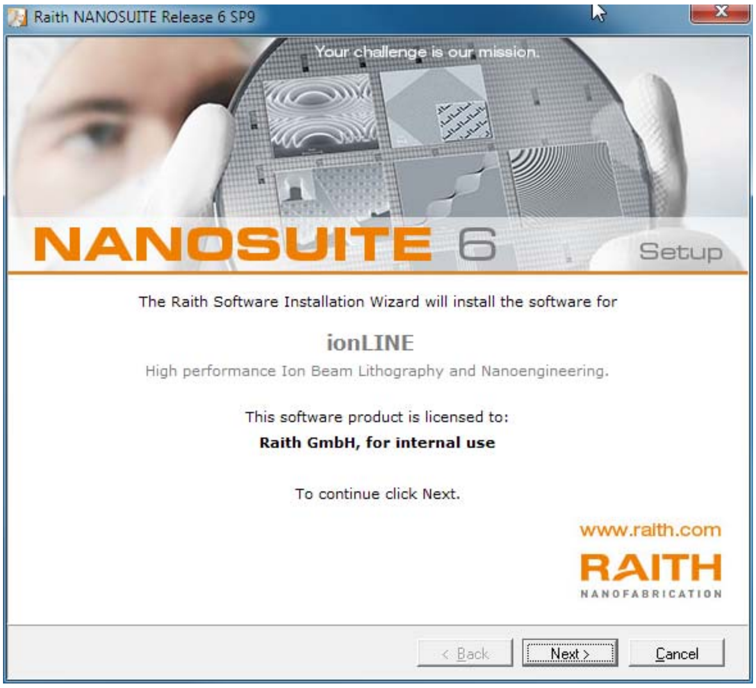


Figure 2-1: Software installation – Step 1 (Example)

▶ Click on **Next** to start the installation routine.

The License Agreement dialog will open:

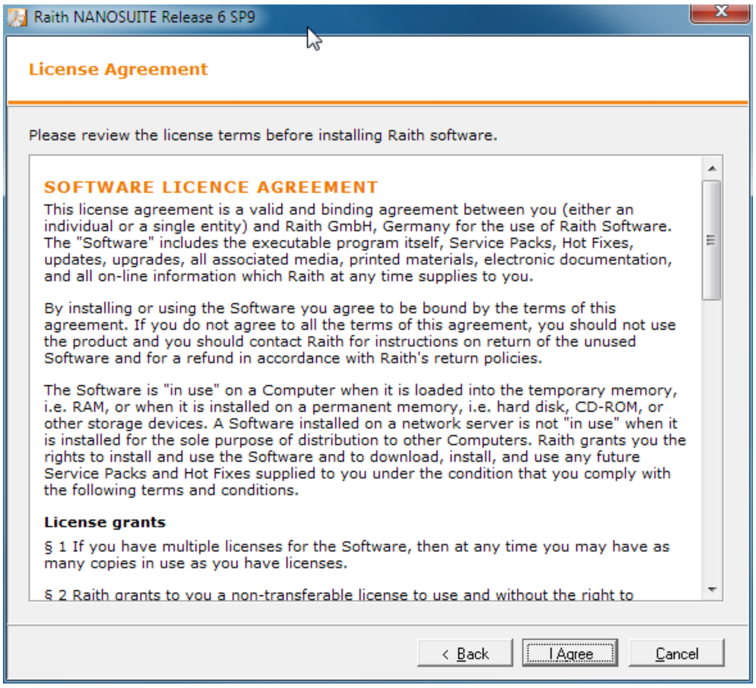


Figure 2-2: Software installation – Step 2 (Example)

▶ Confirm the agreement.

The **Selections** dialog will open, showing information about the current software status:

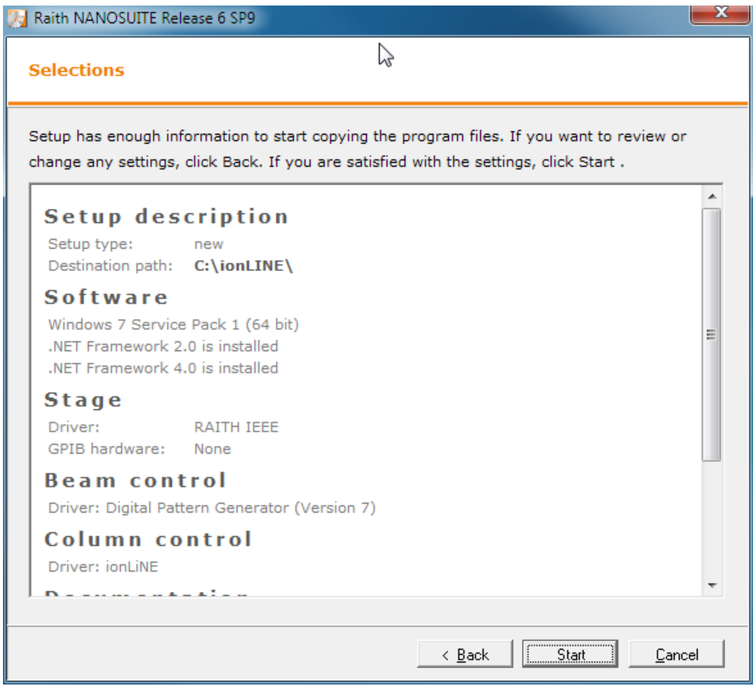


Figure 2-3: Software installation – Step 3 (Example)

▶ Click on **Start**.

The following dialog will open, showing the installation steps currently executed or finished:

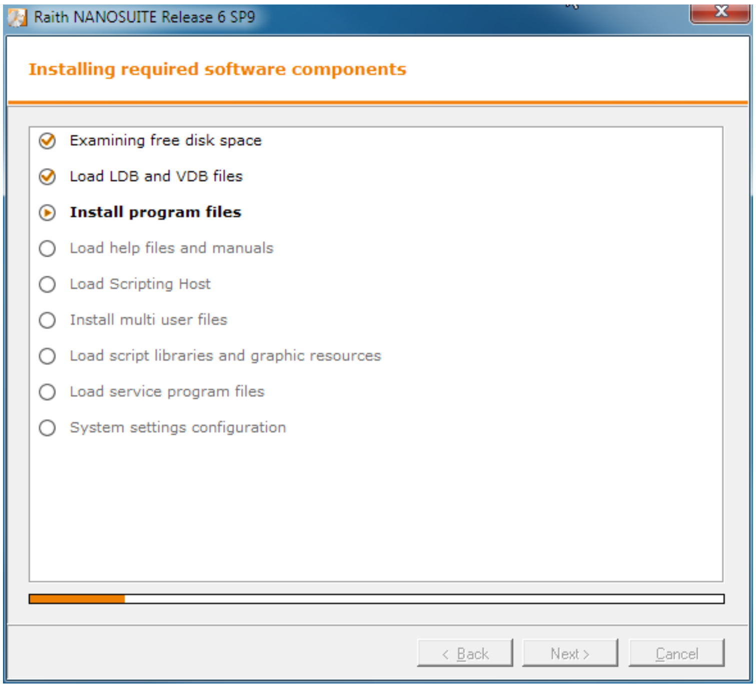


Figure 2-4: Software installation – Step 4 (Example)

If the installation is completed, a corresponding icon link will be displayed on the desktop – e.g.  – with a prompt to finish the installation:

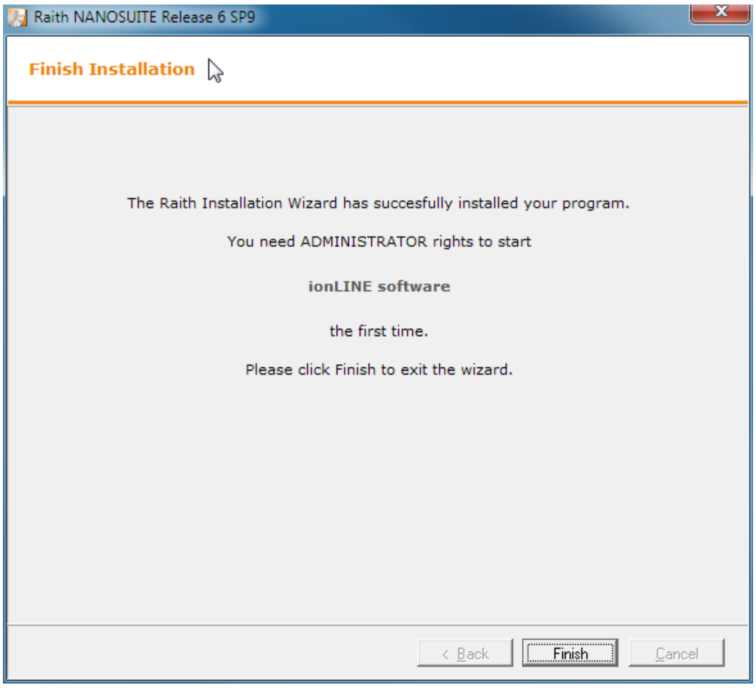


Figure 2-5: Software installation – Step 4 (Example)

- Click on **Finish**.

The software will ask the user to restart the PC:

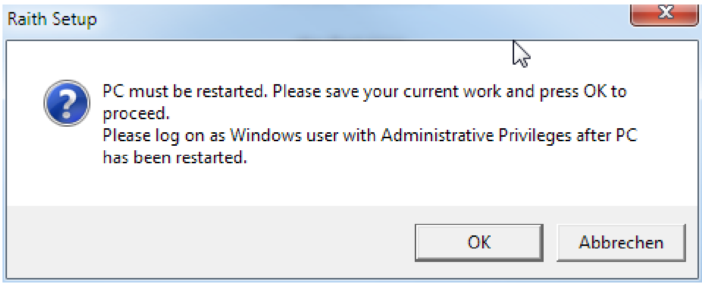


Figure 2-6: Error – Missing hardware license key

- Click on **OK** and restart the PC.

The NANOSUITE software can be maintained remotely by the Raith support team (⇒ *Raith support and remote maintenance* on page 2-13). To enable this feature, the corresponding scripting files have to be registered in the Windows registry. To do this, start NANOSUITE software initially with administrator rights.

- Start the NANOSUITE software
  - by double clicking the corresponding icon link on the desktop or
  - via Windows **Start** directory ➡ **Programs** ➡ **Raith** ➡ **<Product name>**.

The software will ask for the hardware license key:

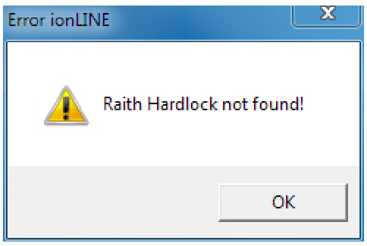


Figure 2-7: Error – Missing hardware license key

- Click on **OK** and plug in the hardware license key into a free USB port.
- Restart and close the NANOSUITE software with administrator rights (**User:** Administrator, **Password:** <empty>).
- Depending on the local regulations, you may change the administrator password and create new users.

For more details about starting the NANOSUITE software and the user administration, refer to (⇒ *Starting the software* on page 4-8).

The software is installed and ready for use.

2.2 Installing service packs

The NANOSUITE software is optimized continuously and can be updated via service packs. New available service packs can be downloaded from the **Home Support** center on the Raith website via a customer login.

The service packs are specific to the Raith lithography systems and the software release. That means the only service packs can be installed which correspond e.g. to a RAITH150*TWO* system and the software release 6.0. The specifications of a service pack are indicated with the file name syntax:
sp <productname><release no.#><service pack no.#.#>.

During the installation process, the currently installed service pack version will be backed up. The back-up can be used for a reset of a service pack in case of a malfunction after the installation (⇒ *Resetting service packs* on page 2-11).

Proceed as follows to install a service pack:

▶ Download an adequate service pack from Raith **Home Support** page.

▶ Close any Raith applications open on your computer.

▶ Double click on the **sp<productname><release no.#><service pack no.#.#>.exe** file.

The service pack files will be extracted and locally stored.

If the extracting process is finished, the **Raith Service Pack** dialog will open, showing basic information about the software and the service pack version:

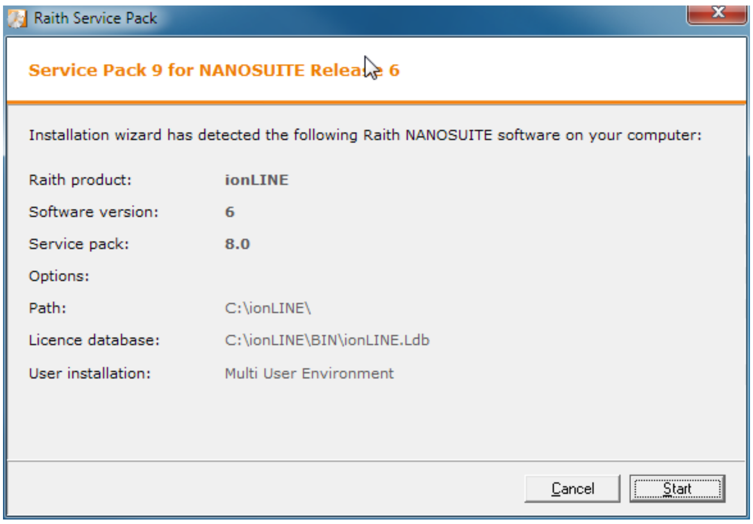


Figure 2-8: Installing service pack – Step 1 (Example)

▶ Check the information displayed. Cancel the installation if the new service pack doesn‘t correspond to the lithography system or the currently installed software release.

- Click on **Start** to start the installation routine.

  The following dialog will be opened showing the installation steps currently executed or finished:

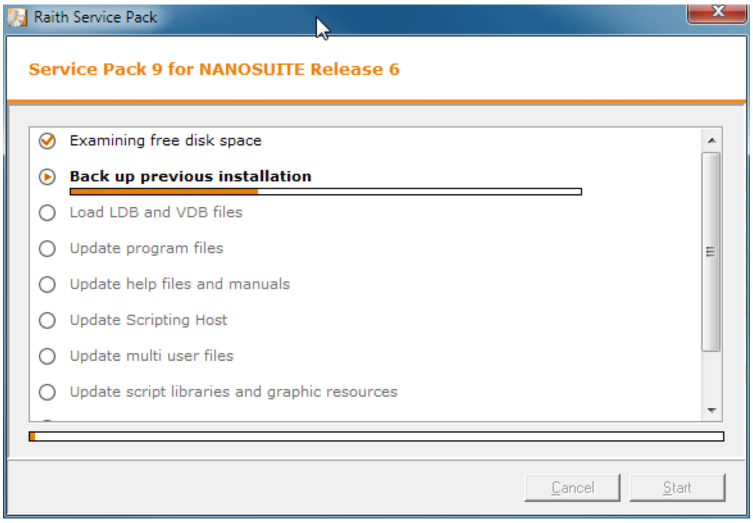


Figure 2-9: Installing service pack – Step 2 (Example)

If the installation is complete, the system will ask to confirm the installation and start the NANOSUITE software:

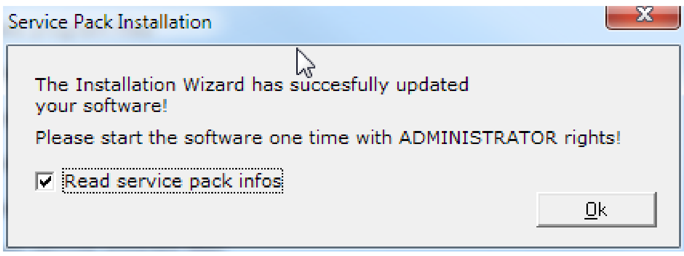


Figure 2-10: Installing service pack – Step 3 (Example)

- Enable **Read service pack infos** and click on **OK**.

  A **readme.txt** file will open showing information about the new service pack.

- Read the information and close the file.
- Start the NANOSUITE software.
  - by double clicking the corresponding icon link on the desktop or
  - via Windows **Start** directory ➡ **Programs** ➡ **Raith** ➡ **<Product name>**.

The software may ask for the hardware license key. In this case, confirm the error message, unplug and replug the key, and start the NANOSUITE software again.

- Login with administrator rights (**User:** Administrator, **Password:** <empty>) and close the NANOSUITE software .
- Depending on the local regulations, you may change the administrator password and create new users.

For more details about starting the NANOSUITE software and the user administration refer to (⇒ *Starting the software* on page 4-8).

The service pack is installed and the software ready for use for any authorized user.

3 Uninstalling the NANOSUITE software

Uninstall the software as follows:

- Close any Raith applications open on your computer.
- Open the Windows uninstallation function via Windows **Start** directory ➡ **Settings** ➡ **Control Panel** ➡ **Uninstall a program.**
- In the **Change or Remove program** tab select your Raith product – e.g. **ionLINE**:

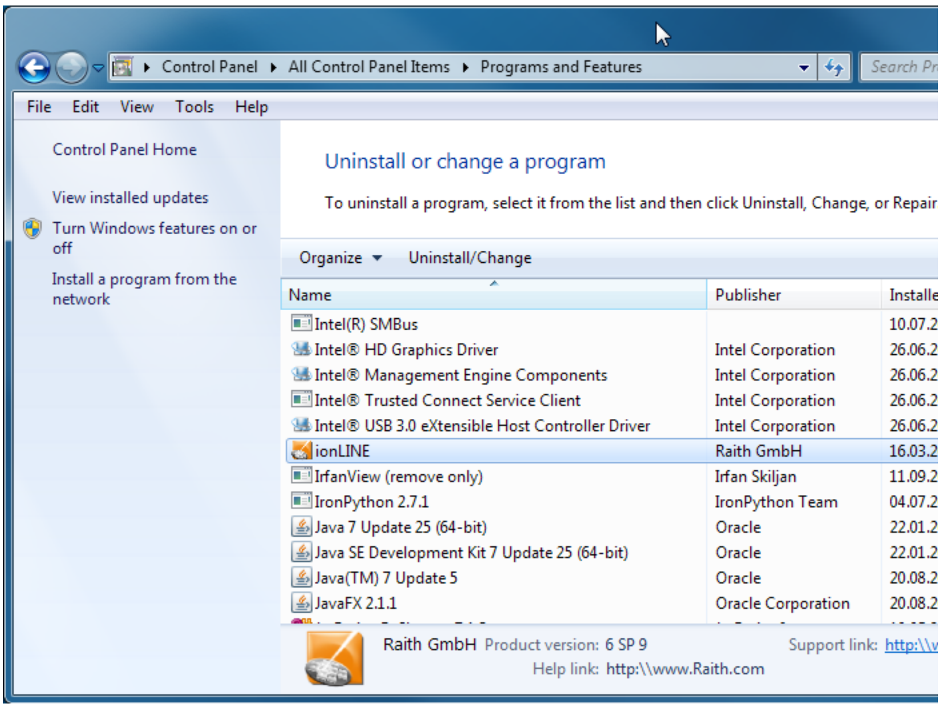


Figure 2-11: Uninstalling software – Step 1 (Example)

- Click on **Change/Remove**.

The **Raith Uninstall** dialog will be opened:

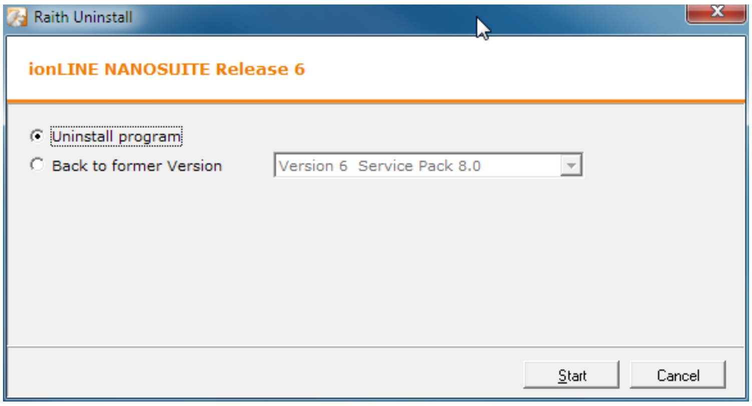


Figure 2-12: Uninstalling software – Step 2 (Example)

- Choose **Uninstall program**.
- Click on **Start** to start the uninstallation routine.

The following dialog will be opened showing the uninstallation steps currently executed or finished:

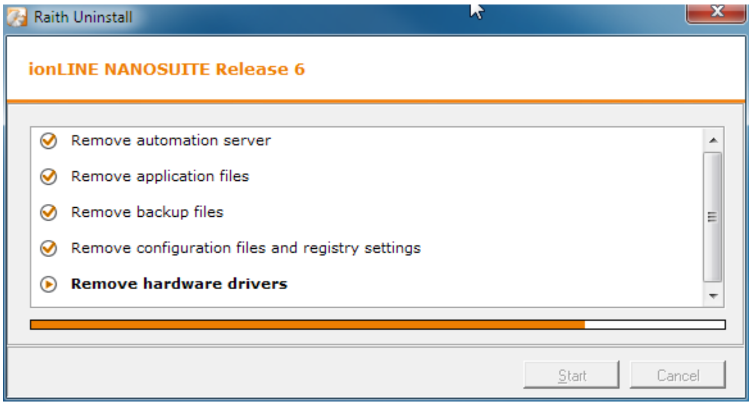


Figure 2-13: Uninstalling – Step 3 (Example)

If the uninstallation routine is completed, the dialog will be closed and all components of the NANOSUITE software will be deleted from your computer.

The **HASP HL Device Driver** remains on your computer in order to accelerate further installation and enable different Raith products to be installed on the same PC.

4 Resetting service packs

Depending on the individual software configuration on your computer, the NANOSUITE software may run with faults after installing a new service pack. To set the system back to a executable operating status as fast as possible, it may be useful to reset the service pack to a former installed version.

Before resetting a service pack, consult Raith support to discuss possible reasons for malfunctions.

Proceed as follows to reset a service pack:

- Close any Raith applications open on your computer.
- Open the Windows uninstallation function via Windows **Start** directory ➡ **Settings** ➡ **Control Panel** ➡ **Add or Remove Programs**.
- In the **Change or Remove program** tab select your Raith software product – e.g. e_LiNE.
- Click on **Change/Remove**.

The **Raith Uninstall** dialog will be opened:

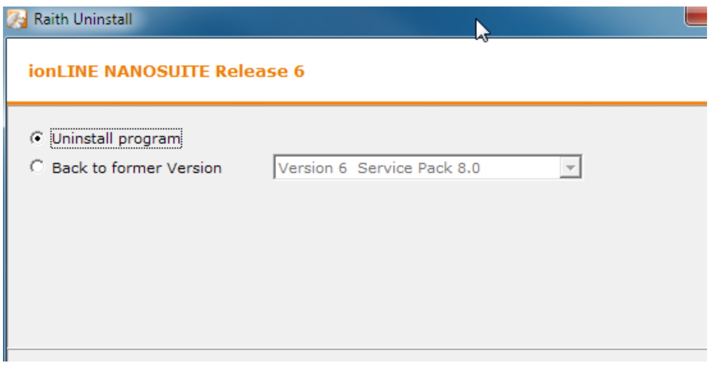


Figure 2-14: Resetting service pack – Step 2 (Example)

- Choose **Back to former version** and select a service pack from the combo box.
- Click on **Start**.

The service pack will be set to the selected former version.

- If the resetting is finished, restart and close the NANOSUITE software with administrator rights.

The service pack is reset and the software is ready for use.

For more details about starting the NANOSUITE software refer to
(⇒ *Starting the software* on page 4-8).

5 Help and support

The handling of the NANOSUITE software is fully documented. The documentation is available as in-product help and print documentation. Additionally, the Raith support centers help with customer specific problems.

5.1 In-product Help

The In-product Help provides access to all documentation and instructional content available at the time the software ships. It is available through the Help menu (**?**) of the main menu bar or by pressing the **F1** key throughout the NANOSUITE software.

A context-sensitive help offers dialog oriented help topics by clicking on **?** in every single dialog. Additionally, a bubble help is available to many functions and input fields offering important information and input recommendations. The bubble help is indicated by moving the cursor above the corresponding field in between the dialog.

5.2 Print documentation

A plain Software Manual as linked PDF file is delivered on the installation CD and, additionally, will be stored in the **<product name>** ➡ **help** directory during the installation process.

Additionally, a short, printed Getting Started Guide is delivered with the installation CD.

5.3 Raith support centers

For further information and support, the following Raith service center are available:

5.3.1 Raith support Europe, Middle East, India

Raith GmbH
Konrad-Adenauer-Allee 8 - PHOENIX West
44263 Dortmund
Germany

Phone: +49 231 95004 499

Email: support@raith.com

5.3.2 Raith support North America

Raith America, Inc.
1377 Long Island Motor Parkway
Suite 101
Islandia, New York 11779
USA

Phone: +1 631 738 9500

Fax: +1 631 738 2055

Email: support@raithamerica.com

5.3.3 Raith support Asia, Pacific

Raith Asia Ltd.
Two Chinachem Exchange Square
No. 338 King's Road
Floor 7, Unit 05-006
North Point
Hong Kong

Phone: +852 2887 6828

Fax: +852 6627 6122

Email: support@raithasia.com

5.3.4 Raith Home Support center

In the **Home Support** center on the Raith website, the following service are available for registered customers:

- Download center which provides the current service packs for the different Raith systems.
- Large File Exchange transfer system for a save and quick data exchange with the Raith support team.

6 Debugging and maintenance

Debugging and maintenance of the NANOSUITE software will usually be performed via the Raith support or a Raith service technician. Basic maintenance tasks relating to the PC hardware and the operating system are to be performed by the user.

6.1 Raith support and remote maintenance

In cases of malfunctions of the operating system or the Raith NANOSUITE software, contact one of the Raith support centers (). The Raith support offers:

- first level support for debugging of minor problems by the user,
- debugging solutions for complex problems, based on generated debugging data. The NANOSUITE software includes a logger function which can protocol the operation status of single system components and software modules. The data will be stored in a specific data file (**<filename>.LOG**) in the software root directory. The data will be evaluated by the Raith support team and serves for further debugging purposes.

For more information about the logger function refer to (⇒ *Using the Logger* on page 4-10).

- remote access to your computer by the Raith support team (Raith remote maintenance). Using this service, the support team can control and maintain the lithography system directly and discuss potential problems with the customer interactively.

To use the Raith remote maintenance service, internet access is needed.

6.2 Maintenance by the user

To guarantee a reliable operating of the lithography system and to prevent data loss, the following maintenance jobs for the PC have to be performed by the user periodically:

- defragmentation,
- data back-up,

Back up your data periodically on an external memory device. In case of a malfunction of the PCs hard drive, all user settings and user files can be lost. Then, Raith can only recover the default settings and system files.

- installation of actual service packs

  The NANOSUITE software is optimized continuously. Therefore it is recommended to install new service packs periodically. The service packs can be downloaded from the Raith website via a user login.

- installation and update of adequate antivirus protection software and firewall, if the PC is connected to the internet or data is to be exchanged with other PCs.

To use the Raith remote maintenance service, internet access is needed (⇒ *Raith support and remote maintenance* on page 2-13).

Chapter 3
Introduction to the NanoSuite Software

This chapter introduces into the concept, structure and most important features of the NANOSUITE software. It gives an overview of the:

- Basic concept and main features
(⇒ *Concept and main features* on page 3-1),
- User-levels an access rights
(⇒ *User levels* on page 3-6),
- Options for customizing offered by the software
(⇒ *Options for customizing* on page 3-7),
- Structure of the NANOSUITE root directory and the file administration
(⇒ *File administration* on page 3-8).

1 Concept and main features

This section describes the basic concept and main features of the NANO-SUITE software.

1.1 Integrated approach and multi-user management

The NANOSUITE software is an integrated, MS Windows based software which offers all lithography tasks from one source, from creating a design up to performing patterning. The software is graphically based and includes a multi-user management with different user levels. This allows an application-specific and individual working of one or more users at one lithography system. Settings, files, and tasks are stored separately from each user.

The NANOSUITE software is designed comprehensively and integrated into Raith electron optic and ion optics-based lithography systems.

1.2 Modular principle

The NANOSUITE software consists of a variety of functional software modules which also provide a seamless link to the system control. Based on a default hardware configuration, the software can be extended by optional modules offering additional lithography capabilities or the control of additional hardware components (e.g. Gas Injection System, Nanomanipulators). Therefore, the NANOSUITE software can be adapted to user needs.

For permanent and fast access, the modules are organized via a control bar on the right hand side of the operating interface. With a click on a corresponding icon link, the module dialogs and control elements will be displayed on the right hand side of the control bar.

For a detailed description of the operating interface and the control bar refer to (⇒ *User Interface* on page 4-1).

1.3 Automation function and drag and drop

The NANOSUITE software offers an interface to Microsoft scripting capabilities which allows the creation of application-specific tasks using the Microsoft scripting language. Storing and recalling of these user-defined scripts allows for powerful task automation. Additionally, a scripting help is available, which offers the description of Raith specific objects that allows the use of the familiar "object.method" syntax with a rich set of properties and methods to control the NANOSUITE software via JScript or VBScript.

The concept of drag and drop operation is extensively used throughout the Raith software to ensure easy operation of rather complex tasks – e.g. metrology measurements, simple drive commands, and the execution of macros. All required setups are handled by the system.

For a detailed description how to use the automation and drag and drop function refer to (⇒ *Automation* on page 15-1, ⇒ *User Interface* on page 4-1)

1.4 Stage to Sample Adjustment and UV windows

By default, the NANOSUITE software uses two different coordinate systems:

- XYZ coordinate system corresponding to the movement of the laser interferometer stage (stage coordinate system).
- UVW coordinate system which represents the individual coordinate systems of the sample (sample coordinate system) and the design.

The software operates UV windows in which the design of an application can be expected and edited in UV coordinates – such as Images, Wafermaps, Chipmaps and GDSII structure elements. For most applications, the user will work in UV coordinates. Individual functions of the UV windows differ depending on the software module currently active.

Working within the UV windows is facilitated by the toolbox with frequently used tools and commands. The tools are context-sensitive to the currently active UV window.

For a detailed description of the sample to stage adjustment and the handling of the UV windows refer to (⇒ *UV windows* on page 4-4, ⇒ *Stage to Sample Adjustment* on page 12-1)

1.5 Live process monitoring

Patterning processes can be monitored live – i.e. patterning and imaging can be performed simultaneously. The live process monitoring function is realized by one column.

1.6 Help functions

The handling of the NANOSUITE software is fully documented. The documentation is available as in-product help and print documentation.

The In-product Help is realized via:

- the Help menu (?) of the main menu bar or the **F1** key,
- a context sensitive help at dialog level,
- a bubble help offering information and input recommendation to single functions and input fields.

For cases of malfunctioning of the operating system or the NANOSUITE software, a logger function is included which can protocol the operation status of single system components and software modules. The data will be evaluated by the Raith support team and serves for further debugging purposes.

For more information about help, support and the logger function refer to (⇒ *Logger* on page 4-35).

1.7 Main lithography capabilities

The following table gives an overview of the main lithography capabilities supported by the NANOSUITE software for performing extensive lithography tasks. Details and tasks are described in the corresponding chapters of this reference manual.

Table 3-1: Lithography capabilities of NANOSUITE software

Feature	Description
Patterning module	for patterning all types of design data directly from the extended GDSII database (⇒ *Patterning* on page 14-1). Main patterning features are: – Stage to sample adjustment (⇒ *Stage to Sample Adjustment* on page 12-1) – Beam to sample, writefield, and overlay alignment (⇒ *Beam to Sample Alignment* on page 13-1) – Mark recognition (⇒ *How to carry out a Writefield Alignment* on page 13-8) – Automatic dwell time correction and field stitching (⇒ *Enhanced Patterning Parameters* on page 14-27)
GDSII editor and viewer	for creating and patterning designs within the same software platform. The GDSII module makes it unnecessary to switch between different software packages for multiple tasks. Main features are: – Import and export filters for foreign file formats – e.g. CAD and Bitmaps. – Dose scaling. – Hierarchical designs. – Patterning attributes. (⇒ *GDSII Data Handling (Design Module)* on page 5-1)
Stage and column control	for the remote control of the stage and column. (⇒ *Column Control (option)* on page 7-1) (⇒ *Stage Control* on page 8-1)
Proximity effect correction	for the correction of the proximity effect via the NanoPECS^{TM} module. This effect can occur by creating uniform nanostructures or very tightly arranged structures of widely varying sizes. (⇒ *Software Reference* on page 6-30)
Imaging capabilities	for recording, modifying, and saving images. (⇒ *Image window* on page 10-32)
Positionlists	for storing various tasks to be executed one after the other. (⇒ *Positionlist* on page 9-1)

Table 3-1: Lithography capabilities of NANOSUITE software

Feature	Description
Wafer and CAD layout navigation	for a quick and easy review of written structures and complete patterns – particularly in the handling of wafers and masks. (⇒ *Working with Wafers* on page 16-1)
Chip scanning	for reverse engineering applications that enable to create GDSII designs on a basis of image scans. (⇒ *Scan Manager* on page 11-1)

2 User levels

The NANOSUITE software is a multi-user platform with up to four different user levels. Any level has defined access to all functions of the NANOSUITE software. The following levels are attainable:

- USR-level:

  An USR-level user has only restricted access to the software modules and functions.

- EXP-level:

  Depending on the systems settings and hardware equipment, users with EXP-level rights have some more capabilities than USR-level user – e.g. with respect to the Gas Injection System.

- SYS-level:

  Users with SYS-level rights

  – have access to all modules and functions of the software, except a small number of service functions for the IONLINE column.

  – can create, delete, and manage users via the **Administration** module.

  – can adjust global system settings.

  By default, the local software administrator and Raith trained system experts possess SYS-level rights.

- Master-level:

  Master-level users are users with special rights. Only for the *ionLiNE* system, an additional Master-level has been created. For some service tools, the SYS-level is not sufficient to be permitted certain aspects of software access. Therefore a Master level has been created to permit this access.

How to create and configure user levels refer to (⇒ *Administration* on page 4-16)

3 Options for customizing

Due to the multi-user management of the software, every user can save and restore individual or project-specific settings. The following features and functions of the NANOSUITE software are customizable:

- Desktop configuration

  The current configuration of the desktop can be saved and restored – i.e. the current position of windows and dialogs used for a special application or project. Up to four configurations are possible.

How to configure the desktop refer to (⇒ *Using projects for customizing* on page 4-7)

- Variable based functions

  Subject to the user level, some functions of the software can be customized or rather enabled or disabled using variables – e.g. the DXF scale factor or the logger function.

How to use variables refer to (⇒ *Status Report* on page 4-14)

- User-specific settings

  Some settings – e.g. the step size and dose for patterning – as well as scripts, files, and tasks are stored for every user individually. They will be restored after the next login and can be reset to default settings via the log-in command **Reset user configuration.**

  More typical user-specific settings are the writefield size, Scan Manager and Column Control parameter sets, as well as linescan filter settings.

How to reset the user-specific settings refer to (⇒ *Starting the software* on page 4-15)

- Project-specific settings

  Beside the desktop configuration, other settings can be stored depending on the application or project. These settings are mainly: The mode how document windows are to be opened after starting the software (document opening mode) and if they have to overlay each other.

How to store project-specific settings refer to (⇒ *Using projects for customizing* on page 4-7).

Generally, all project-, user- and variable-related settings can be prepared on an offline workstation – e.g. for testing purposes. They can be uploaded to the system work station via an intranet or a portable memory medium.

4 File administration

User and project-specific files will be automatically stored in the corresponding directories of the NANOSUITE directory. They will be restored after the users login and choose of a project and are available in the corresponding modules.

Due to the multi-user platform, user and project-specific data files are exchangeable between the NANOSUITE software and other application programs – e.g. GDSII data. Furthermore, users will transfer files from offline work stations to the system workstation and vice versa – e.g. for testing purposes with offline software licenses.

For the automatic restoration of user and project-specific settings, it is necessary to store the files appropriate to the NANOSUITE directory structure. Therefore, all users have to know the structure of the NANOSUITE root directory.

(⇒ Figure 3-1 on page 3-8) shows the NANOSUITE root directory structure:

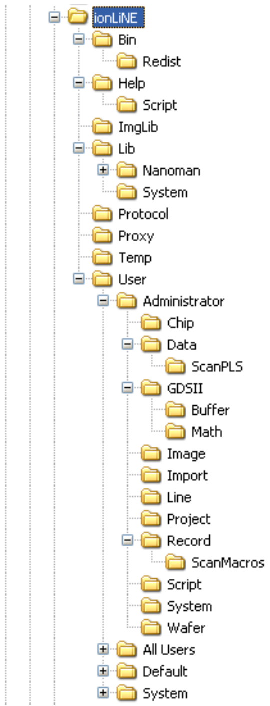


Figure 3-1: Directory tree of the NANOSUITE root directory

The following table describes the contents of the user relevant directories and names the file formats stored inside them:

Table 3-2: NANOSUITE directory structure and file formats

Directory/ Subdirectory	Subdirectory/Files
.LOG	File format generated by the logger function. The file name can be edited by the user. Otherwise the default file name will be used with an index number, which will be incremented every time the logger is started.
Bin	Contains all binary and system configuration files which are necessary for running the software.
Help	Contains the in-product help files (**CHM** files) and the following print documentation as PDF files: – NANOSUITE Software Manual – NANOSUITE Tutorial – Scripting help
ImgLib	Contains all image files used in the GUI of the software.
Lib	Script library which contains all scripting files used to run the software and hardware options:
System	Contains system-specific script files.
.js	Contains script libraries with common functions that are used in the system or user specific script files – e.g. **Column.js**, **Comon.js**, **LIS.js**.
Protocol	Contains Protocol tool and protocol database files.
Proxy	Contains files used for the proximity effect correction of the NanoPECS module (**.EXE**).
Temp	Contains the temporarily files generated by the soft-ware.
User	Contains the user-specific settings and files:
Administrator	Only attainable for SYS- and Master-Level users only. The directory structure corresponds to the <**User**> directory.
All Users	Contains settings and files concerning to all users – e.g. wafer holder maps, GDSII database examples. The structure corresponds to the **User XX** directory.
<User>	Contains all user and project-specific settings and files for each individual user defined. The directory consists of the following subdirectories:

Table 3-2: NANOSUITE directory structure and file formats

Directory/ Subdirectory	Subdirectory/Files
Chip	Contains chip layout files used for creating chip designs (**.CLO**).
Data	Contains positionlist files used for defining positionlist scans (**.PLS**).
GDSII	Contains GDSII database files used for creating GDSII designs (**.CSF**, **.GDS**, **.HIR**, **.LAY**). – **MATH** subdirectory: Mathematically calculated structure elements – e.g. curved elements (**.MAT**). – **Backup** subdirectory: Backups of the GDSII database files generated after every saving of a GDSII database.
Image	– Imagescan datasets used for image scans. **Note:** An imagescan dataset consists of two data files – BMP and SSC files. For moving datasets from one computer to another or another platform both files have to be uploaded. – Image files created by the electron or ion optics – e.g. **.TIFF**-files.
Import	Contains imported files of the software import filters – e.g. **.DXF**-files.
Line	Contains linescan files used for line scan data storage (**.lsc**).
Project	Contains project-specific files stored for special applications (**.DSK**).
Record\ Scan-Macros	Contains macros generated by the Scan Manager.
Script	Contains user defined scripting files created and offered in the **Automation** module.
System	Contains configuration files for Writefield and Scan Manager, proximity correction, linescan and imagescan filter.
Wafer	Contains Wafer layout files (**.WLO**).
System	Directory of a special SYS-level user. This user has no individual VDB settings. Rather, all settings made by this SYS-level user will be stored directly in the system VDB.

Table 3-2: NANOSUITE directory structure and file formats

Directory/ Subdirectory	Subdirectory/Files
Default	Directory with all files which are used when creating a new user (default user directory).

Chapter 4
User Interface

This chapter describes the User Interface as well as some principles of the software. The use of the software is based on a control bar, containing several icons for the various areas of software operation. Selecting the relevant icon will open all associated windows, in which the parameters for the operation of the system can be selected.

The startup of the Raith NANOSUITE software program is described here, as well as the **Administration** dialog, in which new users and user rights can be set up.

The functionality and purpose of the UV windows are described.

The functionality of the **Logger** window, which is used for debugging modules in the software, is described.

- Program workspace
(⇒ *Overview of the operating interface* on page 4-11),
- Control bar
(⇒ *Control bar* on page 4-12),
- Drag and drop concept
(⇒ *Drag and drop concepts* on page 4-31),
- Using projects for customizing
(⇒ *Using projects for customizing* on page 4-7).
- Starting the software
(⇒ *Starting the software* on page 4-8),
- How to edit a password
(⇒ *Changing the password and resetting the user configuration* on page 4-8),
- Explanation of **Administration** dialog to setup User levels
(⇒ *Administration* on page 4-16),
- Explanation of the coordinate systems
(⇒ *Types of coordinate systems* on page 4-5),
- Background information about the functionality of the UV windows
(⇒ *UV windows* on page 4-19),
- Creating a UV window overlay
(⇒ *Creating a UV window overlay* on page 4-8),
- Description of the main menu commands (⇒ *Main menu* on page 4-19),
- Dynamic toolbox description (⇒ *Using the Toolbox* on page 4-23),

- Explanation of the mouse and keyboard commands
(⇒ *Mouse and keyboard commands* on page 4-30).
- Explanation of the Module setting
(⇒ *Module settings* on page 4-31)
- Background information about the functionality of the Logger window
(⇒ *Logger window* on page 4-36),
- Using the Logger (⇒ *Using the Logger* on page 4-10),
- Software description of the Logger settings within the **Settings** module (⇒ *Logger settings* on page 4-35),
- Detailed description of the Logger window
(⇒ *Logger window* on page 4-36),
- Information about the Logger log files
(⇒ *Logger Log folder* on page 4-37).

A Functional Description

The Raith program uses standard Windows controls with windows, icons and mouse. For details of how to operate Microsoft Windows, refer to the Microsoft public domain documentation. This chapter introduces the extensions used in the Raith program.

1 Control bar

The control bar summarizes all controls for the set up of all parameters. The bar itself consists of several icons. Parameters which are used together are opened together by selection of one icon. For example, the **Adjustments** tab contains the parameter set **Adjust UVW**, **Stage Control**, **Scan Manager. Coordinates** appears on all icons. The control bar contains the following icons: **Design**, **Column Control**, **Writefield Control, Adjustment, Patterning**, **Stage Control**, **Automation**, **Administration**, **Load Lock**, **Height Sensing**, **Gas Injection**, **Nanomanipulators** and **Service**. Please note that depending on the options selected, some tabs might not be visible.

2 User hierarchy

Through use of the **Administration** icon on the control bar, each user can be assigned to one of three login (or access) levels: USR (ordinary user), EXP (expert user), and SYS (system level user). The ability to add a new user is available only for those with system level access. By default, the Administrator possesses this level. For all other login levels, access to this routine is denied.

- **USR-level** (User) can be viewed by all other users. It is possible to create several USR-users for the system. A user has only restricted access to the software modules.
- **SYS-level** (Administrator). An administrator has access to most modules of the software, except the service modules. It is possible to create several SYS-administrators. SYS-level can view user management. This level permits the creation of new users or the deletion of users. It is also permitted to adjust global system settings.

Master-level (Master with special rights). An additional SYS-level has been created, since for some service tools, the SYS-level is not sufficient to be permitted certain aspects of software access. Therefore a Master level has been created to permit this access.

3 Overview of windows types

There are several types of windows available, such as document windows, UV windows and add-on windows.

3.1 Document windows

Document windows are resizable windows, which can be placed anywhere on the screen.

Most of the document windows can be opened via the main menu. The following **File** items are resizable document windows: **Image, Linescan**, **Positionlist**, **Wafermap**, **Chipmap**, **Script** and **Navigator**. Some of them, such as the Image, Linescan and Positionlist windows can be opened several times (⇒ *Main menu* on page 4-26).

UV windows are also resizable document windows, showing the UV coordinates, (⇒ *UV windows* on page 4-19).

Some of them can be opened from corresponding control bar modules only. As an example, **GDSII Viewer** and **GDSII Editor** can only be accessed from the **GDSII Database** module.

At the bottom of the workspace is the **Logger** window, which is typically minimized, but can be expanded or maximized when the logging or debugging of the software operation is required. The Logger window is also a resizable window and is explained in detail in the Logger section of this chapter, (⇒ *Logger* on page 4-35).

3.2 UV windows

UV windows are special windows in which each point can be identified by coordinates. Such windows include **Images**, **Wafermaps**, **Chipmaps**, **GDSII Viewer** and **GDSII Editor**. If the cursor is placed within a UV window the status bar indicates the corresponding coordinates.

The U and V coordinates with the globe button are the corresponding sample coordinates, The U and V coordinates with the window button are the corresponding coordinates relative to the window origin. The X and Y coordinates are the corresponding stage coordinates.

Within the Raith NANOSUITE software, two different coordinate systems are used. The **XY** coordinate system relates to the stage movement and an additional **UV** coordinate system defines the design coordinates on the sample. The coordinate systems and transformations between them are explained in **Stage to Sample Adjustment**, (⇒ *Functional Description* on page 12-3), where software procedures carried out to perform a stage to sample adjustment are also described (⇒ *Tasks* on page 12-9).

The UV windows work with the UV coordinates. They enable overlays and other procedures to be carried out.

All UV windows have toolbox features, consisting of a generic toolbox, GDSII toolbox, Image toolbox and a Column toolbox. The toolbox is dynamically adjusted, depending on the type of UV window.

UV adjustment marks can be set to carry out any UV adjustments. (⇒ *Tasks* on page 4-7)

For most applications, the user will work in UV coordinates, including images, wafermap and GDSII design. It is therefore easier to create overlays, when all those windows use UV coordinates.

3.3 Add-on windows

The add-on windows that can be opened depend on the modules open at the time and the options installed on the system. The currently displayed windows and dialogs on the screen determine which further windows may become available.

4 Types of coordinate systems

It is typical, for nearly all application tasks, to find a specific location on a sample and then, for example, to write at this location a GDSII structure. To define the location, it is more convenient to apply a coordinate system to the sample. Finding this location is then simply a drive command within this coordinate system. The success of the principle depends on the accuracy of the applied transformation, as well as the accuracy and precision of the stage, compared to the required accuracy of positioning.

4.1 UVW coordinates

To define and then find a location, a coordinate system can be applied to the sample. For example, the center of a wafer is conventionally (0, 0) and an axis of this coordinate system is perpendicular to the direction of the flat. Within the Raith NANOSUITE software, this sample coordinate system is labeled using **U**, **V**, and **W.**

Please refer to **Stage to Sample Adjustment,** where the relationship between the XY and UV coordinate systems is explained, using schematics (⇒ *Coordinate systems and transformations* on page 12-3).

---

5 Logger functionality

Logger is used to document messages for debugging purposes. Since the Logger is fully integrated into the software, it is normally the first tool to be used, before any other debugging is initiated. The Logger window is easy to use and will give the first insights into any problem, before external modules are utilized for further debugging.

The data generated by the Logger can not be evaluated by the user, but the data does give vital debugging information to the Raith support team.

The Logger should only be initiated when advised by Raith, in order to locate a minor problem. The logger can be set to document a number of modules of operation. The user will receive instructions about which modules to activate.

After the logger data have been generated, the data can be sent by e-mail to Raith and the support team can track the source of the problem.

B Tasks

1 Using projects for customizing

The following tasks describe how the projects can be customized.

1.1 Window position

To store the position of a window:

- Choose 1, 2, 3, or 4 to switch to the desktop that should contain the window.
- Place the window on the desktop at the position at which it should appear in the future and adapt its size.
- Choose the button to open the **Save Desktop** dialog.
- Save this setting by clicking the **OK** button. Optionally, the **Name** for this Desktop can be changed before saving.

1.2 Document opening mode and overlay

To store the mode used when opening a new window or the displayed overlay:

- Activate the corresponding document window.
- Choose from **Menus Windows**➡**Save Settings ...** to open the **Pane** window.
- Select **Display** and an entry from the list to define what to overlay into this document window.
- Select **Close** so that the opened document will be closed before opening another document of the same type.
- Select **Tile** so that all opened documents of the same type will be tiled within the area of the first opened document.
- Optionally, choose **Delete** to restore the default.
- Accept the selection temporarily by pressing the **OK** button.
- Choose the button to open the **Save Desktop** dialog.
- Save this setting by clicking the **OK** button.

2 Starting the software

- To start the Raith NANOSUITE software, choose **Start➡Programs➡Raith** on the Windows desktop and then select the name of the product, e.g Quantum. Alternatively, the Raith NANOSUITE software can be started by double clicking a related icon placed on the Windows desktop.
- If the error message “Error 7: Raith Hardlock/Licence not available” appears, please ensure that the hard lock protection key is connected to a USB port of the computer.
- After starting the application, the login screen appears.
- Select a User name and type it to the dialog. Additionally, an available **Project** can be selected. Projects are necessary for storing and recalling specific session settings such as screen configurations, related files, etc.
- If the entered password is correct, the buttons **Options** and **Login** are enabled. The user may then change the password or reset the user configuration.
- Choose **Login** to start the Raith application with the selected project. Simultaneously, the application loads the settings, such as step size and dose, that were used during the previous session.
- Choose **Cancel** to interrupt the login procedure.

3 Changing the password and resetting the user configuration

- To change the login password, choose **Options➡Change password**. Enter and retype the new password in the appropriate dialog box.
- To restore the default user settings, choose **Options➡Reset user configuration**. This option deletes any user-specific settings, such as step size and dose, of the current user. Other settings which are stored in separate databases, such as microscope settings, writefield alignment values, and scan type definitions are not deleted.

4 Creating a UV window overlay

Overlays are window-specific. For example, it is possible to:

- create an image with a GDSII overlay, (⇒ *Drag and drop GDSII structures onto an Image* on page 10-9)
- create a wafermap with GDSII, (⇒ *Overview of Wafermap* on page 16-2)
- show video in a GDSII structure, (⇒ *Options menu* on page 5-96)
- create overlays with Positionlists

As an example, to create an image with a GDSII overlay, follow the steps outlined below;

- Open a GDSII design structure in the **GDSII Viewer**.
- Open a UV window such as an **Image** or **Wafermap**.
- Go to menu **Options**➡**UV Display**. From the **Overlay** windows list, select the overlay window that is to be overlaid with the currently selected UV window. Now one window will be overlaid with the other.
- Go to the **Origin** field and select **Window** in the dropdown list. Define the location of the Origin for the overlaid window: **UV system** refers to the Origin of the UV coordinate system, **Window** refers to the Origin of the Window and **Chip center** refers to the center of every chip in the Wafermap.
- Check the checkbox for **GDSII Viewer <Structure name>** in order to enable the overlay between the image and the GDSII structure.
- Select **Current** for the **UV-System** in the UV windows properties.
- Select the position of the **Window origin**, either in the center or the lower left.
- You may now define the **Offset** if either **Window+Offset** or **Chip center+Offset** location is selected.
- Press **OK** to apply the settings. Now you will see the **GDSII** structure overlaid on your image.

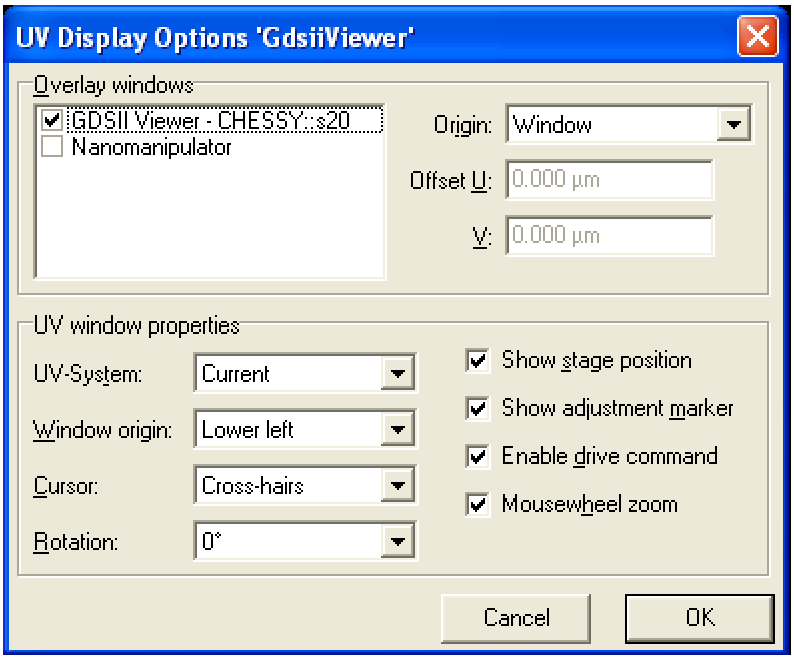


Figure 4-1: Selecting the parameters in the **UV Display Options** dialog

When working with chips, the chip internal coordinate system is very useful. Use the Chip center as the origin to create the overlay in the center of every single chip on the Wafermap.

Use the **Extras➡Max GDSII chip overlays** parameter to define the number of chips that can show the overlays simultaneously.

5 Using the Logger

- Go to **Extras➡Module Status** and locate the module that you wish to select for the Logger, e.g. Virtual Motor Control.
- Double click on the module, and set **Debug** to **On**. Then click the **Reset Module** button to make the changes valid.
- Start the Logger by selecting the **Logger** window, then click the **Start** button.
- As the Virtual Motor Control module was chosen in this example, the logger will now log all data when the stage is driven.
- Click on **Stop** to end the logger documentation. You can now change the data filename if you wish. The logger data file will be saved in the Raith directory **<Raith root >**.

If you do not change the log data filename, the same filename will be used with an index number, which will be incremented every time the logger is started. The history of the log file can be determined by the user.

C Software Reference

Firstly, an overview of the operating interface is given, before an explanation of its basic operation.

1 Overview of the operating interface

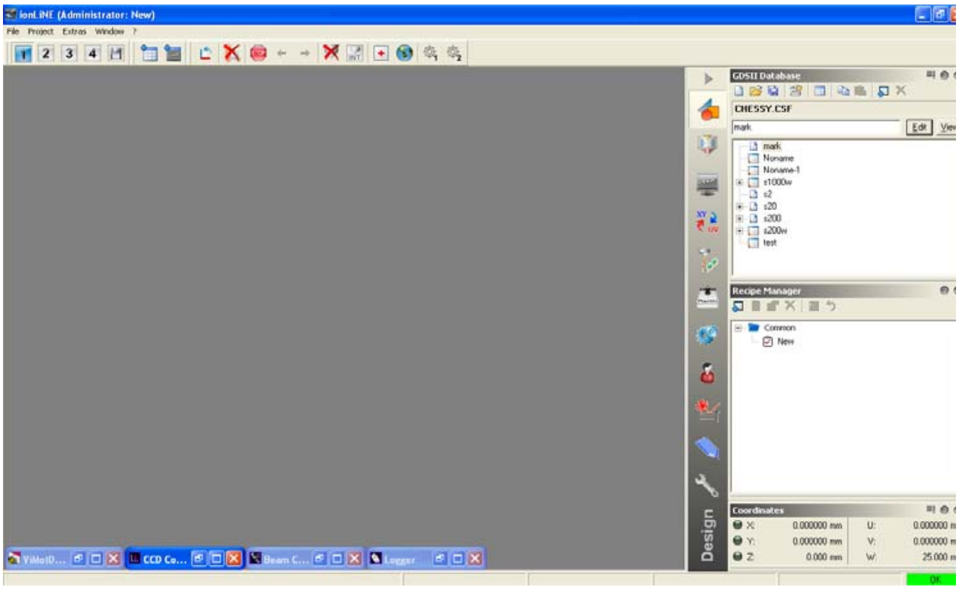


Figure 4-2: Main window of the Raith NANOSUITE software.

Main window item	Function
Title	The title bar shows the name of the software and in brackets the currently logged in user. The currently loaded project is also displayed.
Main menu	The menus available depend on the active window. Standard Windows terminology is used. Please note that for all control settings, the menus are shown in the corresponding section of the control bar.
Tool bar	The tool bar contains buttons with some frequently used menu commands.

Main window item	Function
Desktop area	Within the Desktop area the various documents of the Raith NANOSUITE software are displayed. – Document windows - normally, several documents of the same kind, such as Positionlists or Script Editors can be opened. – UV windows - a subset of documents with a sample position for each position within the window. These windows are displayed as opened or minimized. To open a window, double click on the corresponding icon. To close, minimize, increase, etc. the standard Windows controls are available. Up to four different Desktop areas build one Project. In detail, each Project contains, for each Desktop, information about: – the position of each window – the documents loaded – the behavior when opening another document – the overlay displayed in UV-documents.

1.1 Control bar

Control element	Function
	To open **GDSII Database**, **Recipe Manager**, **Coordinates**
	To open **Column Control**, **Fine Tune**, **Focus Wobbling** and **Signal Mixer**
	To open **Writefield Manager, Scan Manager, Writefield Alignment, Beam Tracking Alignment, Beam Tracking Control**
	To open **Adjust UVW (Global), Stage Control, Scan Manager**
	To open **Patterning Parameters, Beam Current**
	To open **Touching Alarm, Laser Stage Control, Stage Control, Find Home Position**

Control element	Function
	To open **Automation, Scan Manager, Import Position-list Data**
	To open **User Management**
	To open **Height Control**
	To open **Nanomanipulator, Nanomanipulator Control, Nanomanipulator Macro Editor, Nanomanipulator Coordinates**
	To open **Motor Control, Initialize Coordinate System, Limits**
	To minimize or restore the complete control bar
Parameter section subset:	
	On the various buttons of the control bar the module windows are grouped into sections. Choose one of these two buttons to minimize and restore the corresponding module inside the section.
	To open a menu with additional functions and settings. Please note that for all document windows, the menus are shown in the menu bar.
	To start the online help function for the selected module window
Status bar	The status bar summarizes a set of information in different status fields.

The left status field displays online help. Additionally, system messages, e.g. **Motors stopped**, **Emergency Stop**, etc., will appear.

While moving the cursor inside a UV window, such as an image, the stage XY coordinates, the UV sample coordinates, and the window coordinates are displayed in the following status fields.

On the right, a visual indication of program status is given. Double click it to open the **Status Report** dialog or use the right mouse button to obtain a dropdown menu with additional commands. The following statuses are possible:

Control bar - Module Status	Function
OK	No occurrences of errors, warnings or information
INFO	Information, no influence on running of the program
WARNING	Error detected
ERROR	Error, please check before continuing
FATAL	Immediate interaction is required

1.1.1 Status Report

By double clicking into the status field or by selecting the main menu **Extras➡Module status**, a list of all currently loaded modules, their error status and last messages is displayed.

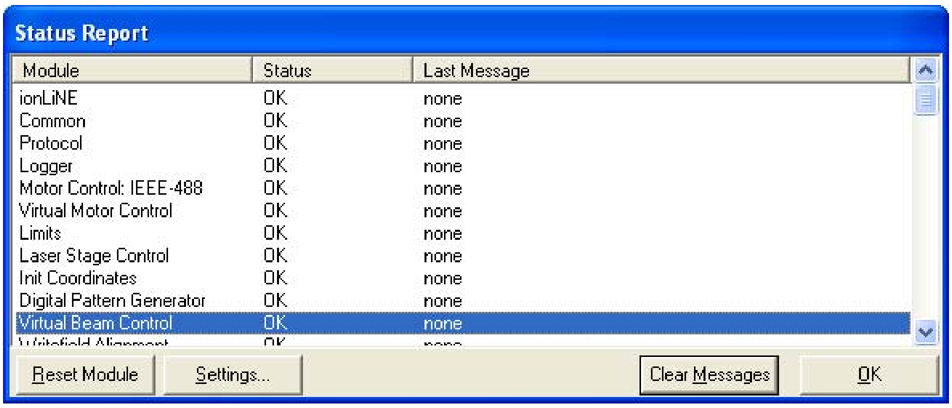


Figure 4-3: **Status Report** dialog

This dialog comprises two applications: editing of variables and display of status messages. There are three functions available.

Control element	Function
Reset Module	To reset the module, select the module name first

Control element	Function
Settings	To modify or to inspect module variables, select the module name. Select the variable name and choose the **Settings** button or double click on the variable name. A list of values will be displayed, if available. Alternatively, enter the value manually. To make the changes valid, reset the corresponding module.
Clear Messages	To clear all messages without resetting the modules

2 Basic operation

The basic operation of the interface will be explained in the following sections.

2.1 Starting the software

First, a Login dialog will open.

2.1.1 Login

The **Login** dialog will open automatically upon starting the software.

Figure 4-4: Login Screen of the Raith program. Example - Login for RAITH150^{TWO}.

Control element	Function
User	To select a user from the dropdown list
Password	To enter the password, specific to the user selected
Project	To enter a project name to save the application-specific parameters
Options subset:	
Change password	To edit the password
Reset user configura-tion	To revert to the default user configuration
Cancel	To cancel the login procedure
Login	To initiate and complete the login procedure

2.1.2 Administration

You may call up the **Administrator** by clicking on the icon in the con-trol bar.

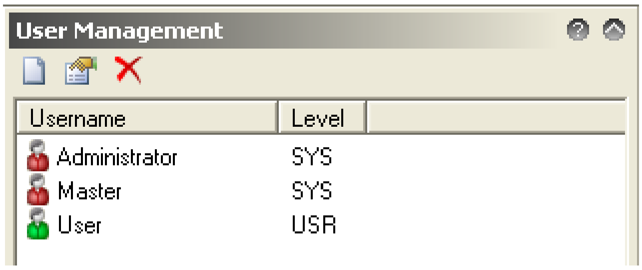


Figure 4-5: User Management

In addition to creating new users, the Administrator may delete users or modify their passwords, names, and levels.

Tool bar

Control element	Function
	To **Create a new user**. After choosing **User properties** or **Create a new user** the **User Properties** dialog will be opened to enter user-specific attributes.
	To change the attributes of a selected user
	To **delete a selected user**. After applying, choose if all files and directories should be deleted too

2.1.3 User management subdialogs

The **User Management** contains the following subdialogs:

Create new user

You may call up the dialog by clicking  in the **User Management** tool bar.

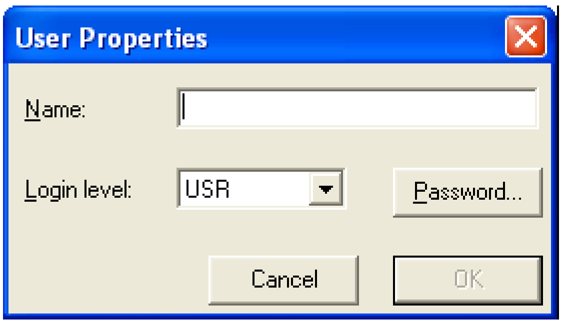


Figure 4-6: **User Properties** dialog

Edit user properties

You may call up the dialog by clicking  in the **User Management** tool bar.

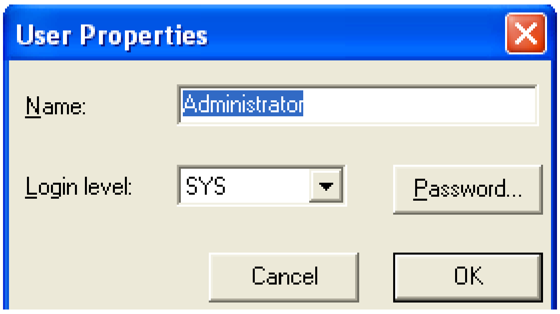


Figure 4-7: Dialog **User Properties** to create or modify user attributes

To create or modify a user, enter the user's **Name**, select the **Login level** and define a password by choosing the **Password** button. This will create or rename a user account named according to the user's name. It is also possible to have no password specified.

2.2 UV windows

UV windows are special windows in which each point can be identified by coordinates. Such windows include **Images**, **Wafermaps**, **Chipmaps**, **GDSII Viewer** and **GDSII Editor**. If the cursor is placed within a UV window, the status bar indicates the corresponding coordinates.

2.2.1 Main menu

The main menu consists of the following menu items.

File menu

File - menu item	**Function**
Close	To close the currently selected UV window
Close all	To close all UV windows currently open

Edit menu

The following Edit menu items are available:

Edit - menu item	**Function**
Adjustment	Choose this command to place the marks used for **3-Point Adjustment**. To make the marks visible in the current UV window, choose **Options**➡**UV display ...** ➡**UV Display Options**➡**Show adjustment marker** while the UV window is open.
Set mark 1, 2, 3	Marks 1, 2 and 3 can be set.
Measure	To measure coordinates, distances, angles and areas, defined by two points within the UV window area. After choosing this command, set the points using two subsequent mouse clicks. The results are displayed in a **Measure** dialog.

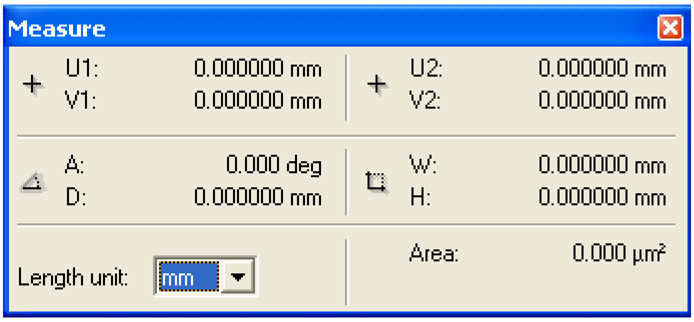


Figure 4-8: **Measure** dialog

Control element	Function
U1, V1 and U2, V2	The U and V coordinates for the first (U1, V1) and second (U2, V2) target measurement points are displayed.
A, D	**A** defines the angle and **D** defines the diagonal of the drawn rectangle.
W, H	**W** defines the width and **H** defines the height of the drawn rectangle.
Length unit	The length unit can be chosen from the dropdown list.
Area	The area information will be displayed.

View menu

The **View** menu items have the following functions:

View menu item	Function
Redraw	To redraw the viewed area. The command can be canceled by pressing the **Esc** key.
Zoom	A cascading menu opens with the following commands:
All	To display the magnified area covered by all of the structure elements
In 2x	To magnify the displayed area by a factor of 2 at the cursor position. Since the new magnified area is twice as large as the former area, the corresponding width and height are magnified by a factor of square root of 2.
Out 2x	To reduce the displayed area by a factor of 2 at the cursor position. Since the size of the reduced area is half as large as the former area, corresponding width and height are reduced by factor of square root of 2.

View menu item	Function
Last	To restore the previous zoom status
Writefield	To show the UV window content of the current writefield size at the current stage position
Zoom in tool	To magnify any portion of the viewed area. There are two possible implementations. After choosing the command, point to any location and press the left mouse button. This will zoom in at this location. Alternatively, point to any corner of the area to be magnified and click the left mouse button. While pressing the mouse button, point to the opposite corner of that area and release the mouse button.
Zoom out tool	To increase the viewed area by pointing to any location and pressing the left mouse button.
Hand	To move the viewed area

It is also possible to zoom in and out using the mouse wheel.

View menu item	Function
Overlays	To display the overlay elements such as **Positionlists**, etc.
Layers	To select the layers. (⇒ *Working with Layers* on page 5-67)
Toolbox	To display the **Tools** add-on window, which will stay on top of all active windows until it is closed. It provides buttons, which allow the user to have permanent and fast access to the most commonly used commands.

Options menu

The **Options** menu items have the following functions:

Options menu item	Function
Fill	To fill the structure
Cursor grid	To select the size of the cursor grid

Options menu item	Function
UV display...	To select items to be overlaid onto the currently displayed UV window. An additional dialog window opens, with the options described below.

UV Display Options subdialog

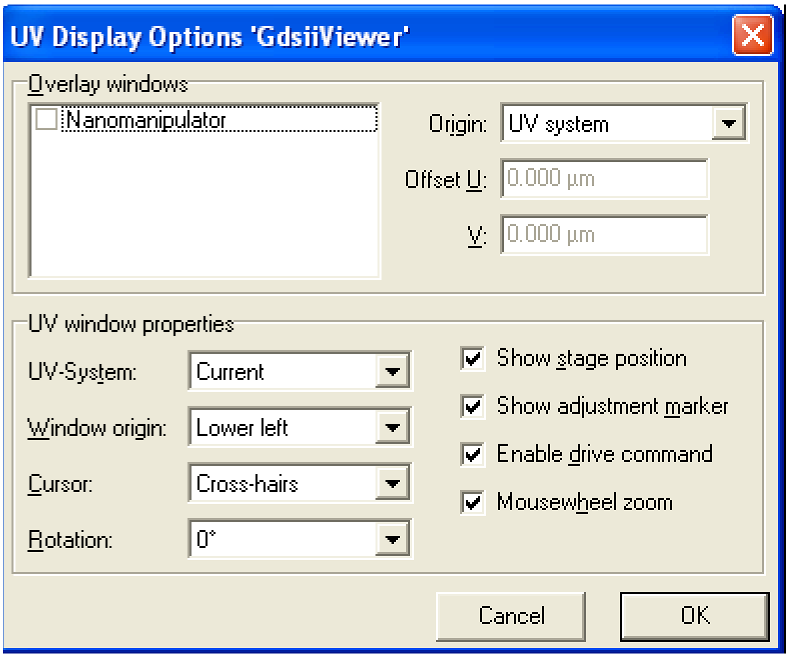


Figure 4-9: **UV Display Options** subdialog.

The list box shows all available overlay windows. Select those to be overlaid.

Control element	Function
Origin	Select the Origin or insertion point of the overlay window within one of the following coordinate systems.
UV System	Origin of the sample coordinate system
Window	Origin of the window
Window + offset	Origin of the window with definable offset
Chip center	Chip center at which the stage is located
Chip lower left	Lower left corner of the chip at which the stage is located

Control element	Function
Chip center + offset	Chip center at which the stage is located with selectable offset.
Offset U, V	If one of the coordinate systems **... + Offset** is selected, enter in fields **Offset U** and **Offset V** the offset values for U and V axes.
U, V System	The UV system can be selected from the dropdown list. the options are **Current**, **Global** or **Local**.
Window origin	**Window origin** selects the origin of the window coordinate system. The window coordinates are displayed within the status bar of the program desktop. The window origin can be selected from the dropdown list, to either **Center** or **Lower Left**.
Cursor	Select the **Cursor** type. The cursor can be set to either **Default** or **Crosshairs**.
Rotation	The rotation can be entered to **0**, **90**, **180** or **270** degrees.
Show stage position	To show the cross-shaped marker graphically, indicating the current location of the stage
Show adjustment marker	To show the adjustment flags
Enable drive command	To enable stage movement using the mouse command **Ctrl + right mouse button**. When unchecked, it will be disabled. For example, when very accurate positioning is required, disabling the command will prevent any accidental movement.
Mousewheel zoom	To enable the **Mousewheel zoom**

All of these checkmark options are only valid for the currently opened UV window. They will not change default settings for the next UV window to be opened.

2.2.2 Using the Toolbox

For a fast access to the frequently used commands it is possible to open an add-on window by choosing **View➡Toolbox**. The toolbox is adjusted dynamically, depending on the windows opened.

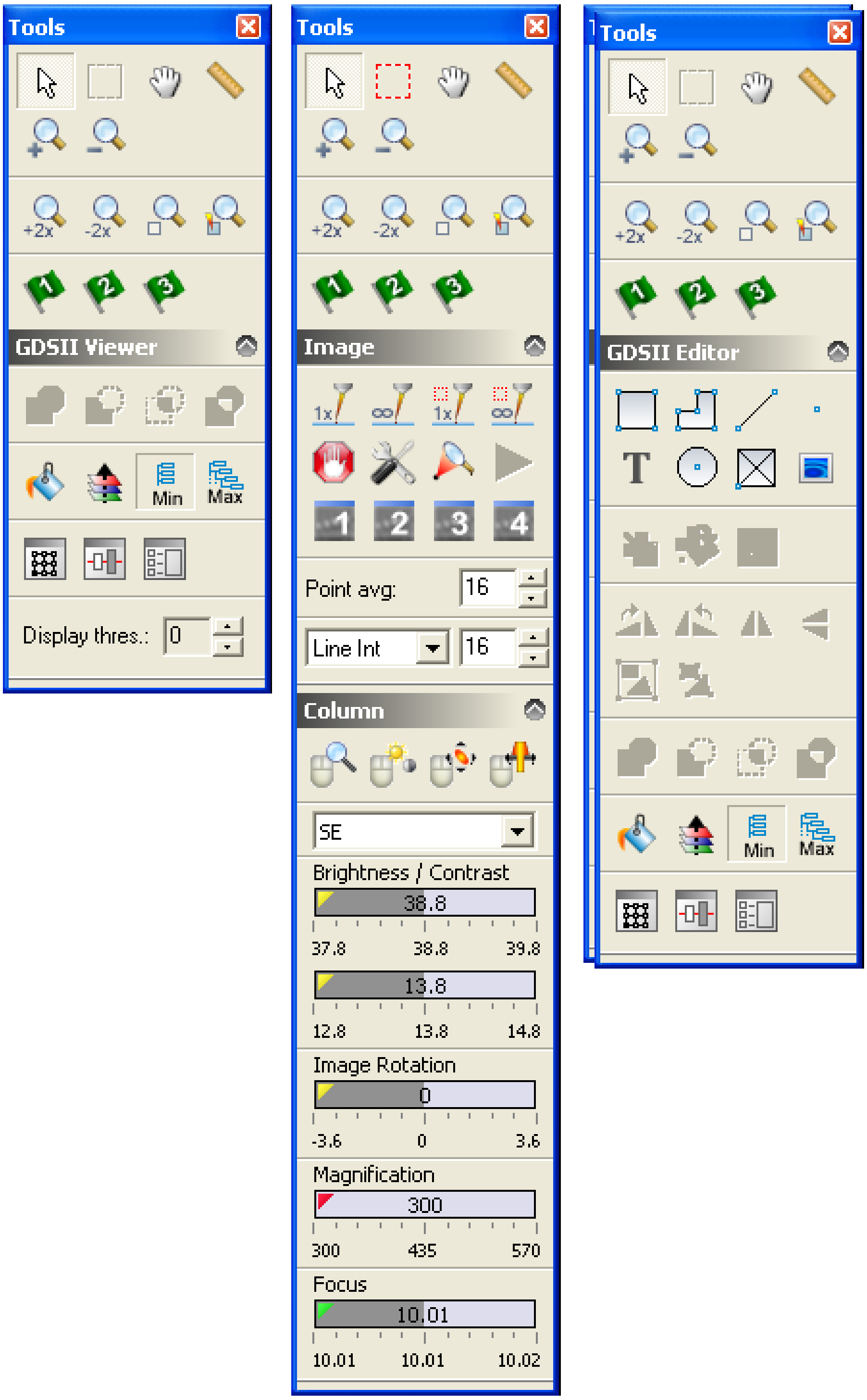


Figure 4-10: Tool dialog

Tools subset

Control element	Function
	To select normal cursor
	To select **Frame selection** mode
	To activate the Hand tool to move the viewed area. Same command as menu **View➡Zoom➡Hand tool**
	To measure a distance. Same command as menu **Edit➡Measure**.
	To zoom into the image. Same command as menu **View➡Zoom➡Zoom in tool**
	To zoom out of the image. Same command as menu **View➡Zoom➡Zoom out tool**
	To magnify the displayed area by a factor of 2 at the cursor position. Same command as menu **View➡Zoom➡In 2x**.
	To reduce the displayed area by a factor of 2 at the cursor position. Same command as menu **View➡Zoom➡Out 2x**.
	To display the magnified area covered by all of the structure elements. Same command as menu **View➡Zoom➡All**.
	To show the UV window content of the current writefield size at the current stage position. Same command as menu **View➡Zoom➡Writefield**.
	To set the adjustment marks. Same command as menu **Edit➡Adjustment➡Set mark 1, 2, 3**

The **Image** toolbox is explained in detail in the **Linescans and Imagescans** chapter, (⇒ *Image tool bar* on page 10-32).

The **Column** toolbox is explained in detail in the **Linescans and Images** chapter, (⇒ *Column subset* on page 10-36).

The **GDSII Viewer** and **GDSII Editor** toolboxes are explained in detail in the **GDSII Editor** chapter, (⇒ *Toolbox* on page 5-135).

2.3 Operating the software

The software menu items and tool bar commands will be explained, as well as the various mouse functions and the drag and drop concept.

2.3.1 Main menu

The main menu consists of the following menu items:

File menu

File menu item	**Function**
Close, Close all	To close the active document. Choose **Close all** to close all document windows of the same type.
New image, Open image...	Generate a new image or open an existing one. A new image is named NONAME#.SSC, # indicating consecutive numbering.
Open linescan...	Opens an existing Linescan
New Positionlist, Open Positionlist...	Generate a new Positionlist or open an existing one. A new Positionlist is named NONAME#.PLS, # indicating consecutive numbering.
New Wafermap, Open Wafermap...	Generate a new Wafermap or open an existing one. A new Wafermap is named NONAME#.WLO, # indicating con-secutive numbering.
New Script, Open Script...	Generate a new Script or open an existing one. A new Script is named NONAME#.JS, # indicating consecutive numbering.
Open navigator	To open an existing navigator (.NAV) file
Exit	To exit the program session

Project menu

Project menu item	**Function**
New	To load the Project file NEW.DSK
Open	To open an existing Project file
Save	To save changes to a loaded Project. The Project will be saved under the same file name.

Project menu item	Function
Save as...	To save a loaded Project under a freely selectable file name
Desktop#	To activate one of the four available desktop configurations. # indicates the number of the desktop.
Save desktop	To save a desktop configuration. After choosing, select the number of the related desktop. In addition, enter a suitable name for the desktop configuration.

Extras menu

Extras menu item	Function
Module status	To obtain an overview of the status of each module (⇒ *Control bar - Module Status* on page 4-14).
Clear messages	To clear any displayed messages
Settings	To set and display the various parameters of your system or software

Window menu

Window menu item	Function
Arrange icons	All Windows icons are arranged alphabetically. The opened windows are not influenced.
Close all	All currently open windows will be closed.
Save settings ...	Opens a dialog to set the behavior of documents when opening a document of the same type, such as an image.
Control bar	The listing contains all available control bar windows in alphabetical order. Below is a list of all activated windows, indicated by a checkmark.

Help menu (?)

Help menu item	Function
Index	Opens the online help at the main topics
Module	After activating the corresponding window, this command opens a context-sensitive online help for this module.
About	In addition to the standard version information, the currently logged in user and the used Project are displayed.

About

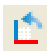
The currently installed software version and product name can be viewed in the main menu **Help➡About**.

Figure 4-11: **About** screen.

Control element	Function
Version number	The currently installed version number is displayed in the upper right corner of the dialog.
User, Login Level	The user and login level are displayed.
Project	The currently selected project is displayed.

2.3.2 Tool bar

The displayed buttons depend on the active window. Generally available buttons are outlined below.

Control element	Function
	To recall Project-specific desktop configurations. Up to four different configurations are possible. The active desktop is highlighted.
	To save the desktop configuration. After choosing, select the number of the related desktop. In addition, enter a suitable name for the desktop configuration.
	To open an empty Positionlist window
	To open a new image window
	To rotate the presentation of the active UV-document by 90° counterclockwise, each time the button is pressed.
	To activate or deactivate beam tracking
	To stop all motors
	To execute the previous drive command in the history of stage movement
	To execute next drive command in the history of stage movement.
	To toggle beam **On/Off**
	To reset the UV coordinate system to XY
	To toggle between global and local adjustment
	Buttons for quick access navigators
	Buttons for quick access scripts

2.3.3 Mouse and keyboard commands

Generally, all commands are accessible using the **Alt** key, together with the key corresponding to the underlined character. See Windows documentation for guidance concerning changes to this behavior. The following shortcuts are available throughout the software:

Mouse commands	Function
none + right	Opens a menu with frequently-used commands
Ctrl + right	Drive to location
Ctrl + left	Move the selected object to a new position
Shift + left	Select elements within an area defined by subsequent mouse clicks.

Keyboard commands	Function
F1	Call up help. Specific help will be called up for the active window.
Ctrl + 1... 4	Recall desktop configuration 1…4
Ctrl + 5	Save desktop configuration
Alt + x	Exit the program session
+	Zoom in by a factor of 2
-	Zoom out by a factor of 2
=	View all
e	Measure dimensions in editor or viewer
h	Move viewed area (hand tool)
t	Open toolbox, toggle mode
z	Zoom in and out, toggle mode
Shift + arrow	Scroll the display in the direction of the arrow
Alt + o	Show overlays, toggle mode
Alt + r	Redraw the display

2.3.4 Drag and drop concepts

The concept of drag and drop operation is extensively used throughout the Raith NANOSUITE software to ensure easy operation of rather complex tasks. This means that some objects can be picked up using the mouse and placed or dropped in another location on the screen. In some areas, where dropping of the object makes no sense, this will be indicated by a no-parking sign at the mouse cursor. After reaching a possible target, a realistic representation of the selected object appears. Dropping is achieved by releasing the left mouse button. All required setups are handled by the system.

2.4 Module settings

Module settings can be called up via the **Extras** menu, which is always available via the menu bar, it is not module-specific. (⇒ *Extras menu* on page 4-27)

The **Module Status** as well as the **Module Settings** can be displayed.

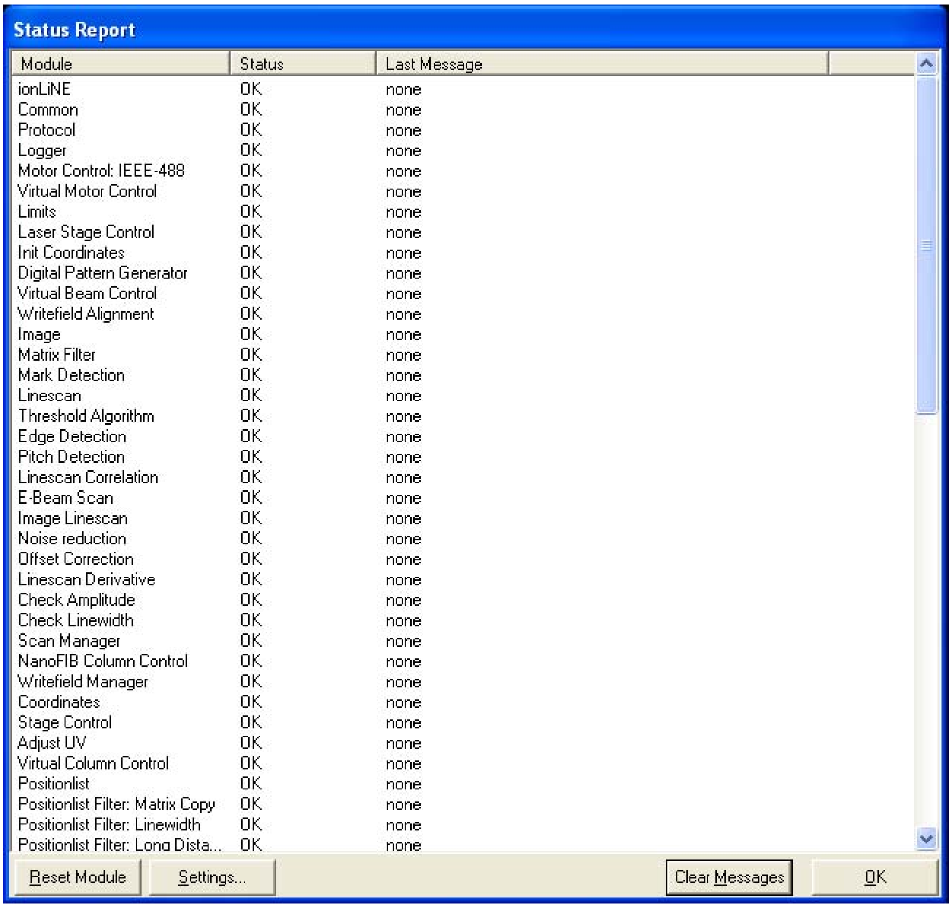


Figure 4-12: **Status Report** window

Control element	Function
Module, Status, Last Message	This will display a list consisting of three columns: - 1st column: Module name - 2nd column: Status such as OK, Info, Error, Warning, Fatal Error - 3rd column: Message, giving a description of the info or error message shown.
Reset Module	All settings for this module will be reset, which will result in re-initialization of all module settings. The parameter set stored in the database will be read in again and initialized. At the same time, any warning or error messages will be reset.
Settings	This will initiate the same command as via the main menu **Extras**➡**Settings**.
Clear Messages	To clear any messages, e.g. Info, Warning or Fatal Error messages in the **Status** bar. All types of messages, except the **OK** message, will be cleared and the OK message will be displayed again. (⇒ *Status Report* on page 4-14)
OK	This will close the dialog without saving.

Whenever a module is loaded, the parameter set from the database will be read in and initialized. These are the default settings. **Reset Module** will reset the default settings.

Click **Clear Message** to clear messages and error warnings for all modules. Clicking on **Reset Module** will only clear messages for the selected module.

2.4.1 Settings subdialog

You can call up the **Setting** subdialog via the menu **Extras➡Settings**.

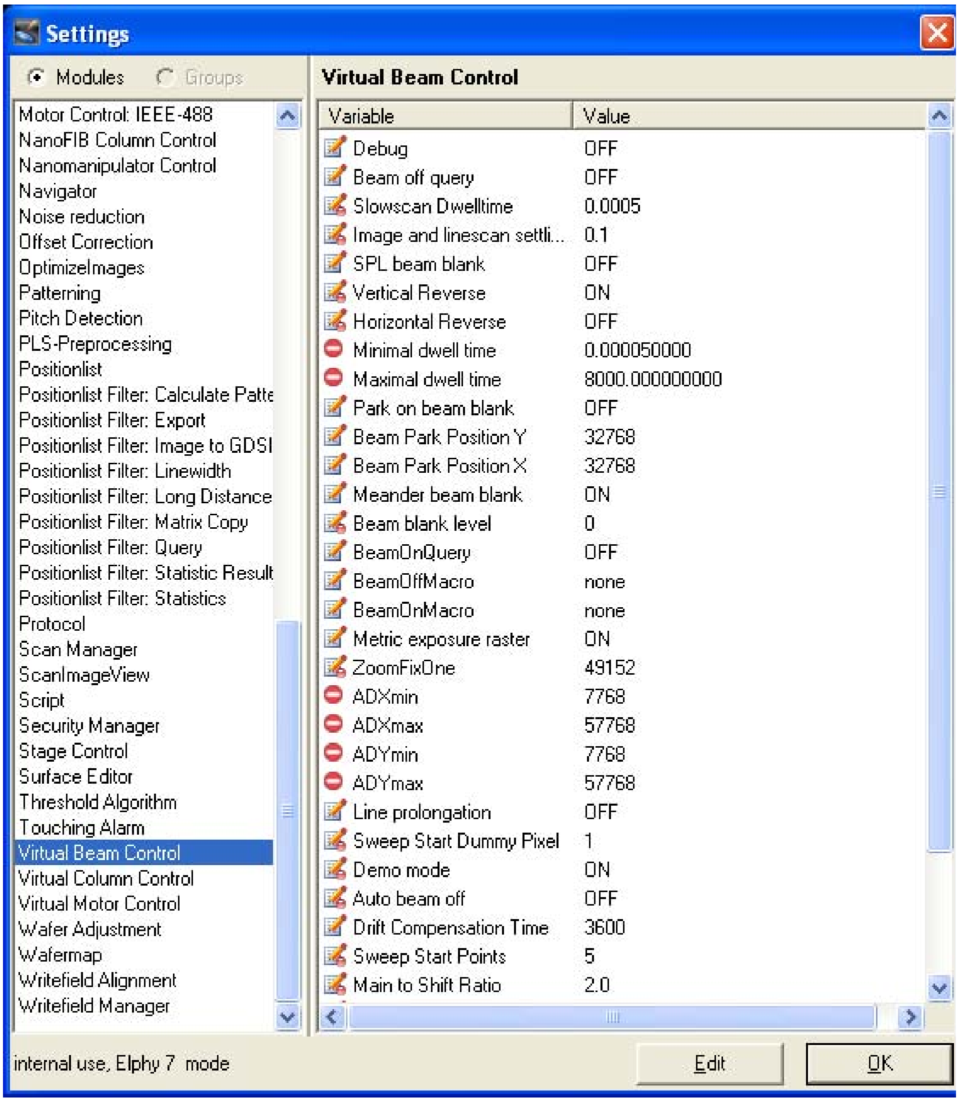


Figure 4-13: **Settings** window

Control element	Function
Module listing	A list of modules will be displayed. A module can be selected. For each selected module, the database variables, which can be edited by the user, will be displayed.

Control element	Function
Database variables	The editable variables displayed will depend on the user login level. Any SYS level user will have greater access to the editable variables than a USR or EXP level user.
Description of variables	When a variable is selected, a description of this variable is displayed at the bottom of this dialog. On the right hand side, the value of the selected variable is displayed.

Double clicking on the value will open the **Edit** dialog, in which the value can be edited. The Edit dialog will change according to the type of variable selected. In some instances, a dropdown list is displayed. In some cases, the user can type in a new value. Click on **OK** to confirm the new value.

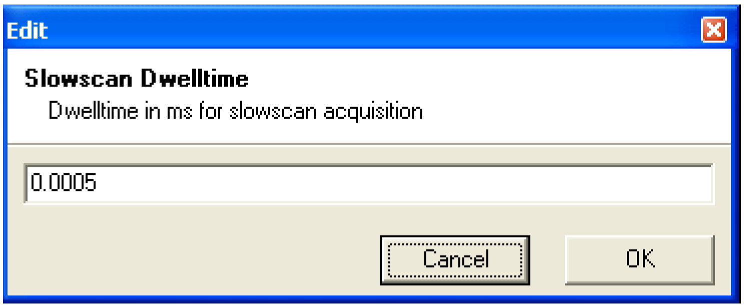


Figure 4-14: **Edit** dialog

The symbols of the variables are different, depending on the type of variable.

Variable symbols	Description
	Editable variable
	Editable variable, but when changed, the new value will be used for all users. These variables are not user-specific.
	Read-only variable. Variable is only visible and cannot be edited

Once a variable has been edited, click on **OK** to confirm. Then go to **Module Status** and click on the **Reset Module** button to initiate the new value. The selected module is then initialized.

2.5 Logger

Logger is used to document messages for debugging purposes.

2.5.1 Logger settings

You may call up the **Logger settings** via the main menu **Extras➡Settings➡Logger**.

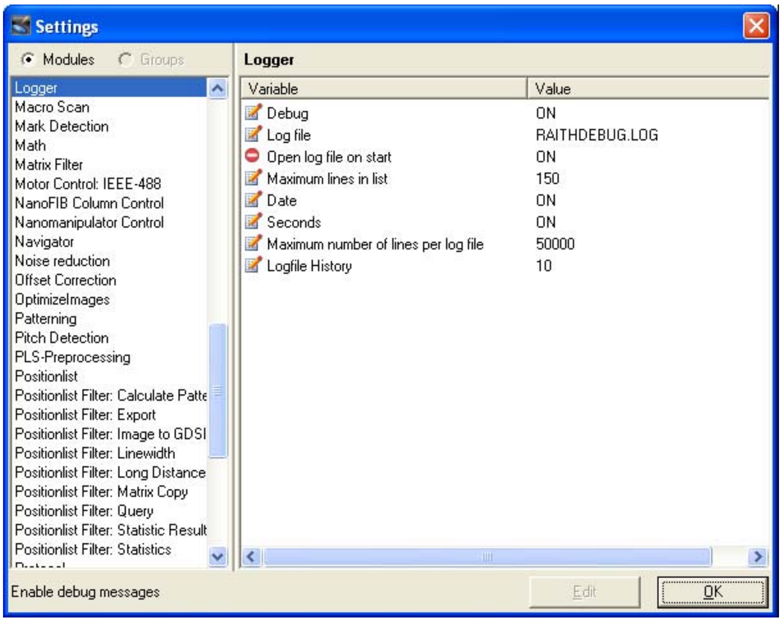


Figure 4-15: **Logger** settings

Control element	Function
Log file	The default **Log File** name is displayed. The user can enter a new file name to log the data.
Open log file on start	To activate the logger immediately when the software is started. This is important if the user wishes to track any problems in the start-up phase of the software.
Maximum lines in list	To determine how many lines will be displayed on the monitor. For example, if set to 150, then the maximum number of lines in the list displayed on the monitor will be 150.
Date	The date can be displayed. The user can toggle between **Date On/Off**.

Control element	Function
Seconds On/Off	The output of the data can be displayed with a resolution of 1 second, if the user wishes. The user can toggle between **Seconds On/Off**.
Max number of lines per Log file	The maximum number of lines in the **Log file** can be set to a specific number, e.g. to 50,000 lines. If the logger runs for a longer period of time, a new log file will be created when the 50,000 line limit has been reached.
Log file history	Set the **Log file history** to a specific number. If, for example, it is set to 10, then 10 log files will be generated with Index numbers. Once 10 log files have been created, the series will be restarted and files will be overwritten. The 11th log file will be saved under the file with index number 1, which will overwrite the old log file.

2.5.2 Logger window

You may call up **Logger** by the menu item **Window➡Logger.**

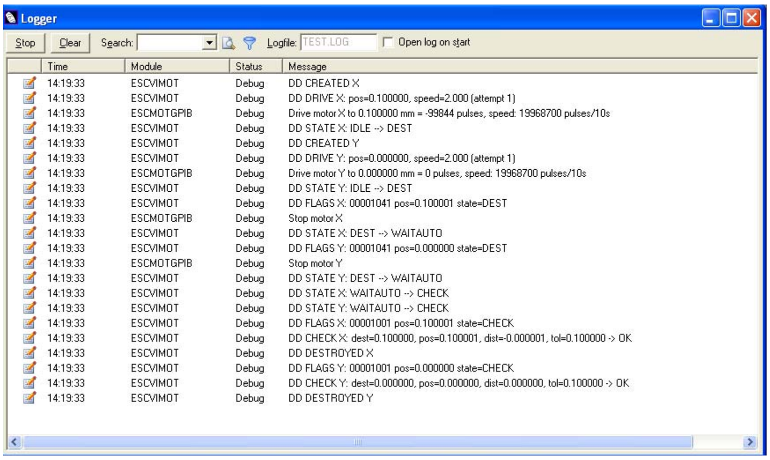


Figure 4-16: **Logger** window

Control element	Function
Start	To initiate the logger, click the **Start** button.
Stop	To end the logger, click the **Stop** button.
Clear	To delete the messages shown in the list. This will not delete the log file.

Control element	Function
Search	The user can **Search** for a certain type of error or for keywords. The occurrence of words found through the search facility will be highlighted.
	Enter the keyword for the search, then select the **Search for keywords beside** to show the keywords found in the search.
	To show only the keywords found in the search. All other items will not be shown.
Logfile	The user can either use the default **Logfile** or enter a different file name.
Open log on start	To activate the logger immediately when the software is started. This is important if the user wishes to track any problems in the start-up phase of the software.

The logger should only be initiated when required. Logging the Logger data can take up valuable processing time, which might slow down other processes, e.g. Patterning, which may be time critical.

2.5.3 Logger Log folder

The logger data file will be saved in the Raith directory **<Raith root>** folder.

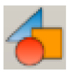
The user can open an existing log file in any text editor program. When the log file is opened, the timestamp and type of module are displayed. The outcome debug or error is also shown together with a description.

Chapter 5
GDSII Data Handling (Design Module)

This chapter describes how to create and edit layout designs as a template for the patterning via the GDSII module of the NANOSUITE software.

This chapter contains the following information:

- Functional description and user interface of the GDSII module (⇒ *Functional Description* on page 5-2).
- Creating and modifying of designs consisting of one or more structures (⇒ *Creating structure elements* on page 5-28).
- Setting up hierarchical designs (⇒ *Creating hierarchical GDSII Databases* on page 5-62).
- Defining working areas to pattern an extract of a design only (⇒ *Using Working areas* on page 5-70).
- Importing structures created in external programs – e.g. CAD files (⇒ *Importing and exporting data* on page 5-8).
- Importing Bitmaps (⇒ *Adding bitmaps* on page 5-37).
- Defining patterning recipes for special patterning strategies – e.g. for special kinds of resist (⇒ *Working with patterning recipes* on page 5-72).
- Performing mark scans (⇒ *Adding Mark Scans* on page 5-65).
- Complete description of the dialogs, menus and control elements of the module (⇒ *Software Reference* on page 5-77).
- Available mouse and keyboard commands (⇒ *Mouse and keyboard commands* on page 5-150).

You call up the GDSII module via the  button in the control bar. The corresponding dialogs will be displayed on the right hand side of the control bar. On the control bar **Design** will be displayed.

A Functional Description

The GDSII module is a graphical software based on the Calma GDSII standard for electronic design automation and is optimized for the use in Raith lithography systems. Designs are created graphically via a graphic editor and stored in a separate database called GDSII database.

The GDSII module is made up of the following components:

- GDSII database to manage the designs and structures (⇒ *GDSII Database* on page 5-2).
- GDSII editor and GDSII viewer to create and inspect designs (⇒ *GDSII Editor and Viewer* on page 5-5).
- Recipe manager to define and manage patterning recipes for special patterning strategies (⇒ *GDSII Recipe Manager* on page 5-6).

1 GDSII Database

The GDSII database is a file, which contains all structures of a design.

Each structure consists of one or more structure elements. The structures can be set up hierarchically in order to minimize the file size – i.e. a main structure can contain numerous referenced structures. Therefore, a structure only need to be created one time and can be used multiple times in different main structures. Also, it is possible to create matrices of structures, which will be repeated numerous times in a main structure.

The control takes place via the **GDSII Database** dialog:

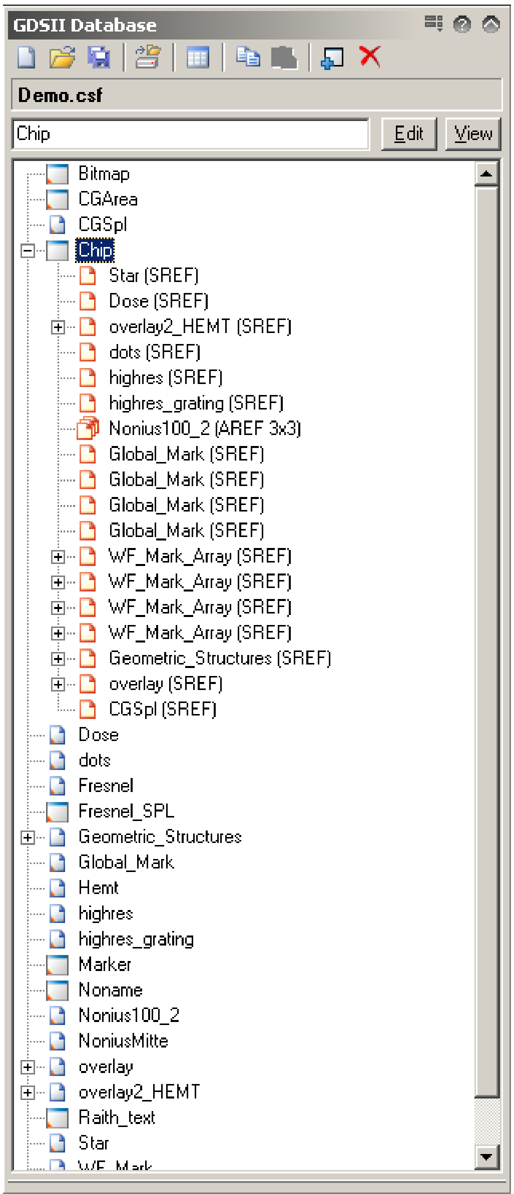


Figure 5-1: **GDSII Database** dialog

The dialog is divided into the following subsets:

- Tool bar with commands and fields to set up, edit and view databases and single structures. The **Edit** and **View** buttons toggle between the GDSII Editor and Viewer.
- **GDSII Database** tree view to display the structures and their hierarchical order.

For a detailed description of all control elements of the dialog refer to (⇒ *GDSII Database dialog* on page 5-77).

1.1 GDSII file types

A whole GDSII dataset can consist of up to five different file types, which are stored in the GDSII User directory. Only one file type contains the design and is provided in the GDSII Database tree view for editing. The other four file types contain control commands, which are needed for editing a design in any other GDSII program or on another PC platform – e.g. to keep the hierarchy or layer specifications. A GDSII dataset can consist of the following file types:

Table 5-1: GDSII module file types

File type	Function
*.CSF	GDSII database file, which contains the design.
*.hir	Text file with commands for streamed loading of structures of a GDSII database. The function is especially useful for very voluminous database files.
*.wor	Text file to keep working area information.
*.lay	Text file to keep layer information.
*.ann	Text file to keep annotations made to a design.

The GDSII module of the NANOSUITE software offers some functions, which are not supported by GDSII or graphic software of other vendors. Therefore, to keep the whole Raith specific functionality of a design, all files types that belong to the GDSII design file (files with the name of GDSII database but different file extensions) have to be saved together, if a design may be edited externally.

2 GDSII Editor and Viewer

The GDSII Editor and Viewer are UV windows with GDSII specific functionalities. The editor serves to create and modify structures, the viewer offers a fast access to a design for inspecting purposes.

The GDSII editor and viewer cannot be opened at the same time, i.e. when opening the editor the viewer is closed automatically and vice versa. But it is possible to open more than one viewer window at the same time.

The control takes place via the main menu bar of the NANOSUITE software and the **GDSII Editor** and **GDSII Viewer** dialogs. The design will be displayed in an UV coordinate systems:

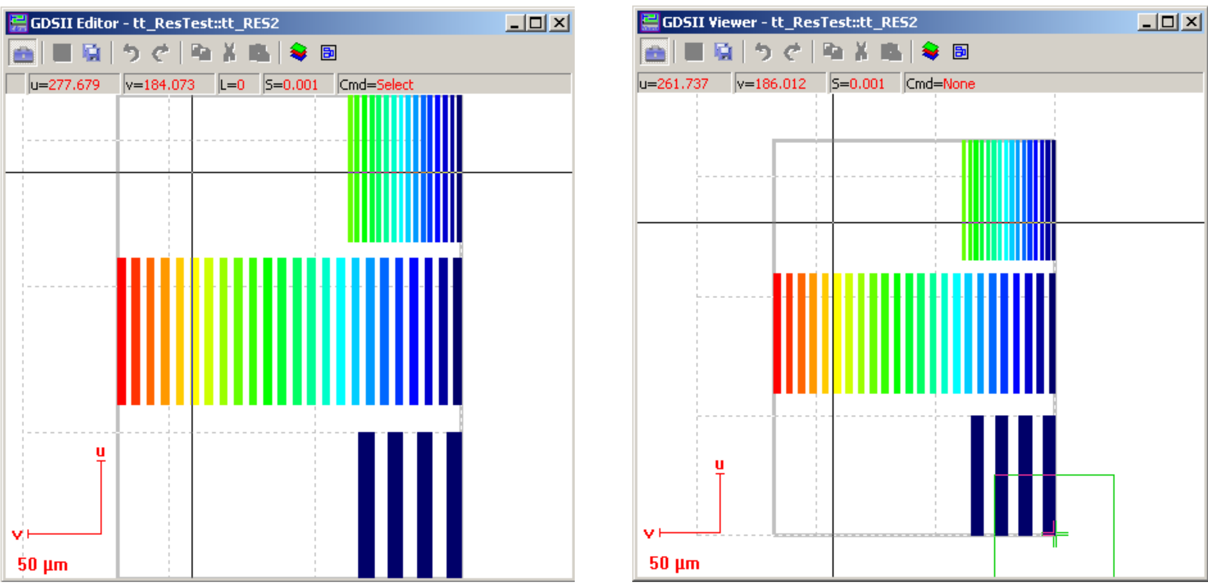


Figure 5-2: **GDSII Editor** and **Viewer** dialogs

The **Editor** and **Viewer** dialogs are divided into three subsets:

- Tool bar with commands for saving, basic handling of structure elements, and add-on window call up.
- Status bar with information about the current UV position of the cursor, active layer, grid, and currently active command.
- UV window in which the design is displayed.

For a detailed description of all control elements of the dialog refer to (⇒ *GDSII Editor and Viewer dialogs* on page 5-89).

---

3 GDSII Recipe Manager

The Recipe Manager offers the possibility to set up so called patterning recipes for special patterning strategies.

Dependent on the resist being used and the aim of patterning, the parameters affecting the patterning can be vary highly. Therefore, it is helpful to set up datasets with experienced values and store them as a recipe. The patterning recipes can be assigned to nearly any structure element.

The control takes place via the **Recipe Manager** dialog:

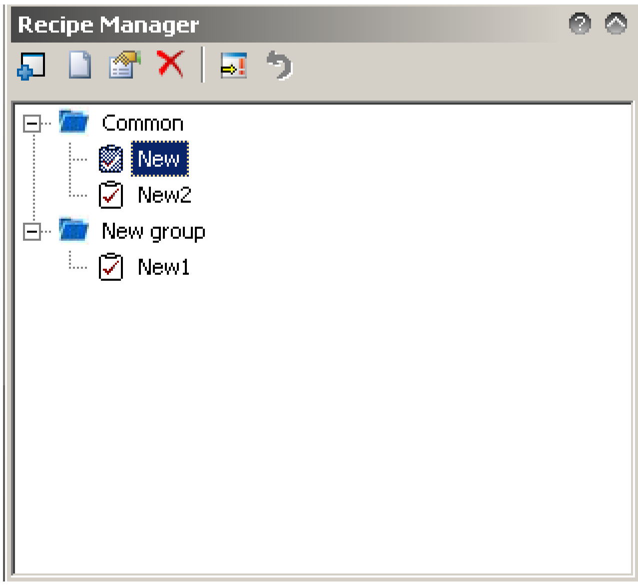


Figure 5-3: **Recipe Manager** dialog

The Recipe Manager dialog is divided into two subsets:

- Tool bar
- **Recipe Manager** tree view to display single and groups of patterning recipes.

For a detailed description of all control elements of the dialog refer to (⇒ *Recipe Manager dialog (option)* on page 5-125).

B Tasks

This chapter describes the basic and most important tasks.

1 Opening a GDSII Database

▶ In the **GDSII Database** dialog choose .

A file select box will be opened showing all existing GDSII database files of the actual path.

▶ Select a GDSII database and open the file.

Within the **GDSII Database** tree view, the complete hierarchy and set of structures will be displayed. To display the GDSII in such a way, a build of the hierarchy file will be performed automatically if the database is opened for the first time.

Raith supports the GDSII™ Calma Stream Format. However, there are occasions when you cannot import a GDSII file. This is due to third party applications not conforming to the standard or storing the file in an internal binary format.
Check the documentation for the third party program first you have problems importing a GDSII file. The definition of the GDSII format can be found at numerous WWW sites.

---

2 Setting up a GDSII Database

▶ In the **GDSII Database** dialog choose .

A file select box will be opened.

▶ Enter a proper database name and save the file.

By default, the new GDSII database will be stored as *.CSF, *.GDS format in **User** directory ➡ **GDSII**. The name will be displayed in the tool bar of the dialog.

3 Adding and opening a Structure

- In the **GDSII Database** dialog choose .

  The **Create New Structure** dialog will be opened.

- Enter a proper structure name and confirm with **OK**.

  The structure will be displayed in the **GDSII Database** tree view. The name will be displayed in the tool bar of the dialog.

- Double click the structure in the tree view.

  The **Viewer** dialog will be opened and the selected structure will be displayed in the UV window.

- Click on **Edit** in the tool bar of the GDSII Database dialog.

  The **Viewer** dialog will be closed and the **GDSII Editor** dialog will be opened. The structure will be displayed in the UV window and you can create or edit structure elements (⇒ *Creating structure elements* on page 5-28).

4 Importing and exporting data

To enable handling of even extended GDSII files on a PC based data system, the hierarchical structure of the data is kept as long as possible, e.g. until final patterning. Externally created GDSII files can be handled as well as data files from other design systems. In particular, the Raith GDSII module offers the following filters:

- ASC import and export filter for reading and saving of ASCII-type data files,
- ELM import filter for import of data files from Raith PROXY and PROXY-WRITER,
- DXF import filter for import of structures from most common CAD programs, e.g. AutoCAD, DesignCAD, etc,
- CIF import filter for import of data files stored in Caltech Intermediate Format by California Institute of Technology.

4.1 Importing files in ASCII format

The import filters will be explained in the following sections.

4.1.1 Import filters and syntax

It is possible to import elements into a GDSII database using a file in ASCII format as a source. Thus these filters are ideal to import self-created structures, e.g. by means of calculation programs. The commands are **Load *.ELM** and **Load *.ASC** in the **GDSII Database** menu.

The main difference between the two commands **Load *.ELM** and **Load *.ASC** is the handling of negative signs. While the ELM import filter erases all negative signs, the ASC import filter leaves them as they are. The reason for this is the internal handling of negative signs by the PROXY-WRITER software within the ELM format. A negative sign characterizes within this software hidden lines. So, if structure elements are located in any but the first quadrant of the GDSII design coordinate system, the ASC import filter has to be used instead of the ELM import filter.

Any standard editor or program, which is able to save data as ASCII text, may be used. Within the currently opened GDSII database, there is a structure named **<filename>.asc** or **<filename>.elm** respectively after loading.

The file format for the two import filters is defined as follows. Each element starts with a character defining the type of GDSII element followed by several parameters separated by space.

The following table lists available GDSII structure elements and the corresponding character.

Table 5-2: List of available structure elements in ASCII format

GDSII element	Character	Name in GDSII programs from other vendors
box or polygon	1	BOUNDARY
arc	A	not available
circle	C	not available
ellipse	E	not available
FBMS path	F	not available
path	L	PATH
point	P	PATH, starting point = end point
text	T	not available

The syntax for the different elements and their parameters is as follows using the general rules that µm is the base unit for all length, width, etc. Degree is the base unit for all angles and the dose is given in percentage.

Table 5-3: Syntax of GDSII structure elements in ASCII format

GDSII element	Syntax	Comment
polygon	1 <dose> <layer>	BOUNDARY

Table 5-3: Syntax of GDSII structure elements in ASCII format

GDSII element	Syntax	Comment
	<u1> <v1>	coordinates 1st point
	<u2> <v2>	coordinates 2nd point
	...	
	<u1> <v1>	must be repeated
	#	end of element
arc	A <dose> <layer> [<width>]	not filled, width usage
	<u-center> <v-center>	center coordinates
	<radius> [<radius2>]	2nd radius for ellipse
	<vertices>	
	<rotation> <angle1> <angle2>	angles defining arc
	#	
circle	C <dose> <layer> [<width>]	not filled, width usage
	<u-center> <v-center>	center coordinates
	<radius>	
	<vertices>	
	<rotation>	
	#	
ellipse	E <dose> <layer> [<width>]	not filled, width usage
	<u-center> <v-center>	<u-center> <v-center>
	<radius> <radius2>	
	<vertices>	
	<rotation>	
	#	
FBMS path	F <dose> <layer> <width>	

Table 5-3: Syntax of GDSII structure elements in ASCII format

GDSII element	Syntax	Comment
	<node type>	node type (0 = start, 1 = line, 2 = arc)
	<node attribute>	node attribute (currently not evaluated, must be set to zero)
	<U><V>	U and V coordinates of the node point in µm
	<radius>	curvature radius in µm
	#	
path	L <dose> <layer> <width>	
	<u-center> <v-center>	coordinates 1st point
	<u-center> <v-center>	coordinates 2nd point
	...	
	#	
point	P <dose> <layer> <width>	with w/o meaning
	<u-center> <v-center>	coordinates point
	#	
text	T <dose> <layer> <width>	
	<u-center> <v-center>	coordinates starting point
	<height> <angle>	
	<u-align> <v-align>	alignment values 0 to 2
	<text>	text itself

The following example contains five elements. All elements are assigned with 100% dose in five different layers:

Example for ASCII Syntax:

```
1 100.0 0
-5.000 5.000
-5.000 -5.000
5.000 -5.000
5.000 5.000
-5.000 5.000
#
L 100.0 1 1.000
0.000 -7.000
3.000 -10.000
7.000 -10.000
10.000 -7.000
#
C 100.0 2 0.200
10.000 10.000
5.000
100
#
T 100.0 3 0.000
3.000 3.000
1.000 0.000
2 2
Example
#
F 100.0 4 0.000
0, 0, 0.000, 0.000, 500.000
#
F 100.0 4 0.000
0, 0, -818.817, -32.619, 0.000
1, 0, 811.113, -32.619, 0.000
1, 0, 811.113, 717.782, 0.000
1, 0, 998.713, 717.782, 0.000
1, 0, 998.713, -500.802, 0.000
2, 0, 900.713, -580.802, 20.000
1, 0, 50.000, -580.802, 5.000
2, 0, -0.198, -497.965, 20.000
#
```

4.1.2 Importing an ASCII file

- Open a GDSII database.
- In the **GDSII Database** dialog choose ➡ **File** ➡ **Load*.ASCII**.

  A file select box will be opened.
- Choose an ASCII file and is open

  The file will be imported and displayed in the **GDSII Database** tree view as a structure. The original filename and file extension are kept.

4.2 Saving files in ASCII format

The export filter and save options will be explained in the following sections.

4.2.1 Export filter

The ASCII export filter enables the exporting of a selected structure into an ASCII file format. Hierarchy will be made flat. Structure modification has to be done by means of the GDSII editor before exporting (⇒ *Preparing GDSII files for export* on page 5-17).

Import and export of ASCII files offer the capability to copy flat designs within one database or to transfer them into other databases.

4.2.2 Saving a structure as ASCII file:

- Open a GDSII database.
- Select a structure in the **GDSII Database** tree view.
- In the **GDSII Database** dialog choose ➡ **File** ➡ **Save*.ASC.**

  A file select box will be opened.
- Select a destination directory and save the file.

  The structure will be saved as ASCII file and stored in the **GDSII** directory.

4.3 Importing files in DXF format

It is possible to import structures created in other CAD files – e.g. by means of AutoCAD, DesignCAD, etc. – via the DXF interchange file format. The imported data are stored within structure **<filename>.DXF** in the current GDSII database.

When choosing this import filter several parameters can be set to control the import process. For example, if the CAD file contains several layer definitions, these may be assigned to different GDSII layers. In addition, some layer properties – e.g. the dose factor – can be modified for each DXF layer.

Import DXF files as follows:

▶ In the **GDSII Database** dialog choose ➡ **File** ➡ **Load*.DXF**.

A file select box will be opened.

▶ Select a DXF file and open it.

The **DXF Import Options** subdialog will be opened:

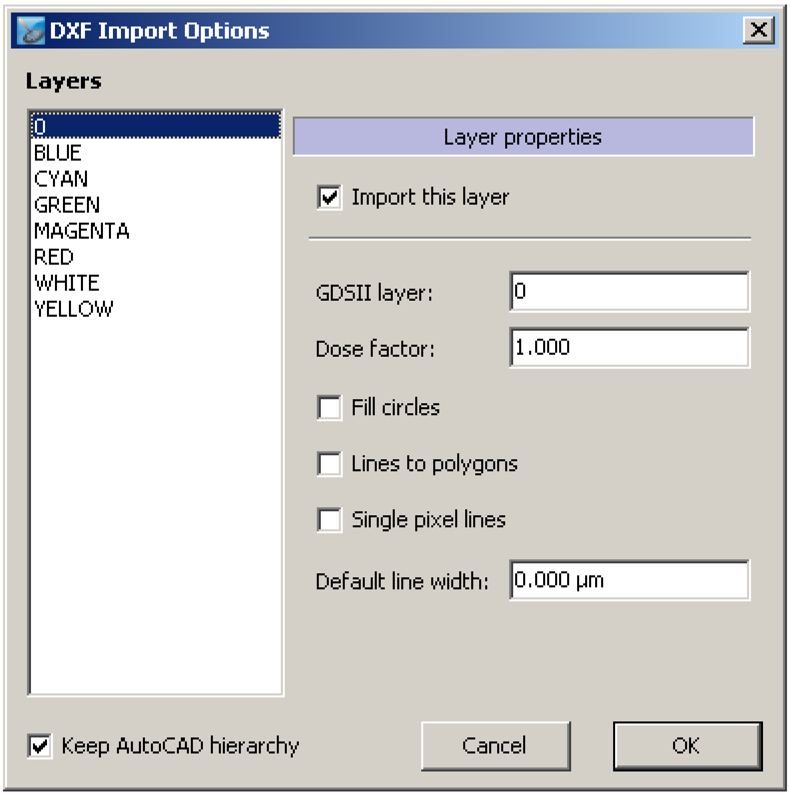


Figure 5-4: **DXF Import Options** subdialog

▶ Select a DXF layer from the **Layers** list.

▶ Select **Import this layer** to import the DXF layer and define the **Layer properties**.

▶ Select **Keep AutoCAD hierarchy** if you want to keep the hierarchical structure of the DXF file.

▶ Confirm with **OK**.

The DXF file will be added to the design and displayed in the **GDSII Database** tree view.

If the size of the imported structure does not meet the original CAD-structure size, you have to change the value of the variable **DXF Scale Factor** via **Extras** ➡ **Settings** ➡ **GDSII database** ➡ **DXF Scale Factor**. For example:

Example for scale factors for modifying the structure size of imported DXF Files

Original size = 200 µm
Imported size = 200 nm
Set scale factor = 1000

The DXF file definition is a comprehensive standard for 2D and 3D graphical drawings. Each new release adds additional features to the definition. The current release DXFTM 2004 has over 1000 tag ID's. Because the NANOSUITE software supports only the relevant subset of the DXF standard, one may occasionally come across features, which are not supported.
Therefore, it is highly recommended to keep the standard elements listed

Table 5-4: Available DXF sections

DXF section	Is converted to GDSII element:
HEADER	Will be ignored.
TABLES	Will be ignored.
BLOCKS	See section (⇒ *DXF import features* on page 16).
ENTITIES	Will be ignored.

Table 5-5: Available hierarchical DXF elements

Hierarchical DXF elements	Is converted to GDSII element:
LINE	PATH with 2 points
CIRCLE	BOUNDARY with 50 points - Filled circle, if checkbox option **Fill circles** is selected. - Circle ring, if checkbox **Fill circles** is cleared.
ARC	BOUNDARY with 50 points as circle arch

Table 5-5: Available hierarchical DXF elements

Hierarchical DXF elements	Is converted to GDSII element:
SOLID	BOUNDARY with 4 points
INSERT	SREF leaving hierarchy AREF leaving hierarchy and multi insert

Table 5-6: Available flat DXF elements

Flat DXF element	Is converted to GDSII element
POLYLINE	BOUNDARY Polygon, if checkbox option **Lines to Polygons** is selected, open paths will be closed automatically PATH If checkbox **Lines to Polygons** is cleared.
VERTEX	Belongs to POLYLINE.
SEQUEND	Belongs to POLYLINE.
LAYER	DATATYPE via dose class file LAYER.DOS.
INSERT	SREF leaving hierarchy. AREF leaving hierarchy and multi insert.

4.3.1 DXF import features

During conversions, additional structures may be added into the GDSII database, like ***model_space**, ***paper_space**, ***paper_space0**. These are internal definitions in a DXF BLOCKS section. They can be safely ignored and deleted. They will however cause a warning message of an illegal character for the structure name. The DXF name is ***model_space**, however the GDSII standard allows only:

"A" through "Z", "a" through "z", and „0“ through „9“ and the special characters Underscore "_", Question mark "?", and Dollar sign "$".Therefore, the "*" is an illegal character. The Raith software simply replaces the "*" with a "_" and issues a warning renaming the structures as:

_model_space, _paper_space, _paper_space0.

4.4 Importing files in CIF format

It is possible to import data files stored in Caltech Intermediate Format by California Institute of Technology (CIF). The hierarchy of CIF files is equal to the GDSII hierarchy and will be kept during import. Supported data types for CIF import are listed in the following.

Table 5-7: Available elements in Caltech Intermediate Format (CIF)

CIF element	Is converted to GDSII element
B	BOX
P	BOUNDARY
9	STRUCTURENAME
C	SREF
R0	STRANS with angle 90°
L	LAYER
(*...*)	comment

Dose values are related via dose classes (file LAYER.DOS) – the same procedure as for import of DXF files. A (symbolic) layer name is assigned via the file **CIF.INI**. For example:

Example for CIF.INI file

```
[LAYER]
TLEN=1
META=2
BND=31
```

5 Preparing GDSII files for export

Some functions of the GDSII module are Raith specific and are not supported by GDSII or graphic software of other vendors. Therefore, a design may be prepared for further externally editing by the following measures.

5.1 Converting to standard GDSII

Most of the CAD systems use different definitions to store special structure elements like text, curves, circles, etc. In addition, the dose information, e.g. as dose factor to scale the patterning time, may be stored in different ways.

To become compatible with common CAD systems, it is necessary to create a general GDSII format. The **Convert userdefs** tool converts all Raith specific structure elements, e.g. text or circles into polygons or open paths. In addition, the dose information can be removed. All changes are done in the same GDSII file.

Applying the **Convert userdefs** command overwrites the original structure. Therefore, we recommend working on a copy.

Convert Raith specific structure elements into standard GDSII format as follows:

▶ Select a structure from the **GDSII Database** tree view and copy it via drag and drop inside the same tree view.

▶ In the GDSII Database menu, choose **Tools** ➡ **Convert userdefs**.

The **Convert Userdef Elements** dialog will be opened:

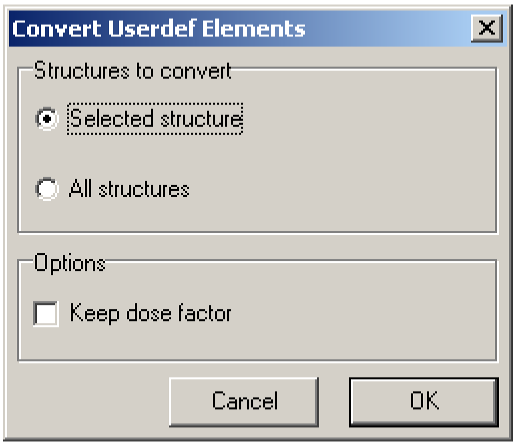


Figure 5-5: **Convert Userdef Elements** dialog

▶ Choose, if only the selected or all structures are to be converted.

▶ Select **Keep dose factor** to preserve the dose factor.

Otherwise, the dose factor will be set to 0.001 so that the number 1 is stored in the GDSII database file, which will be ignored by other CAD systems.

▶ Confirm with **OK**.

All Raith specific structure elements will be converted to a standard GDSII format.

5.2 Removing GDSII hierarchy

Not all GDSII programs from other vendors support hierarchical files. If you want to edit a structure in a GDSII application, which doesn‘t support hierarchical structures, you have to resolve all references. As a result, you get one large flat structure. The structure will be stored in a new GDSII database.

(⇒ Figure 5-15 on page 5-25) shows an example of a hierarchical structure with hierarchy level 0 before unifying:

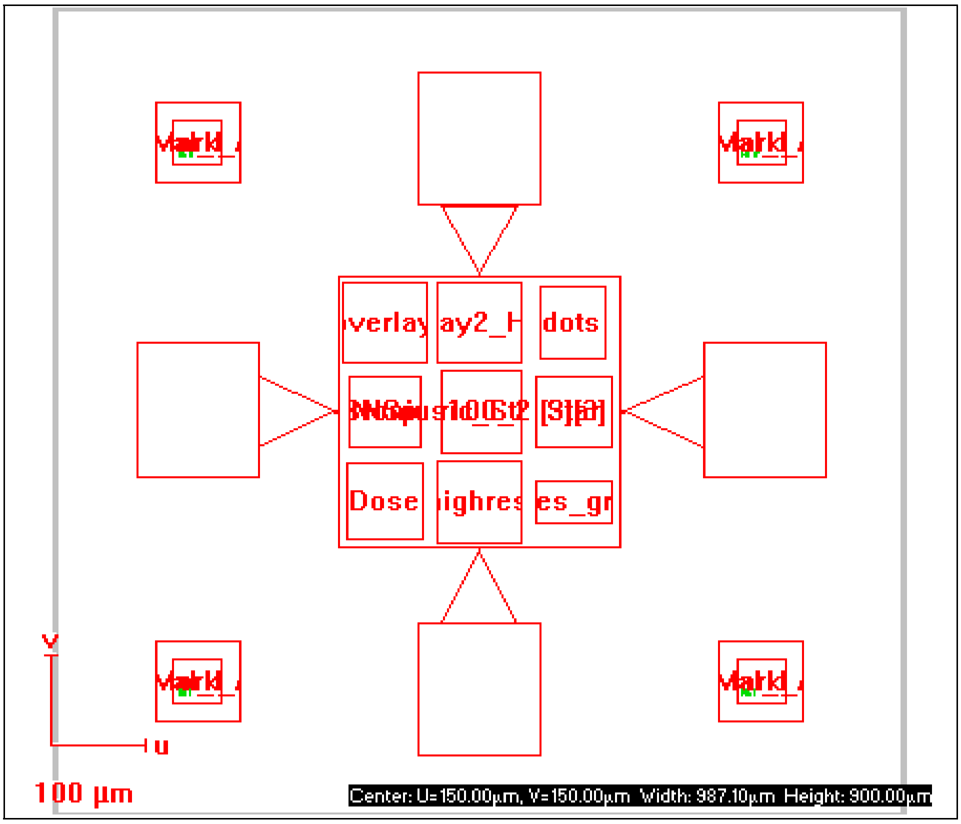


Figure 5-6: Hierarchical structure before unifying, shown with hierarchy level 0

Proceed as follows to unify the structure.

- Select a hierarchical structure from **GDSII Database** tree view.
- In the GDSII Database menu, choose **Tools** ➡ **Unify structure**.

The **Unify structure** dialog will be opened. The name of the selected structure is displayed at the top:

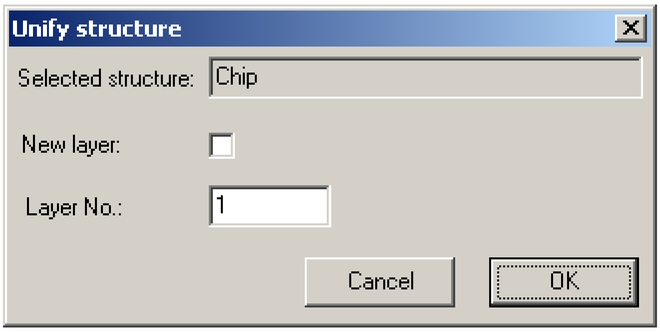


Figure 5-7: **Unify structure** dialog

- Select **New layer**, if you want to store the result in a new layer and enter the number of the layer in **Layer No.**.
- Confirm with **OK**.

All references will be resolved and a large flat structure will be generated:

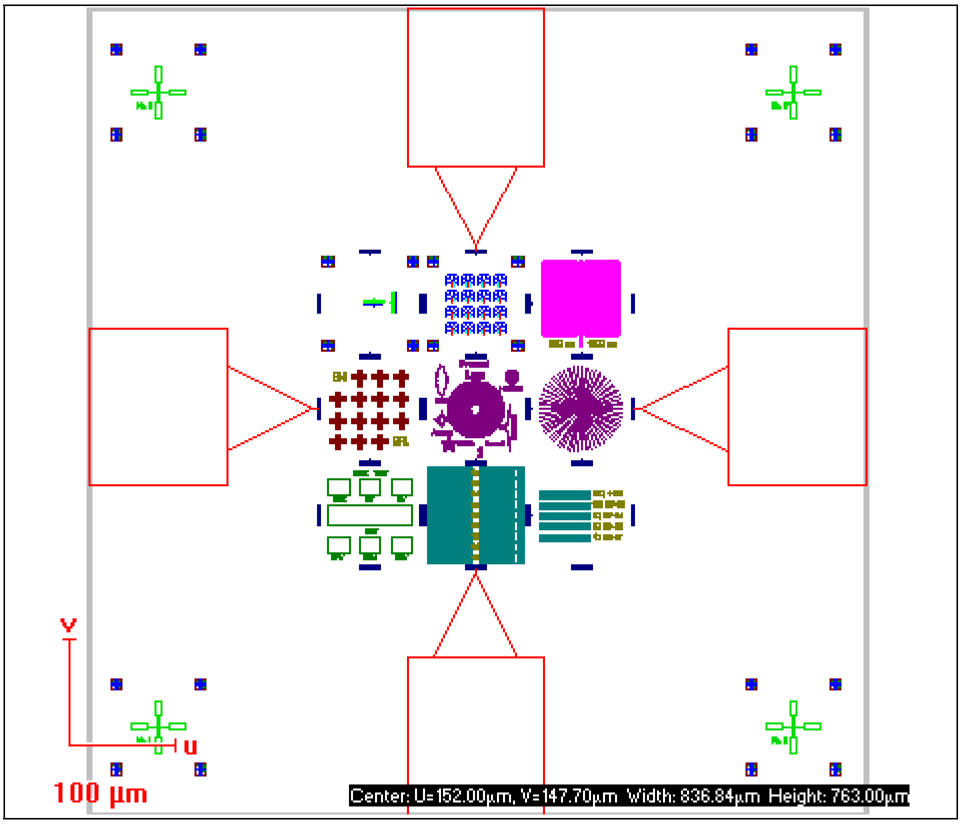


Figure 5-8: Hierarchical structure after unifying, shown with hierarchy level 0

6 Optimizing design for patterning

To avoid long patterning times or the overexposure of overlapping areas of structure elements, the design can be optimized for patterning by the following measures.

6.1 Removing overlaps

To avoid an overexposure of overlapping structure elements it is useful to remove the overlaps before starting a patterning.

(⇒ Figure 5-9 on page 5-21) shows an example for overlapping ares of structure elements in between one structure:

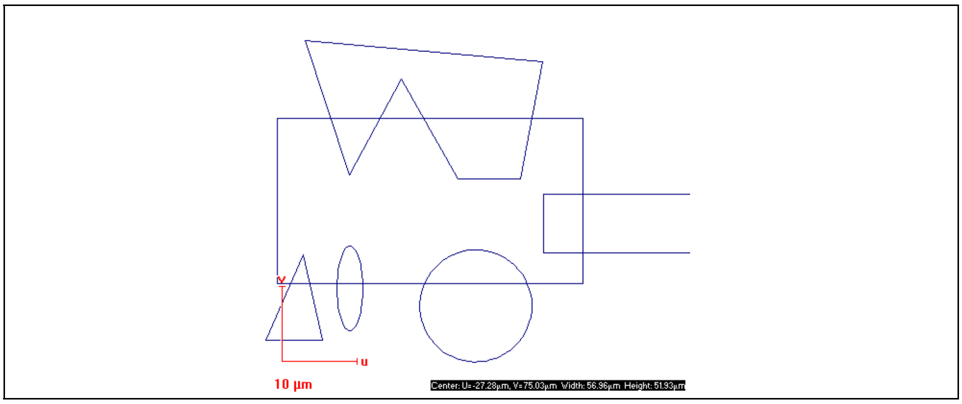


Figure 5-9: Overlapping ares of structure elements

The command for removing overlaps overwrites the original structure. Therefore, we recommend working on a copy.

The command is not removing overlaps within different structure references.

Proceed as follows to remove overlaps:

- ▶ Select a structure from **GDSII Database** tree view and copy it via drag and drop.
- ▶ In the GDSII Database menu, choose **Tools** ➡ **Overlap removal**.

The **Overlaps out** dialog will be opened. The name of the selected structure is displayed at the top:

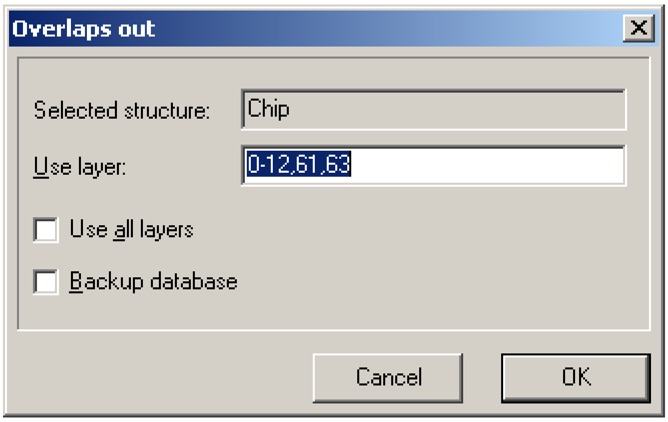


Figure 5-10: **Overlaps removal** dialog

- Do one of the following:
  - In the field **Use layer**, enter, separated by comma, the layers in which you want to remove overlaps.
  - Activate the checkbox **Use all layers** if you want to remove overlaps in all layers of the structure.
- Select **Backup database** if you want to create a backup copy of the database file.
- Confirm with **OK**.

The overlaps will be removed:

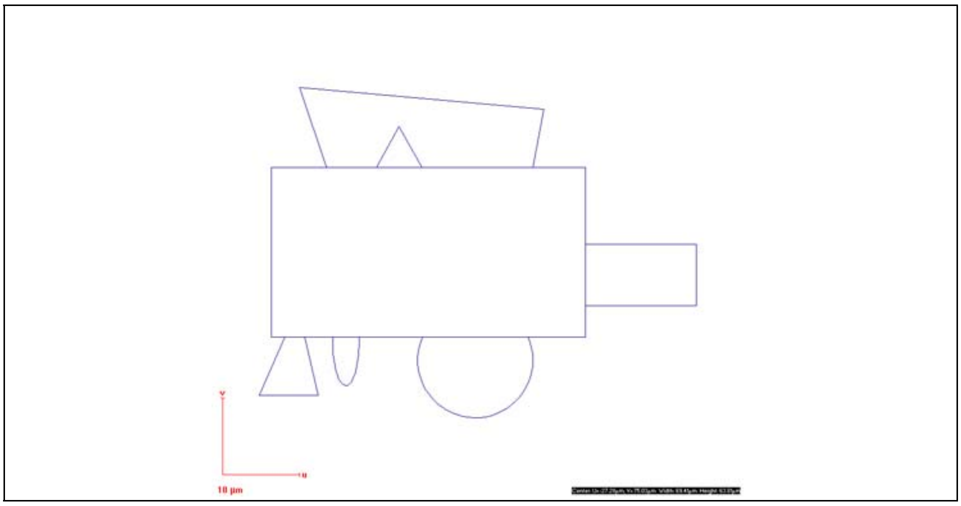


Figure 5-11: Removed overlaps of structure elements

6.2 Sorting elements

Structure elements are numbered in order of their creation time. The first created structure element gets number 01, the second 02, etc. – i.e. the structure elements are numbered independently of their UV positions.

To shorten patterning times, the element sequence can be modified in order to minimize the jump distance from element to element during the patterning. To optimize the structure for patterning, it is useful to change the order of all elements within the selected structure.

(⇒ Figure 5-9 on page 5-21) shows an example of unsorted structure elements in between one structure:

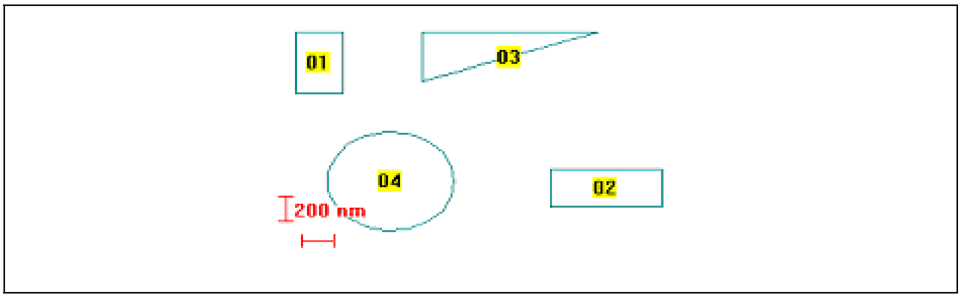


Figure 5-12: Unsorted structure elements

The command for sorting elements overwrites the original structure. Therefore, we recommend working on a copy.
The command is not sorting within different structure references.

Proceed as follows to sort structure elements:

- Select a structure from the **GDSII Database** tree view.
- In the GDSII database menu, choose **Tools** ➡ **Sort elements**.

The **Sort elements** dialog will be opened. The name of the selected structure is displayed at the top:

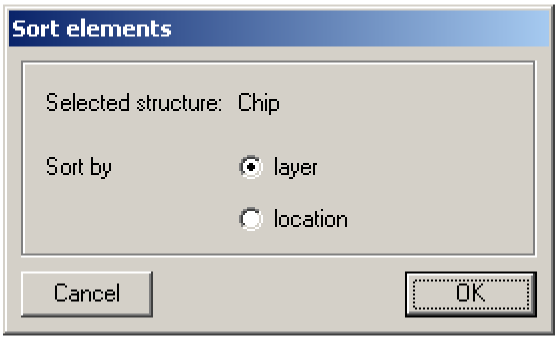


Figure 5-13: **Sort elements** dialog

- Do one of the following:
  - Select **Sort by layer** to sort elements layer by layer successively.

This makes it easier to sort elements within a certain layer manually by using the command **Modify** ➡ **Order** in the **GDSII Editor** dialog. The result can be, for example:

Example for sorted structure elements by layer:

Layer 0: elements #1 - #4
Layer 1: elements #5 - #15
Layer 2: elements #16 - #158

- Select **Sort by location** to sort all elements independently of the layer so that the jump distance from element to element during patterning is minimized automatically.

▶ Confirm with **OK**.

The structure elements will be sorted:

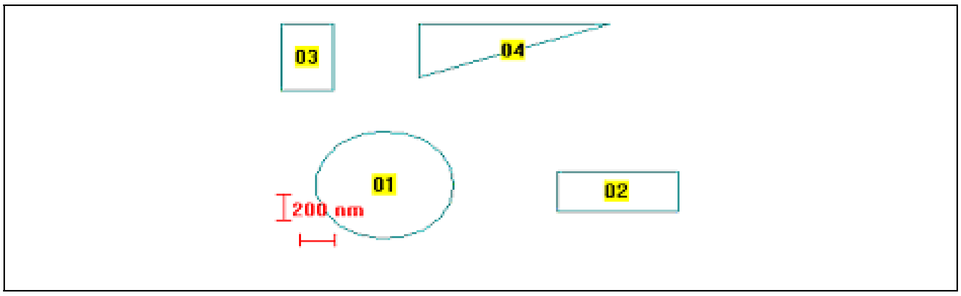


Figure 5-14: After sorting elements by location.

6.3 Merging GDSII elements

To optimize the patterning, it is useful to remove gaps between single structure elements.

(⇒ Figure 5-15 on page 5-25) shows an example of neighboring structure elements separated by little gaps:

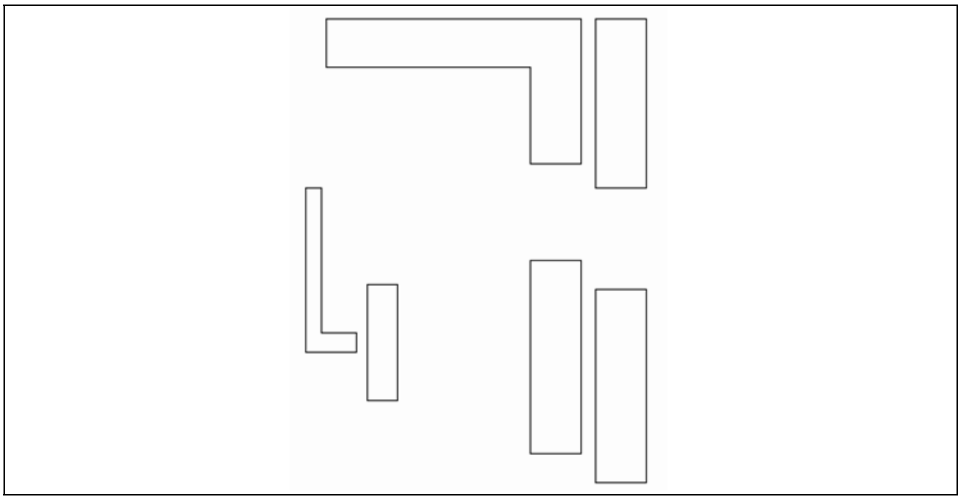


Figure 5-15: Example of structure elements before merging

The command for merging GDSII elements overwrites the original structure. Therefor, we recommend working on a copy.
The command is not merging elements within different structure references.

Proceed as follows to merge structure elements:

- Select a structure from **GDSII Database** tree view and copy it via drag and drop.
- In the GDSII database menu, choose **Tools** ➡ **Merge elements**.

The **Merge elements in selected structure** dialog will be opened. The name of the selected structure is displayed at the top:

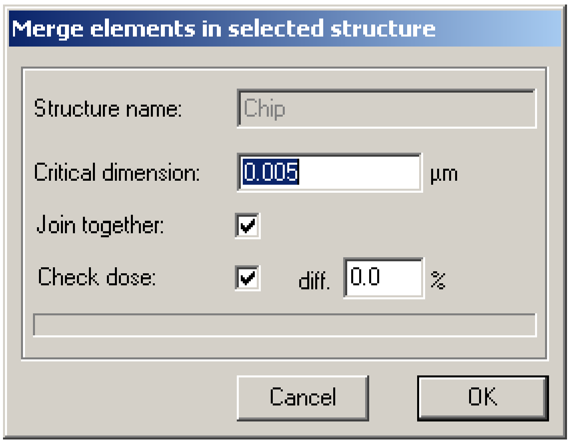


Figure 5-16: **Merge elements in selected structure** dialog

- Type in a **Critical dimension** to merge elements, if the distance between them is the same or less than the entered value.
- Select **Join together** to merge elements, i.e. the elements will not only be combined visually but they will also become one single structure element.
- Select **Check dose** to merge only elements which have a dose difference of less than the entered value.
- In the field **diff.**, type in a dose difference as threshold under which the structure elements are to be merged.
- Confirm with **OK**.

The gaps between the structure elements will be removed:

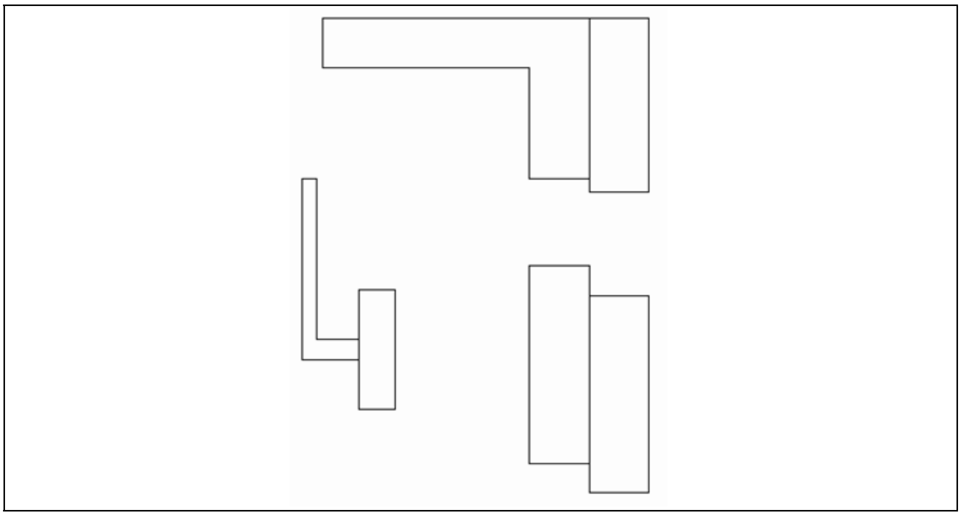


Figure 5-17: Example of structure elements after merging

6.4 Joining GDSII elements together

To optimize the patterning, it is useful to join in the minimum two GDSII elements to one larger GDSII element.

The command for joining GDSII elements together overwrites the original structure. Therefore, we recommend working on a copy.

Proceed as follows to join structure elements together:

- Select a structure from **GDSII Database** tree view and copy it via drag and drop.
- In the GDSII database menu, choose **Tools** ➡ **Sort elements** and sort the elements by location (⇒ *Sorting elements* on page 5-23).
- In the GDSII database menu, choose **Tools** ➡ **Join sorted elements**.

The **Join sorted elements in selected structure** dialog will be opened. The name of the selected structure is displayed at the top:

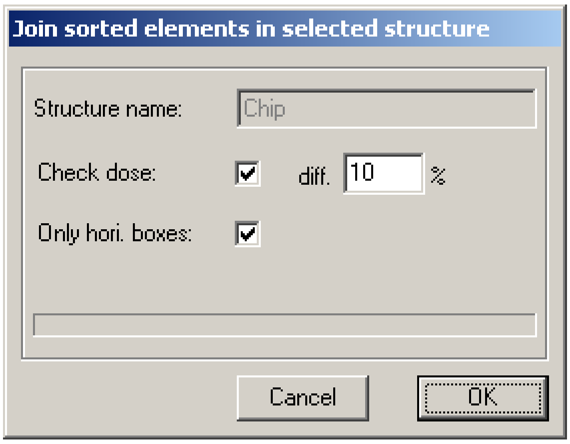


Figure 5-18: **Join sorted elements in selected structure** dialog

- Select **Check dose** to join only elements which have a dose difference of less than the entered value.
- In the field **diff.**, type in a dose difference as threshold under which the structure elements are to be joined.
- Select **Only hori. boxes** to join boxes along U direction only.
- Confirm with **OK**.

The structure elements of the selected structure are joined together.

7 Creating structure elements

Adding and modifying of structure elements is only possible in the **GDSII Editor** dialog – except boolean operations tools (⇒ *Using Boolean operations* on page 5-40).

7.1 Working with the Toolbox

The toolbox offers a fast access to often used commands for creating and modifying structure elements. The toolbox is an add-on window, which stay on top of all activated windows until they are closed.

All of the structure elements, described in the following sections, you can add and modify via the toolbox tools.

- Open the toolbox by clicking on in the tool bar of the **GDSII Editor** dialog.

The selected tools will be repeated automatically until pressing **ESC** or choosing the cursor tool.

For a detailed description of all tools in the toolbox refer to (⇒ *Toolbox* on page 5-135).

7.2 Adding ellipses

Every drawn ellipse will be surrounded by a rectangle. The rectangle serves to define the coordinates.

7.2.1 Adding ellipses graphically

- In the toolbox choose .
- Move the cursor to the desired position within the structure and click the left mouse button.

The starting point for drawing the ellipse is set.

- Move the cursor diagonally until the ellipse has the desired size and position.
- Click the left mouse button again.

The ellipse will be finished and a rectangle will be displayed surrounding it.

- Press **ESC** or choose to complete the tool.

Each mouse click can be replaced by pressing the **Space bar**. By pressing **Shift** after the first mouse click the first point is interpreted as the center of the ellipse. By pressing **Control** the ellipse will be made a perfect circle. Both keys can be pressed simultaneously.

The ellipse will be added to the active layer.

7.2.2 Adding ellipses digitally

- In the tool bar choose .
- Press any digit key (0 through 9) to call up the **Enter Point** dialog for digital input.

  The input fields are preset with the coordinates of the current cursor position.

- Enter the coordinates of the first corner of the rectangle surrounding the ellipse on the U and V axis.
- Choose **Next** to type in the coordinates of second corner.
- Choose **OK** to complete the ellipse.

  The ellipse will be added to the active layer.

7.3 Editing ellipses

To access all properties of a circle double click it using the left mouse button. The **Circle Properties** dialog will be opened providing the according options.

For a detailed description of the control elements of the dialog, refer to (⇒ *Circle Properties dialog* on page 5-112).

7.4 Adding dots

A dot is a single point with zero extension.

7.4.1 Adding dots graphically

- In the toolbox choose .
- Move the cursor to the desired position within the structure and click the left mouse button or press the **Space bar**.

  The dot will be added to the active layer.

7.4.2 Adding dots digitally

- In the toolbox choose .
- Press any digit key (0 through 9) to call up the **Enter Point** dialog for digital input.

  The input fields are preset with the coordinates of the current cursor position.
- Enter the UV coordinates of the dot.
- Choose **OK** to complete the input.

  The dot will be added to the active layer.

7.5 Editing dots

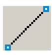
To modify the position, the dose factor, and the layer of the dot double click it using the left mouse button. The **Edit Dot** dialog will be opened providing the according options.

For a detailed description of the control elements of the dialog, refer to (⇒ *Edit Dot dialog* on page 5-117).

7.6 Adding open and closed paths

There is a difference between a closed path and a polygon: A polygon has an area whereas a path consists of a border only. This difference can be made visible by choosing the menu command **Options** ➡ **Fill**.

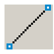
7.6.1 Adding open and closed paths graphically

- In the tool bar choose .
- Move the cursor to the desired position within the structure and click the left mouse button at each corner of the path sequentially. Each mouse click can be replaced by pressing the **Space bar**. Simultaneously, an open path will be drawn.
- Click the right mouse button at the last corner to complete the path. This mouse click can be replaced by pressing **Enter**.

  The open or closed path will be added to the active layer.

It is possible to assign each open path a line width and zero value leads to single pixel lines. These single pixel lines are not standard GDSII elements but useful for ultra-high resolution applications.

7.6.2 Adding open and closed paths digitally

- In the tool bar choose .
- Press any digit key (0 through 9) to call up the **Enter Point** dialog for digital input.

  The input fields are preset with the coordinates of the current cursor position.
- Enter the coordinates of the first corner.
- Choose **Next** to type in the coordinates of the following corners.
- After entering the coordinates of the last corner, choose **Last** to complete the path.
- Choose **OK** at any time to position the the next corner with the mouse again.

  The path will be added to the active layer.

7.7 Editing open and closed paths

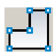
To access all properties of a path double click it using the left mouse button. The **Edit Path** dialog will be opened providing the according options.

For a detailed description of the control elements of the dialog, refer to (⇒ *Edit Path dialog* on page 5-115).

7.8 Adding polygons

The following sections explain how the polygons can be added.

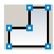
7.8.1 Adding polygons graphically

- In the tool bar choose .
- Move the cursor to the desired position within the structure and click the left mouse button at each corner of the polygon sequentially.

  Each mouse click can be replaced by pressing the **Space bar**. Simultaneously, a polygon will be drawn.
- Click the right mouse button at the last corner to complete the path. This mouse click can be replaced by pressing **Enter**.

  The polygon will be added to the active layer.

7.8.2 Adding polygons digitally

- In the tool bar choose .
- Press any digit key (0 through 9) to call up the **Enter Point** dialog box for digital input.

  The input fields are preset with the coordinates of the current cursor position.
- Enter the coordinates of the first corner.
- Choose **Next** to type in the following corners.
- After entering the coordinates of the last corner, choose **Last** to complete the path. Choose **OK** at any time to position the next corner with the mouse again.

  The polygon will be added to the active layer.

7.9 Editing polygons

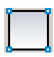
To access all properties of a polygon double click it using the left mouse button. The **Edit Polygon** dialog will be opened providing the according options.

For a detailed description of the control elements of the dialog, refer to (⇒ *Edit Polygon dialog* on page 5-113).

7.10 Adding rectangles

The following sections explain how the rectangles can be added.

7.10.1 Adding rectangles graphically

- In the tool bar choose .
- Move the cursor to the desired position within the structure and click the left mouse button.
- Move the cursor to the opposite corner of the rectangle and click the left mouse button again. Simultaneously, the rectangle will be drawn.

Each mouse click can be replaced by pressing **Enter** or by pressing the **Space bar**. By pressing **Shift** after the first mouse click the first point is interpreted as the center of the rectangle. By pressing **Control** the rectangle will be made a perfect square. Both keys can be pressed simultaneously.

The rectangle will be added to the active layer.

7.10.2 Adding rectangles digitally

- In the tool bar choose .
- Press any digit key (0 through 9) to call up the **Enter Point** dialog for digital input.

  The input fields are preset with the coordinates of the current cursor position.
- Enter the coordinates of the first corner of the rectangle.
- Choose **Next** to type in the coordinates of the second corner.
- Choose **OK** to complete the input.

  The rectangle will be added to the active layer.

7.11 Editing rectangles

To access all properties of a rectangle double click it using the left mouse button. The **Edit Rectangle** dialog will be opened providing the according options.

For a detailed description of the control elements of the dialog, refer to (⇒ *Edit Rectangle dialog* on page 5-111).

7.12 Adding text elements

STEP 1: Entering and positioning text

- In the toolbox .
- Move the cursor to the desired position within the structure and click the left mouse button.

  The **Text Properties** dialog will be opened:

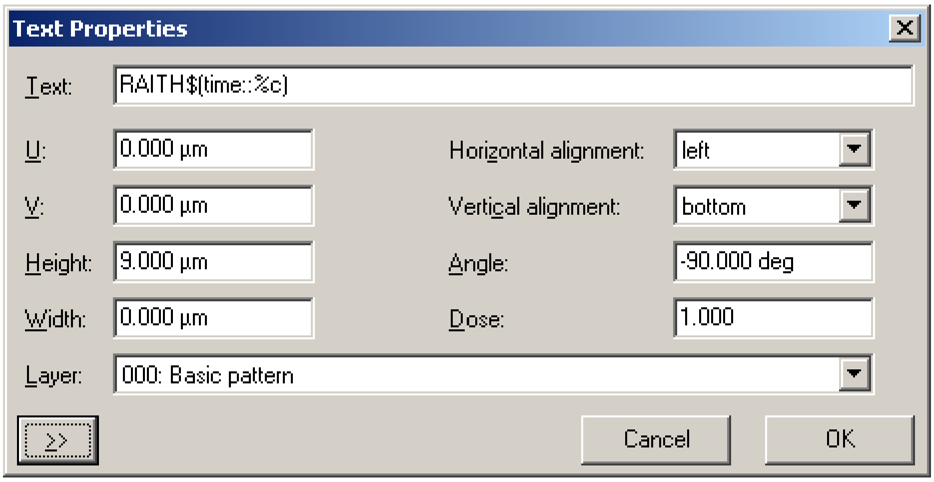


Figure 5-19: **Text Properties** dialog

- Enter the text as clear text into the **Text** field.
- Define a reference point for the text in µm within the field **U** and **V**.
- Define the alignment of the text with respect to this reference point by selecting the **Horizontal alignment** and **Vertical alignment**.
- Enter the **Height** of the letters in µm.
- Enter the **Width** of the letters in µm.

  A value of zero will lead to letters consisting of single pixel lines. The preset line width, defined via **Add** ➡ **Preset** ➡ **Line width** is ignored.
- Enter the rotation **Angle** of the text in degrees.
- Enter the **Dose** to be assigned as a factor with 1 = 100% of clearing dose.
- Select the **Layer** to be assigned to the text element.

STEP 2: Adding time strings and VDB variables

- Click on **>>** to maximize the **Text macros** subset:

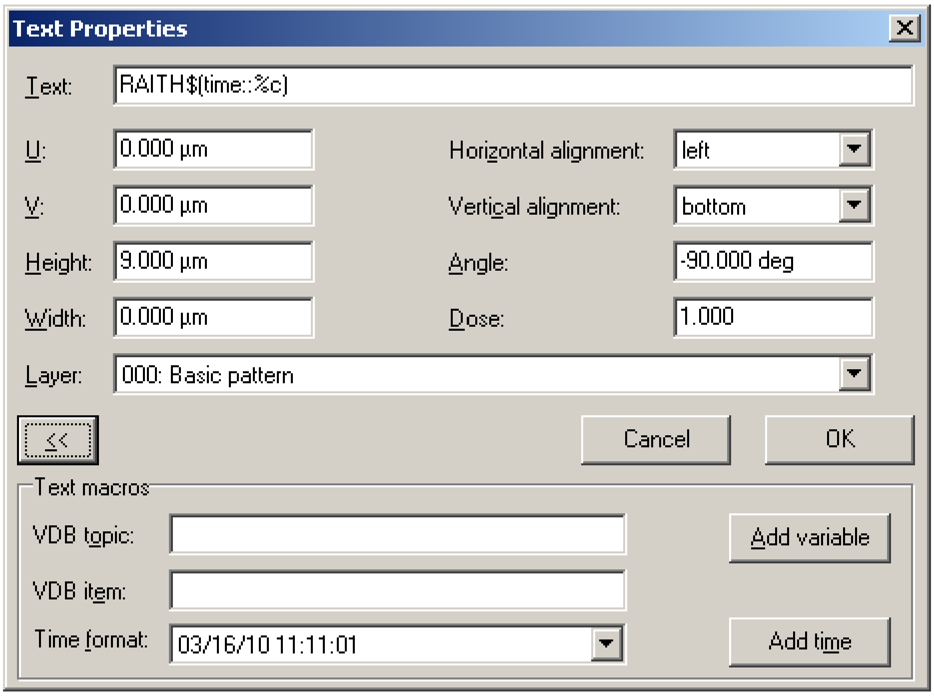


Figure 5-20: **Text Properties** dialog – **Text macros** subset

- Enter the **VDB topic** and **VDB item** and press **Add variable** to copy corresponding string into the **Text** field. The syntax is $(vdb:<topic>.<variable>) and can be edited afterwards.

  The parameters <topic> and <variable> define the name of the topic and the name of the variable listed under that topic within the variables database file (VDB) – for example:

> **Example for the use of a VDB variable**
>
> The dwell time used for a patterning is stored in the variable "dwelltime" under the topic "variables". One may want to pattern this used dwell time together with the pattern for test purposes. For this, use the string:
>
> Dwell time = $(vdb:variables.dwelltime) ms
>
> If the dwell time is set to 0.01, this leads to a patterning of
> Dwell time = 0.01 ms.

- Select time and date from the **Time format** list and press **Add time** to copy the time string into the **Text** field.

  The syntax for this kind of text entry is $(time:<topic>.<variable>:<format>). The parameters <topic> and <variable> have the same meaning as described above. If these parameters are left free, the actual date and time is used. If parameters are used then the VDB is interpreted as the number of seconds since 1/1/1970 1:00:00. The parameter <format> defines the output format of the time string and can be edited afterwards. Following formats are available.

Table 5-8: Format of time string

<format>	Result	Description
%a	Sun, Mon, …	Short name of weekday
%A	Sunday, Monday, …	Full name of weekday
%b	Jan, Feb, Mar, …	Short name of month
%B	January, February, …	Full name of month
%c	e.g. 04/21/01 10:21:33	Date and time
%d	01 - 31	Day number of month
%D	e.g. 04/21/01	Date
%H	00 - 23	Hour (24 hour format)
%I	01 - 12	Hour (12 hour format)
%j	001 - 366	Day number within year
%m	01 - 12	Month number within year
%M	00 - 59	Minute
%p	am - pm	Used together with %I
%S	00 - 59	Seconds
%U	00 - 52	Week number (first day = Sun)
%w	0 - 6	Weekday number (0 = Sun)
%W	00 - 52	Week number (first day = Mon)
%x	e.g. 04/21/01	Date
%X	e.g. 10:21:33	Time
%y	00 - 99	Year without century

Table 5-8: Format of time string

<format>	Result	Description
%Y	e.g. 2004	Year with century

If the parameter <format> is left free, %c is used as a default value.

7.13 Adding bitmaps

You can embed bitmaps into the design. There are two possibilities to embed a bitmap:

- Bitmap as reference (⇒ *Adding bitmaps as reference* on page 5-37)
- Bitmap as elements (⇒ *Adding bitmaps as elements* on page 5-38)

7.13.1 Adding bitmaps as reference

You can add the link to a black and white image, which will be patterned in a raster mode – e.g. if you would like to transfer the boundaries of the structure. Areas which are not black or white will not be patterned.

1 bit black/white bitmaps are supported by default. If 3Lith license is provided, 8 bit gray scale bitmaps import is also possible (please refer to the chapter **Patterning** to learn more about **Grayscale Bitmap** handling ⇒ *Grayscale Bitmap (option)* on page 14-54). If you try to reference to any other bitmap type, a corresponding error message will be displayed.

Proceed as follows to embed a Bitmap as reference:

▶ In the toolbox select .

▶ Move the cursor to the desired position within the structure and click the left mouse button.

The **Bitmap Properties** dialog will be opened:

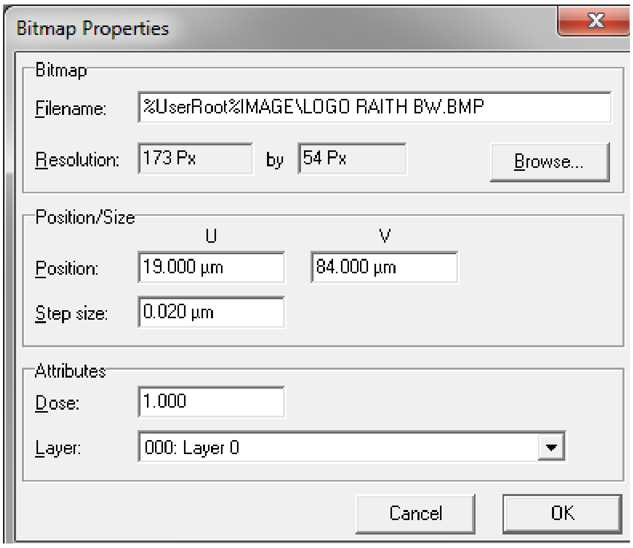


Figure 5-21: **Bitmap Properties** dialog

- Enter the **Filename** and the **Resolution** in pixels or use **Browse...** to search for the bitmap file.
- Enter the **Position** of the lower left corner of the bitmap in µm.
- Enter the **Step size** in metric units used between pixels within the bitmap.

Take care that the product 'Stepsize' x 'bitmap pixel' does not exceed the writefield size.

- Enter the **Dose** to be assigned as a factor with 1 = 100% of clearing dose.
- Select the **Layer** to be assigned.
- Confirm with **OK**.

The bitmap will be displayed in the UV window.

7.13.2 Adding bitmaps as elements

You can import a bitmap so that each pixel of the bitmap is a separate GDSII element. All GDSII elements are automatically grouped.

Proceed as follows to import a bitmap:

- In the menu bar choose **Add ➡ Bitmap ➡ As elements** and click into the design.

The **Insert Bitmap** dialog will be opened:

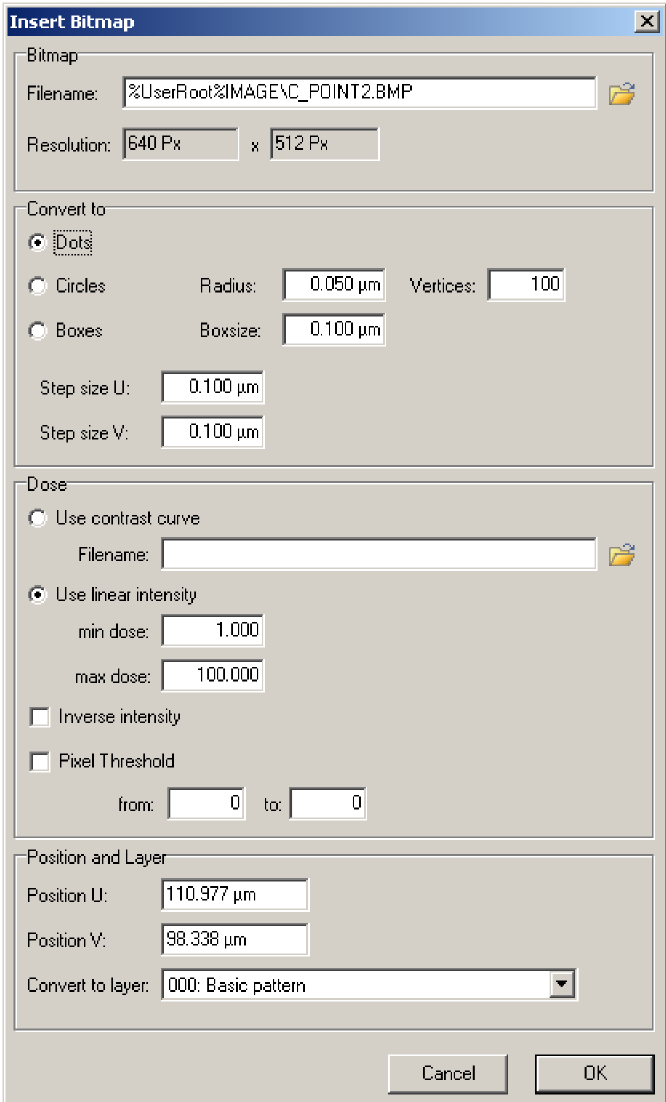


Figure 5-22: **Insert Bitmap** dialog

- Enter the **Filename** or choose to browse for the bitmap file.
- In the subset **Convert to**, select **Dots**, **Circles**, or **Boxes** to specify the GDSII element. Enter their properties when required.
- Enter **Step size U/V** to specify the distance between the elements.
- Define the dose as follows:

  It is possible to set the dose by using the colors values of the image. The intensity for each pixel is determined using the following formula:

  Intensity = (Red + Green + Blue) /3.

  This intensity is then transferred into a dose for the corresponding GDSII element.

  - Select use linear intensity.

- Use **min dose** and **max dose** to specify the minimum and maximum dose factor.
- Select **Inverse intensity** to apply the smallest intensity to the maximum dose factor and vice versa.
- Select **Pixel threshold** to exclude a certain intensity range.

Optional, you can define the dose using a contrast curve.
The contrast curve is a text file, in which the assignment of the dose values to the resist thickness is defined. The following format is used per row:
Dose value as a real figure > TAB sign > resist thickness as a real figure.
You can find some examples for contrast curve definitions in the **<Raith root>** directory ➡ **Proxy** folder.

▶ Enter the Position in U and V, if necessary.

▶ Select the Layer to be assigned. The preset layer, defined via **Add** ➡ **Preset** ➡ **Layer** is ignored.

7.14 Using Boolean operations

The Boolean operation tools are available in the **Editor** and **Viewer** dialog.

To create complex forms of structure elements, which cannot be created with the rectangle, polygon and circle tools, you can use some boolean operations. It is possible to perform boolean operations between elements on different layers for the whole structure.

(⇒ Figure 5-23 on page 5-41) shows the result of Applying boolean operations on structure elements in different layers as an example:

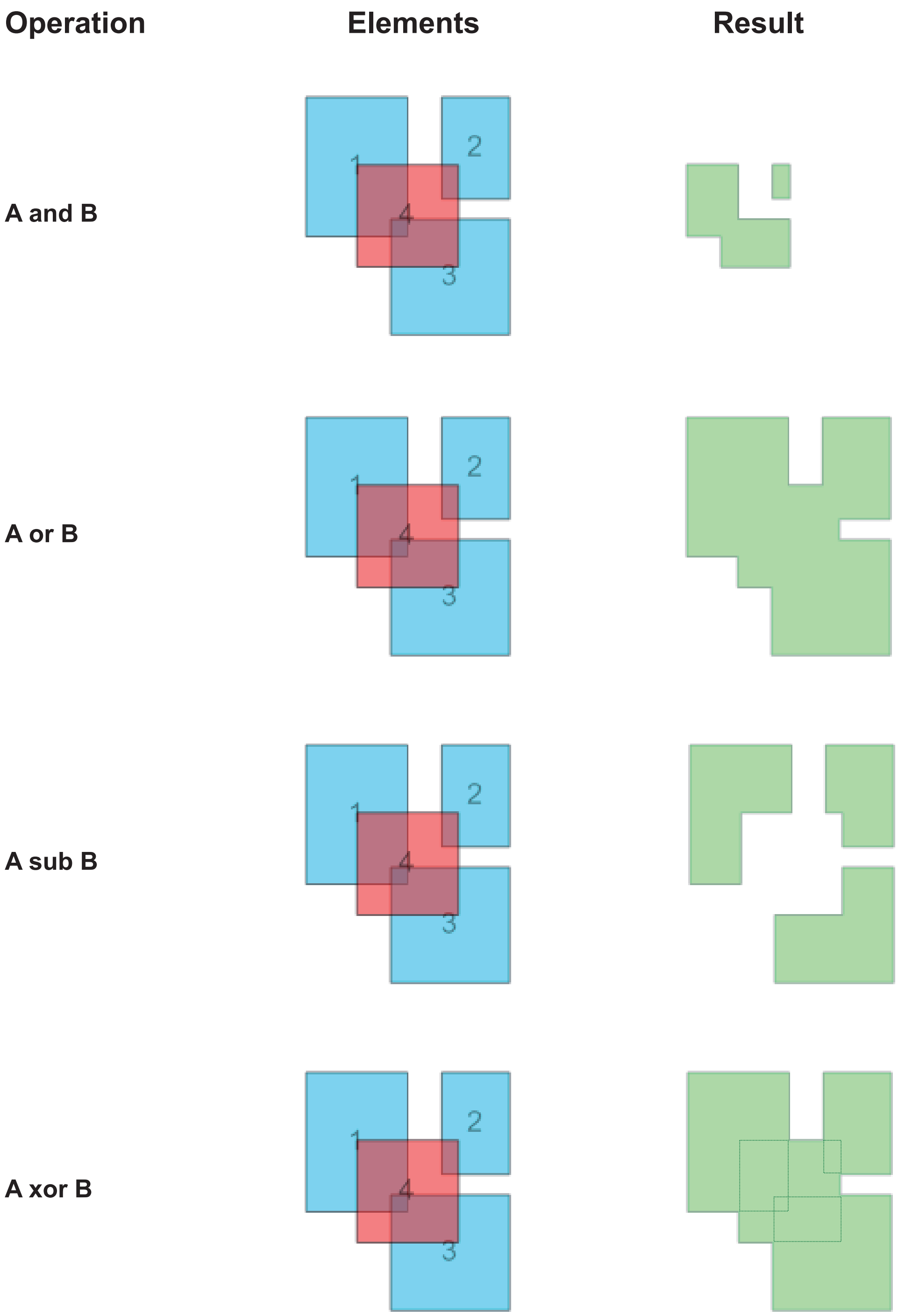


Figure 5-23: Result of Boolean operations on structure elements in different layers. Elements 1, 2, 3 are on one layer whereas 4 is on a different layer.

Proceed as follows to perform boolean operations in different layers:

- ▶ Select a structure in the **GDSII Database** tree view.
- ▶ In the **GDSII Database** menu choose **Tools** ➡ **Boolean operations**.

The GDSII Editor/Viewer will be closed and the **GDSII Boolean Operations** dialog will be opened:

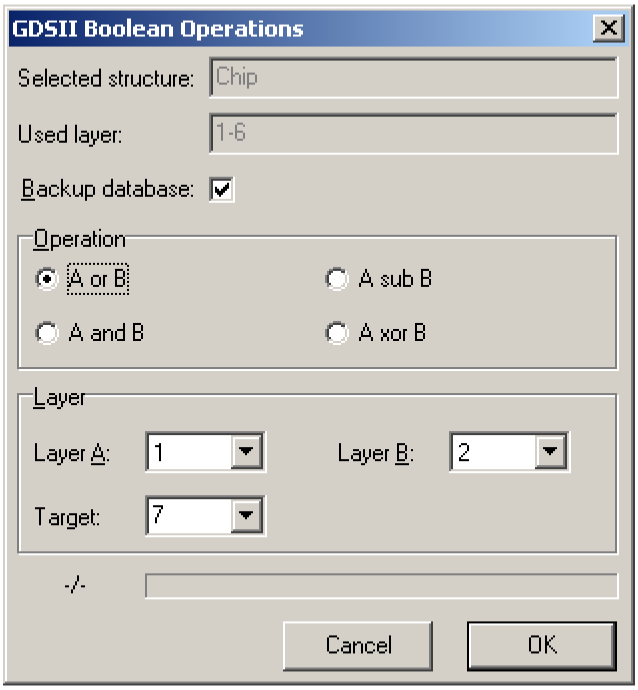


Figure 5-24: **GDSII Boolean Operations** dialog

- Select **Backup database**.

The Boolean operations function overwrites the original structure. Therefore, we recommend to select **Backup database:** or working on a copy of the structure.

- In the **Operation** subset, select the Boolean operation you want to apply (⇒ Figure 5-23 on page 5-41).
- Select **Layer A** and **Layer B** as source layers.
- In the combo boxes Layer A and Layer B select the number of the source layers.
- In the **Target** combo box select the number of the layer, in which the result should be displayed.
- Confirm with **OK**.

  The Boolean operation will be applied and a back up of the structure will be saved in the **<User>** ➡ **GDSII** ➡ **Backup** directory.

- Reopen the structure in the GDSII viewer or editor and check the result in the target layer.

7.15 Creating FBMS elements (option)

In FBMS (Fixed Beam Moving Stage) mode, straight and curved lines of any shape and length can be exposed with a stationary or expanded electron beam by controlled continuous stage motion. This way of exposing patterns avoids fractioning of pattern at writefield boundaries, resulting in stitch-free exposure of path elements.

The lines to be fabricated by this mode are handled and stored in a special way, called **UserDefs**, integrated into the GDSII data file. The lines are assigned and identified with the type “curved paths”, which is a Raith proprietary GDSII format extension. “Curved Paths” can be defined and exposed with zero or finite line widths. Moreover, there is a separate dose factor assignment for each curved path. Please refer to the chapter ⇒ *FBMS (option)* on page 14-6 to learn more about patterning with FBMS.

7.15.1 Adding FBMS paths

FBMS paths can be defined straight or as an arc of circle, and with zero or finite width.

Proceed as follows to add a FBMS path element to a structure:

- Select **Add** ➡ **FBMS** ➡ **Path** menu command.
- Move the cursor to the desired position within the GDSII Editor and click the left mouse button at each corner of the path sequentially. Each mouse click can be replaced by pressing the **Space bar**. Simultaneously, an FBMS path will be drawn.
- Click the right mouse button at the last corner to complete the path. This mouse click can be replaced by pressing **Enter**. The FBMS path will be added to the active layer.
- To modify the position, the dose factor, the layer and the width of the FBMS element, double click it using the left mouse button. The **Edit FBMS Path** dialog will be opened providing the according options.

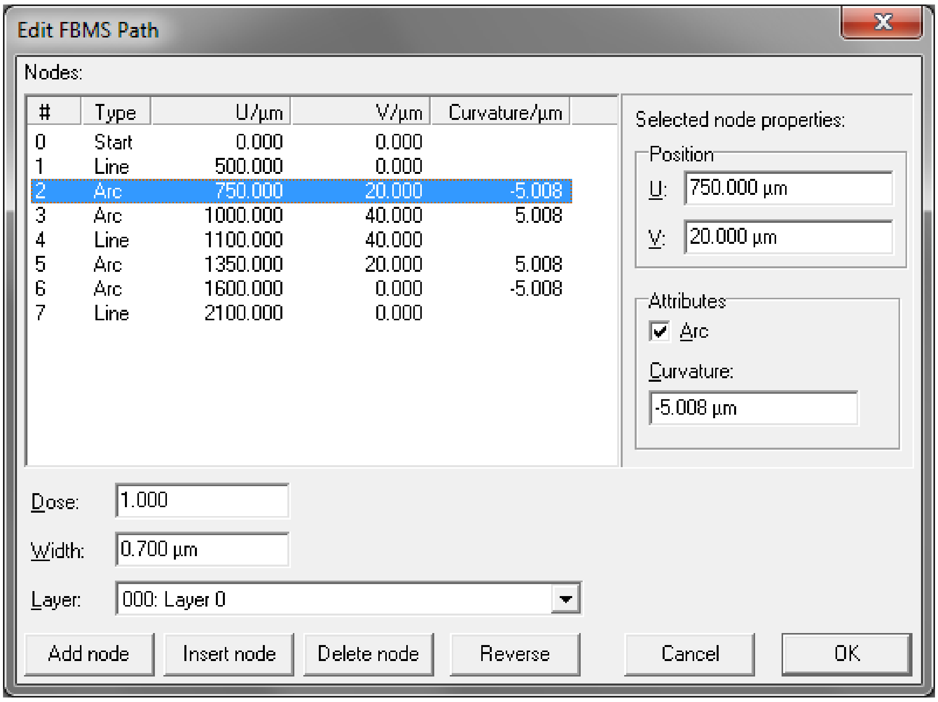


Figure 5-25: **Edit FBMS Path** dialog to define FBMS elements.

- To change a straight segment into a curved segment, select a node, tick the element **Arc**, and enter the wanted height of the segment in **Curvature**. The arc will be added between the selected node and the previous node. A positive height results in a curved path left of the former straight line. A negative height generates the arc on the opposite site.

7.15.2 Adding FBMS circles

Proceed as follows to add a FBMS circle path element to a structure:

- Select **Add** ➡ **FBMS** ➡ **Circle** menu command.
- Move the cursor to the approximate corners of the circle area within the GDSII Editor and click the left mouse button at the two corners sequentially. Simultaneously, a closed circular path will be drawn.
- To modify the center and the radius of the circle as well as the dose factor, the layer and the width, double click it using the left mouse button. The **Edit FBMS Path** dialog will be opened providing the according options.

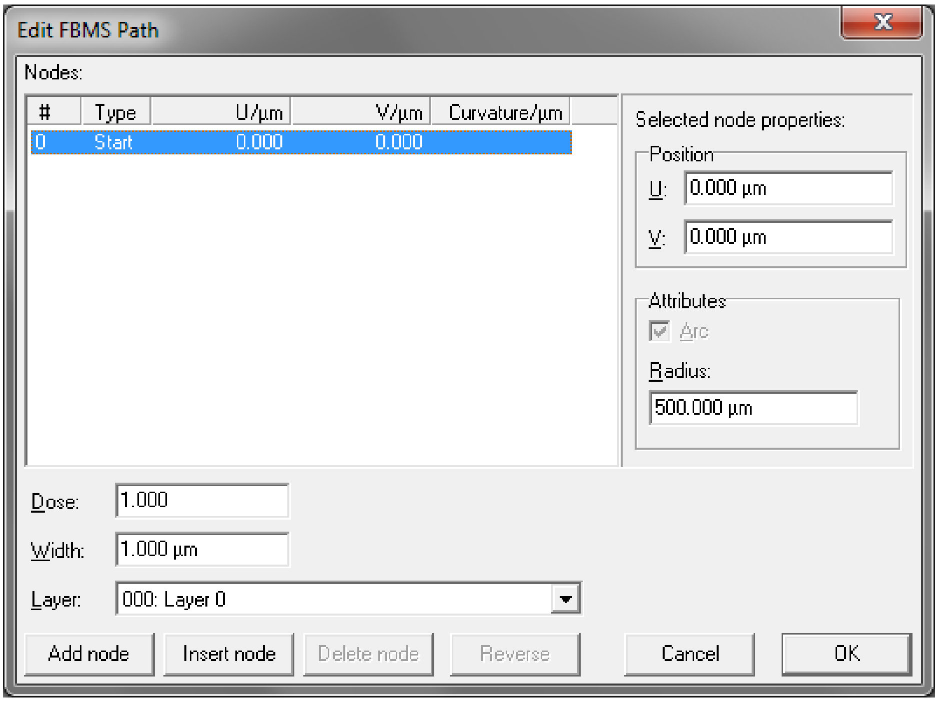


Figure 5-26: **Edit FBMS Path** dialog to define FBMS elements.

7.16 Creating MBMS elements (option)

In MBMS (Modulated Beam Moving Stage) exposure mode, the beam movement is defined such that the combination of repetitive patterning and synchronized continuous movement of the laser interferometer stage results in stitch-free, strip-shaped periodic structures. MBMS elements are handled and stored in a Raith proprietary GDSII format extension (UserDefs), which is integrated into the GDSII data file. The elements are defined as straight paths with a basic pattern that is repeated parallel and perpendicular to the path. Please refer to the chapter ⇒ *MBMS (option)* on page 14-11 to learn more about patterning with MBMS.

7.16.1 Adding MBMS paths

A photonic crystal is a typical application for MBMS. As example, this section explains how to define the corresponding MBMS element.

- The base element for a photonic crystal is a circle. Create a new structure called "circle" in the GDSII database and define a single circle, e.g. with a diameter of 250 nm (⇒ *Adding ellipses* on page 5-28).

The orientation of the base element needs to be defined such that the rotation fits to an MBMS path in direction of the positive u axis. For any MBMS path in another direction the pattern will be automatically rotated during exposure. This allows to use a single base element for different MBMS paths in different directions.

Proceed as follows to add an MBMS path element to a structure:

- Create another new structure called "photonic crystal".
- Select **Add** ➡ **MBMS path** menu command.
- Move the cursor to the desired starting point of the path within the structure and click the left mouse button. Simultaneously, an MBMS path will be drawn. Afterwards click the left or right mouse button at the endpoint of the path. The MBMS path will be added to the active layer. Each mouse click can be replaced by pressing the **Space bar**.
- Double click the element using the left mouse button. The **Edit MBMS Path** dialog will be opened providing the according options.

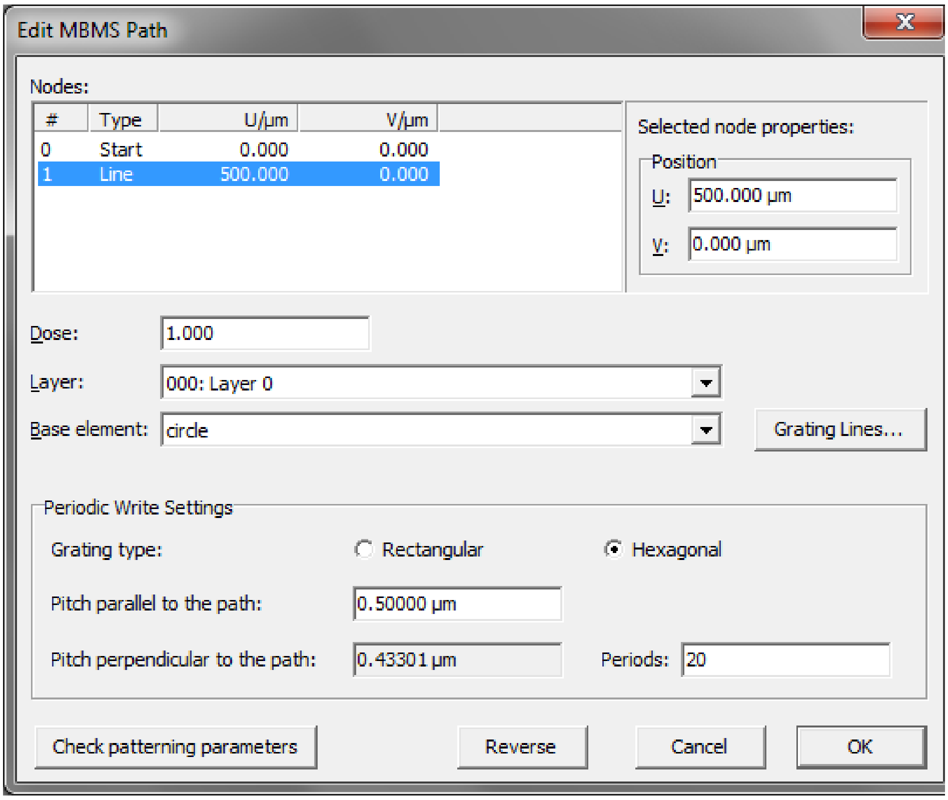


Figure 5-27: **Edit MBMS Path** dialog to define MBMS elements.

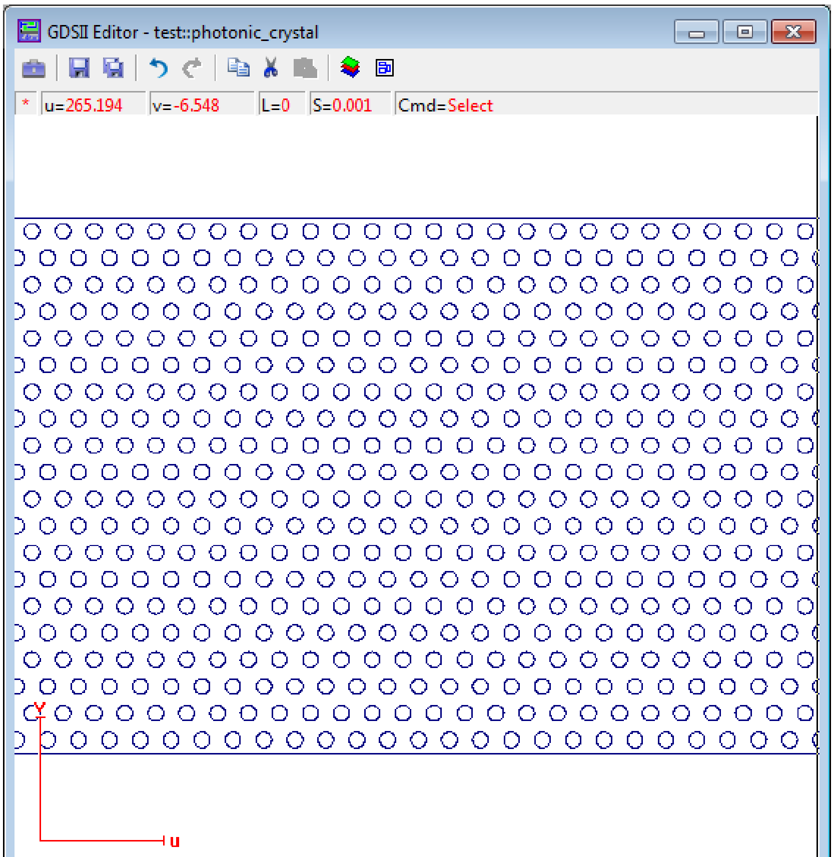


Figure 5-28: **GDSII Editor** with **MBMS** element.

- If necessary, modify the positions of start and end point of the path by entering the correct numbers.
- If necessary, modify **dose factor** and **layer**.
- Select the previously defined circle as **base element**.
- For the here shown example with the photonic crystal, select **hexagonal** as **grating type**.
- Enter the **pitch parallel to the path**, e.g. 0.5µm. The pitch perpendicular to the path will be calculated automatically for hexagonal gratings.
- Enter the number of periods arranged perpendicular to the path, e.g. 20.
- Choose **OK** to complete the MBMS path.

The MBMS element will be added to the active layer.

In order to visualize the placement of the base elements within the MBMS path (⇒ Figure 5-28 on page 5-47), select **Options** ➡ **MBMS periodic cells** or press **B**.

In order to check if the layout is feasible with respect to the current patterning settings (⇒ *MBMS (option)* on page 14-11), press the button **Check patterning parameters**. The number of pixels in the base element, the total number of pixels within a period and the resulting stage speed will be displayed (⇒ Figure 5-29 on page 5-49).

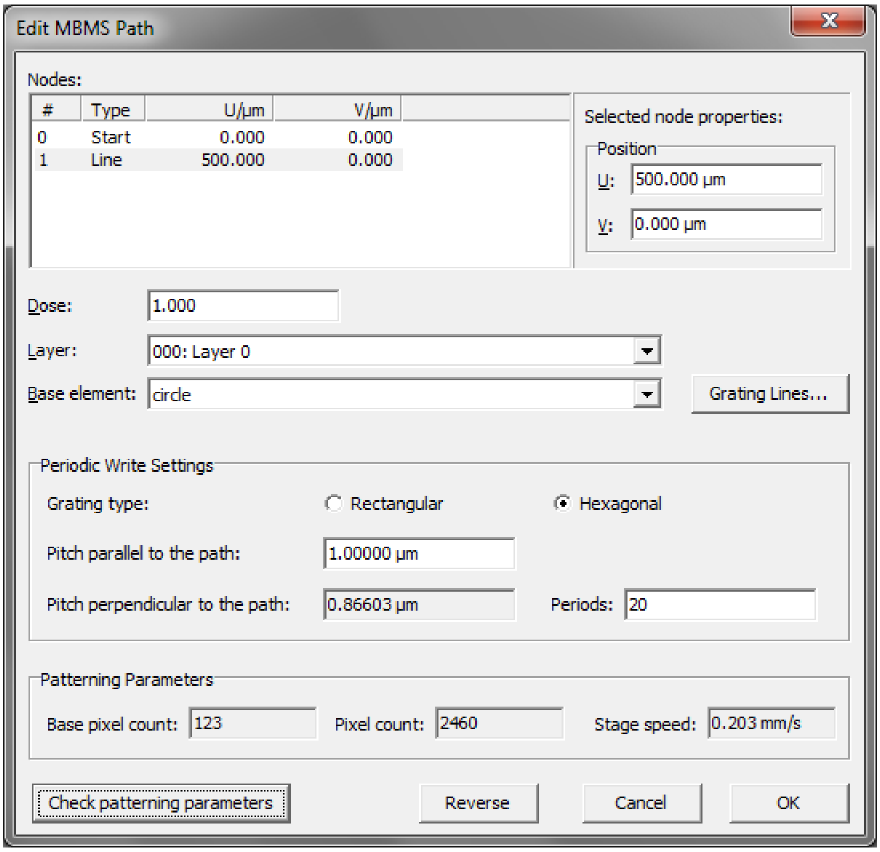


Figure 5-29: **Edit MBMS Path** dialog after pressing of the **Check Patterning parameters** button.

7.16.2 MBMS paths with gratings

In general, MBMS paths with binary gratings are defined as described in the previous section. Furthermore, the base element can be automatically generated within the **Edit MBMS Path** dialog.

- Press the button **Grating Lines…** (⇒ Figure 5-27 on page 5-47). The **Create Grating Line Structure** dialog will be opened.

Grating lines parallel to the stage movement

- Select **Parallel to the stage movement** as direction (⇒ Figure 5-30 on page 5-50).
- Enter the **pitch** and **line width**, either **relative** in percent of the pitch or as **absolute** number.
- Enter the **write period**, after which the patterning is repeated and which is also the length of the box in the base element.

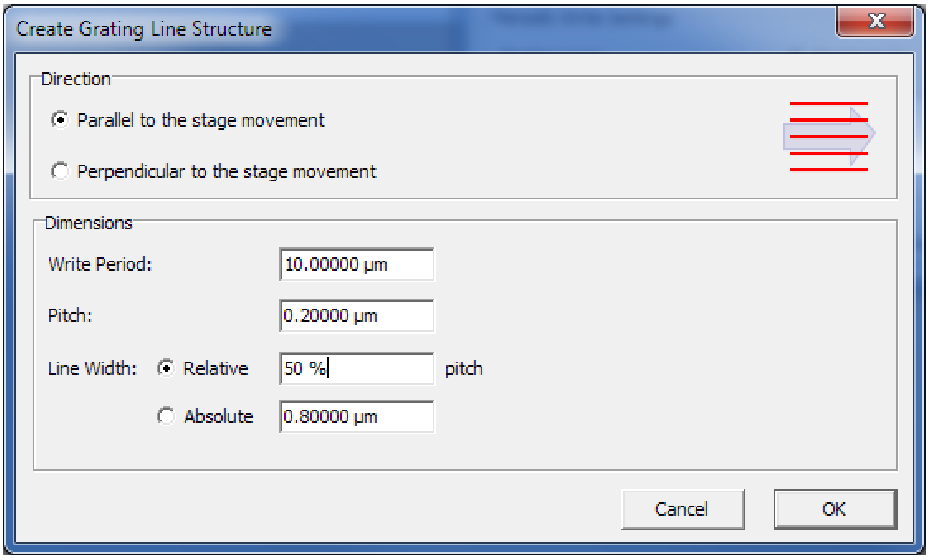


Figure 5-30: **Create Grating Line Structure** dialog with selection **Parallel to the stage movement**.

Creating lines perpendicular to the stage movement

- Select **Perpendicular to the stage movement** as direction (⇒ Figure 5-31 on page 5-50).
- Enter the **pitch** and **line width**, either **relative** in percent of the pitch or as **absolute** number.
- Enter the **write width**, which is also the height of the box in the base element.

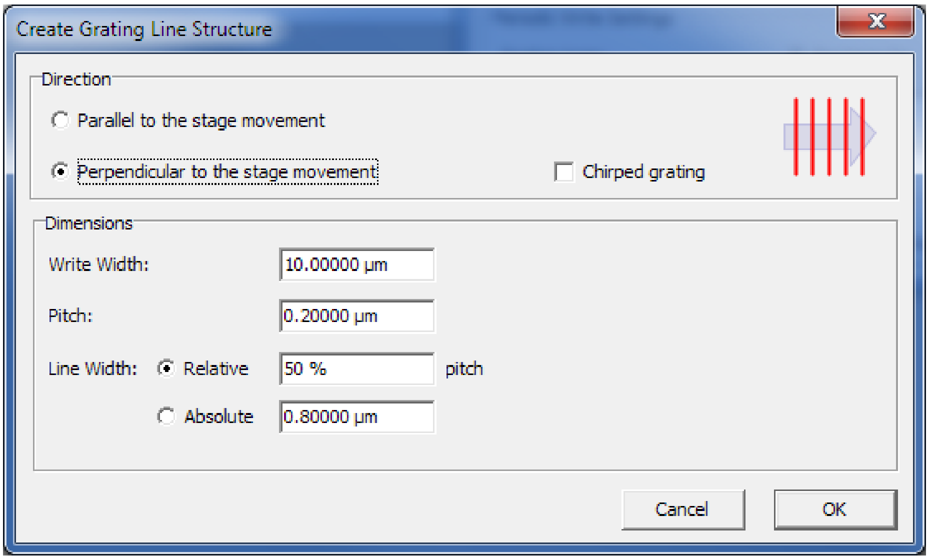


Figure 5-31: **Create Grating Line Structure** dialog with selection **Perpendicular to the stage movement**.

In addition to gratings with a fixed pitch, chirped gratings with a linear variation in the grating pitch can be defined:

- Select **chirped grating** (⇒ Figure 5-32 on page 5-51).
- Enter the start pitch, the end pitch and the pitch step.

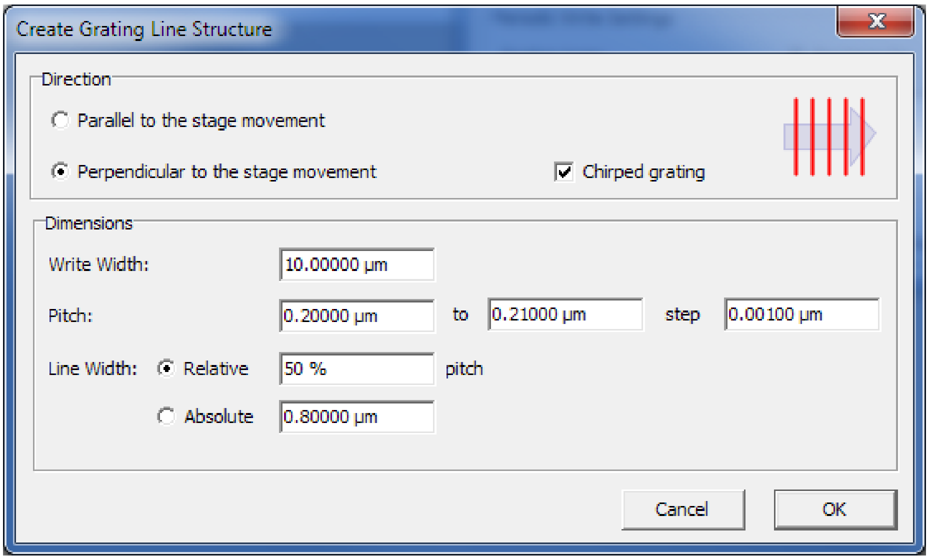


Figure 5-32: **Create Grating Line Structure** dialog with selection **Chirped grating**.

8 Creating matrices

In many cases it is more efficient to duplicate structure elements than to create new ones. In the **GDSII Editor** dialog, you can create matrices consisting of duplicated structure elements. The position of the single elements will be defined via coordinate values in the UV coordinate system. There are two possibilities to create a matrix:

- Using cartesian coordinates to duplicate structure elements horizontally and vertically.
- Using polar coordinates to duplicate structure elements around their center point – e.g. clockwise.

Create a matrix as follows:

▶ Open a structure and change to the **GDSII Editor** dialog.

▶ Create or select a structure element you want to duplicate.

▶ In the main menu bar choose **Modify** ➡ **Duplicate** ➡ **Matrix**.

The **Duplicate Elements** dialog will be opened.

▶ Do one of the following:

- Change to the **Cartesian Coordinates** tab if you want to duplicate the elements horizontally and vertically:

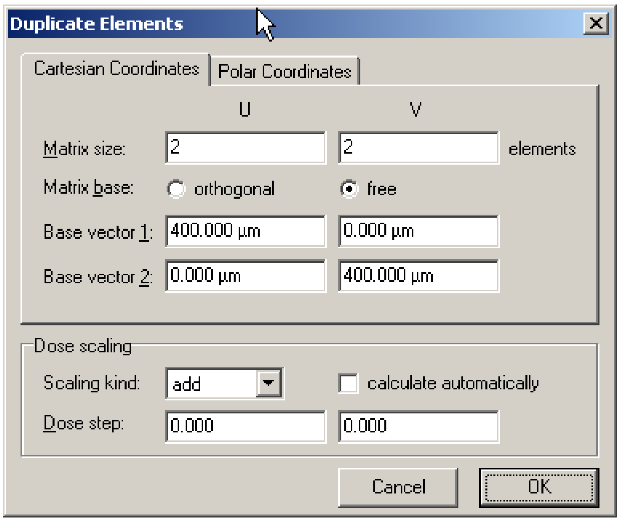


Figure 5-33: Duplicating Elements with cartesian coordinates

For a detailed description of the parameters in this tab refer to (⇒ *Duplicate Elements dialog – Cartesian Coordinates tab* on page 5-122).

- Change to the **Polar Coordinates** tab if you want to duplicate the elements around their center point:

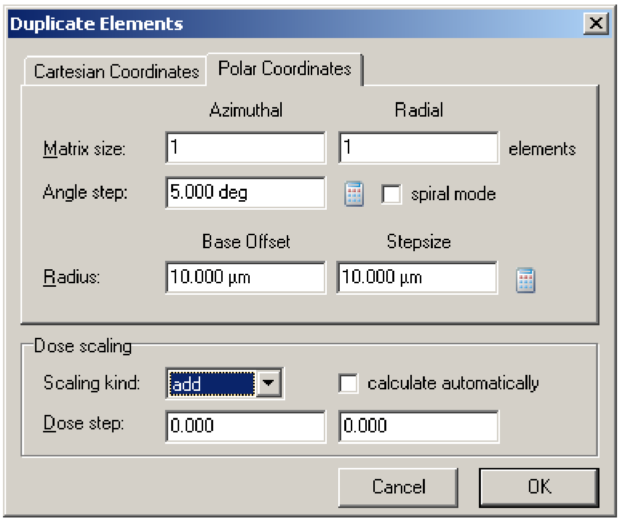


Figure 5-34: Duplicating Elements with polar coordinates

The following table shows graphically, how the parameters affect the structure of the matrix:

Table 5-9: Parameters for matrix creation with polar coordinates

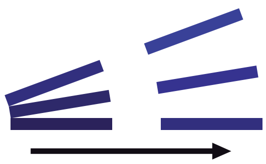
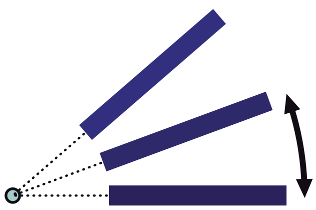
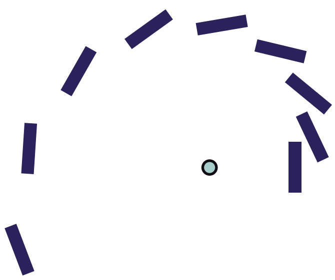
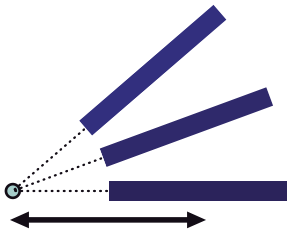
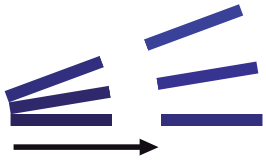
Parameter	Effect on matrix
Matrix size – Azimuthal	
Matrix size – Radial	
Angle step	
spiral mode	

Table 5-9: Parameters for matrix creation with polar coordinates

Parameter	Effect on matrix
Radius – Base Offset	
Radius – Stepsize	

For a detailed description of the parameters in this tab refer to (⇒ *Duplicate Elements subdialog – Polar Coordinates tab* on page 5-123)

- In the **Dose scaling** subset, define the dose assignment for the duplicated elements.

For a detailed description of the parameters in the Dose scaling subset refer to (⇒ *Duplicate Elements subdialog – Dose scaling subset* on page 5-124).

9 Creating mathematically defined groups of curves

You can create groups of curves, which can be specified by flexible mathematical formula. Each curve is defined as an open path, i.e. the connections between adjacent steps are straight lines.

The available mathematical functions and fundamental arithmetic operations are listed below/in section (⇒ *Conventions for mathematical formulae* on page 5-59).

9.1 Creating a group of curves

STEP 1: Creating a single curve

- Select a structure and change to the **GDSII Editor** dialog.
- In the main menu bar choose **Add** ➡ **Group of curves.**

The **Curve Generator** dialog will be opened:

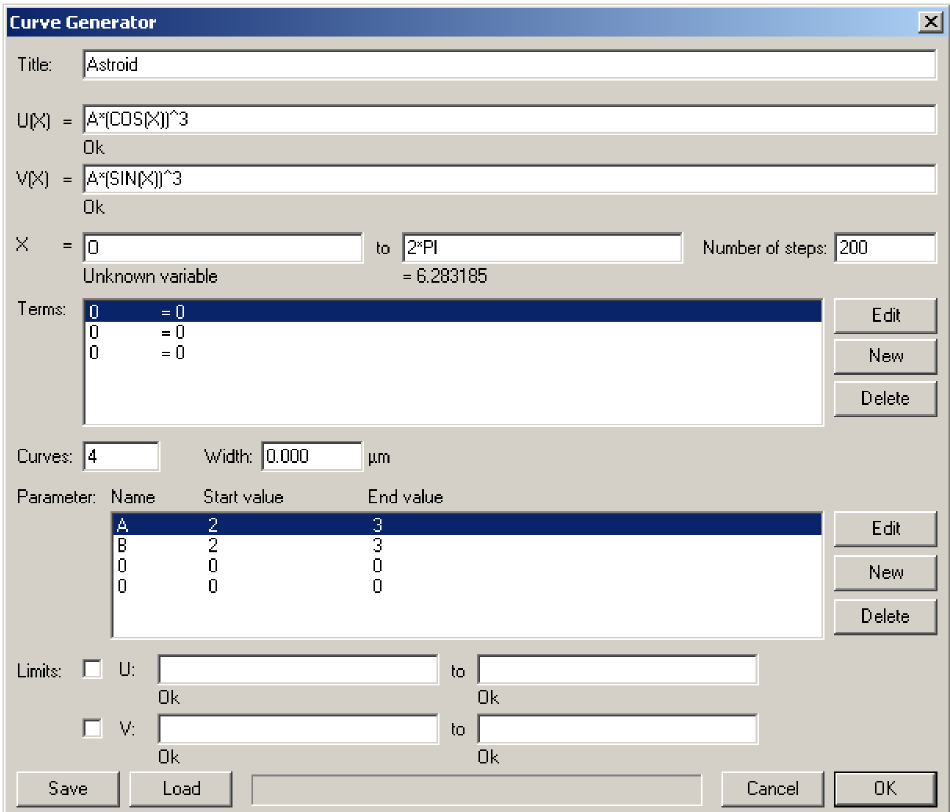


Figure 5-35: **Curve generator** dialog to add mathematically defined curves.

- Enter a suitable **Title** or comment for a newly generated group of curves.
- Define the mathematical formula for the curves:

In general, each curve is described by a set of points and the positions of these points are calculated by the two equations **U(X)** and **V(X)**.

The curve itself is described in a parameter form with the running variable **X**. This variable **X** is increment ed in a range defined by a start value and an end value with the specified **Number of steps**. This **Number of steps** is limited to 2048, i.e. if one enters a number greater than 2048, it will be reduced to 2048 automatically.

If you enter functions or variables, which are not defined, an error message will be displayed below the corresponding input field.

The examples below show the mathematical definitions of two often used curves:

Examples for mathematically defined curves:

Example 1:	A single circle with radius 10 and center coordinates 3/5 can be generated by U(X) = 10*COS(X)+3 V(X) = 10*SIN(X)+5 X = -PI to PI.
Example 2:	A single standard parabola can be generated by U(X) = X V(X) = X^2 X = -4 to 4

▶ If required, define basic function **Terms** to simplify the entry of complex or interlaced equations:

- Next to the **Terms:** text field, choose **New** to create a term or select an existing term and choose **Edit** to modify it.

  The Edit Term dialog will be opened.

- Enter a suitable **Name** for the term to be used within all other equations.

Do not use any of the signs for fundamental arithmetic operations, i.e. + - * / ^. The length of the name is limited to the size of the input field.

- Enter the **Term**.

The term may contain any combination of numbers and predefined mathematical functions as listed in section (⇒ *Predefined mathematical functions* on page 5-60). The length of the term is not limited.

▶ Enter the number of **Curves** to be generated.

If this number is greater than 1, it is a good idea to include **Parameters** within the equations for **U(X)**, **V(X)**, or the **Terms**. Otherwise, all curves will be located one upon the other, which may lead to an overexposure.

▶ Define the **Width** of the paths. A width of zero means, hat the curves are created as single pixel lines.

The curves are generated and stored as structure elements of the type **Open path**.

STEP 2: Creating a group of curves

In order to generate a group of curves and not only a single one, a set of **Parameters** can be defined each with a **Start value** and an **End value**. Within the resulting range each parameter is varied as often as specified. For the variation from curve to curve these parameters must be included within **U(X)**, **V(X)**, or the **Terms**.

- Choose **New** to create a parameter or select an existing parameter and choose **Edit** to modify this parameter.

In both cases you will be assisted by a dialog box to enter the parameter characteristics.

- Within this dialog, enter a suitable name for the parameter but do not use any of the signs for fundamental arithmetic operations, i.e. + - * / ^. Within the same dialog, enter a **Start value** and an **End value** for the parameter. When generating a group of curves, all specified parameters are varied in a linear mode from applying the start values to the first curve and applying the end values to the last curve.

Hence, the step size of the parameters from curve to curve is defined as (**End value - Start value**) / (**Curves - 1**). Both **Start value** and **End value** may contain not only numbers, but also any combination of numbers and predefined mathematical functions as listed below.

- The resulting curves may be limited in their U and V range. Activate the checkbox **Limits** and enter the lower and the upper limits for U and V into the input fields. If the checkbox **Limits** is cleared, the limits settings will be ignored.
- Choose **Save** and **Load** to store and reload all specified settings to dedicated files with the extensions *.MAT.

A group of curves will be generated. (⇒ Figure 5-36 on page 5-59) shows a group of curves generated with the settings above:

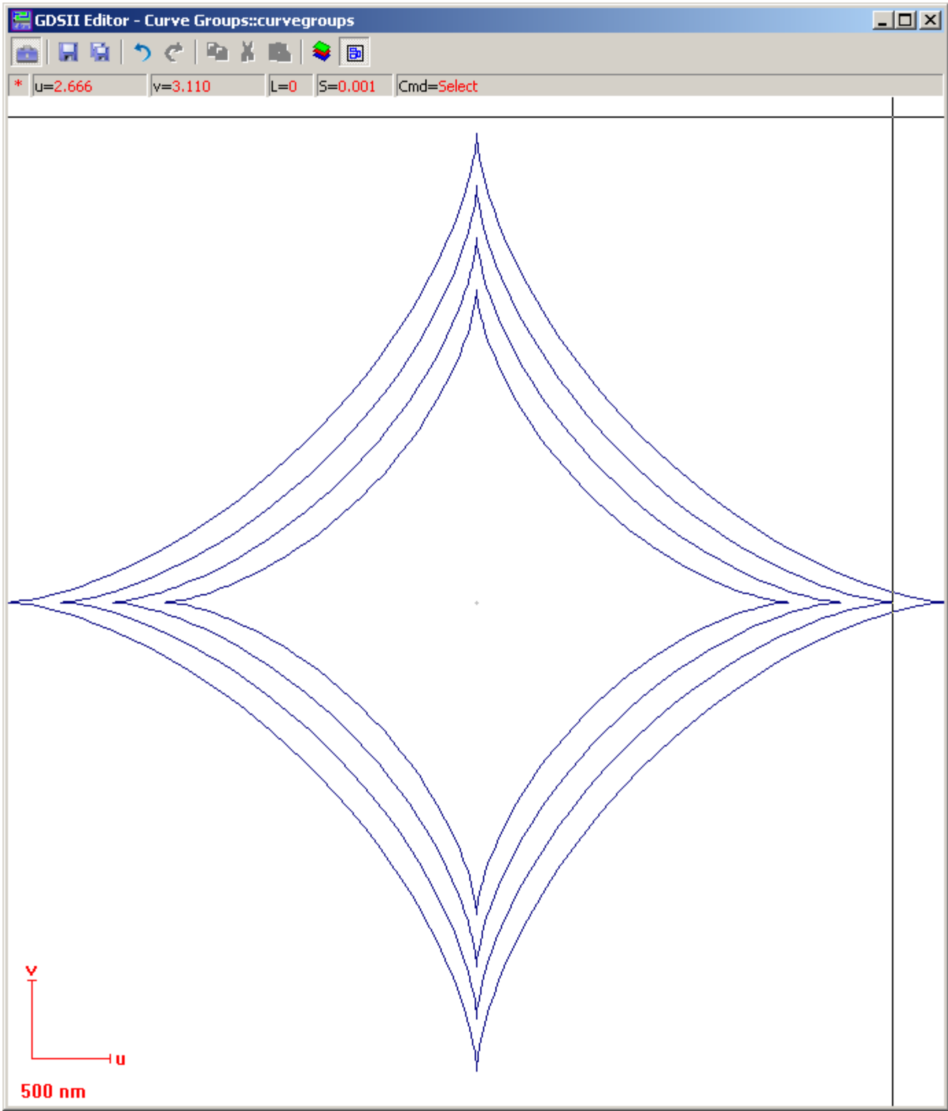


Figure 5-36: Group of curves generated with the settings given above

9.2 Conventions for mathematical formulae

For the mathematical curve generation the normal notation for arithmetic operations is applied (Infix notation) – e.g. 1 + 2 = 3. In order to achieve a good readability, one may add spaces anywhere within the formulas. Arguments for mathematical functions must be enclosed in round brackets – e.g. SIN(X). The circular constant PI = 3.14159... is predefined.

Table 5-10: Fundamental arithmetic functions

Key	Function
+	Addition or positive sign
-	Subtraction or negative sign

Table 5-10: Fundamental arithmetic functions

Key	Function
*	Multiplication
/	Division
^	Power

9.3 Predefined mathematical functions

The following mathematical functions are available to define almost any type of 2-dimensional curves.

Table 5-11: Predefined mathematical functions

Function	Description	Return values
S	Step or Heaviside function	S(X) = 0 for X < 0 S(X) = 1 for X ? 0
R	Ramp	R(X) = 0 for X < 0 R(X) = X for X ? 0
SGN	Signum	SGN(X) = -1 for X < 0 SGN(X) = 0 for X = 0 SGN(X) = +1 for X > 0
ABS	Absolute	ABS(X) = -X for X<0 ABS(X) = X for X ? 0
SQRT	Square Root	SQRT(X) = X^1/2
CBRT	Cube Root	CBRT(X) = X^1/3
EXP	Exponential to base e = 2.71828…	EXP(X) = e^X
LN	Natural Logarithm, base e = 2.71828...	–
LOG	Decimal Logarithm, base 10	–
FAK	Factorial	FAK(X) = X! = 1* 2*3*...*(X-1)*X
RND	Random	X ? RND(X) ? 0 for X < 0 0 ? RND(X) ? X for X ? 0

Table 5-11: Predefined mathematical functions

Function	Description	Return values
SIN	Sine	Argument in radiant
COS	Cosine	Argument in radiant
TAN	Tangent	Argument in radiant
COT	Cotangent	Argument in radiant
ARCSIN	Inverse Sine	Result in radiant
ARCCOS	Inverse Cosine	Result in radiant
ARCTAN	Inverse Tangent	Result in radiant
ARCCOT	Inverse Cotangent	Result in radiant
SINH	Hyperbolic Sine	–
COSH	Hyperbolic Cosine	–
TANH	Hyperbolic Tangent	–
COTH	Hyperbolic Cotangent	–
ARSINH	Inverse Hyperbolic Sine	–
ARCOSH	Inverse Hyperbolic Cosine	–
ARTANH	Inverse Hyperbolic Tangent	–
ARCOTH	Inverse Hyperbolic Cotangent	–

10 Creating hierarchical GDSII Databases

A hierarchical GDSII databases consists of referenced structures. To create a hierarchical GDSII database, it is necessary to

- create single structures which can be used as part of another structure,
- set links in between the structures.

Any structure of the loaded database, except of structures containing the opened structure as a reference, can be referenced. These referenced structures itself of course may contain also references to other structures.

In addition, the referenced structures can be arbitrarily placed, repeated in a matrix form, rotated and scaled within the opened structure.

Proceed as follows to create a hierarchical GDSII database:

▶ In the **GDSII Database** tree view select a structure.

▶ Change to the **GDSII Editor** dialog.

▶ In the toolbox choose .

The **New Structure Reference Properties** dialog will be opened:

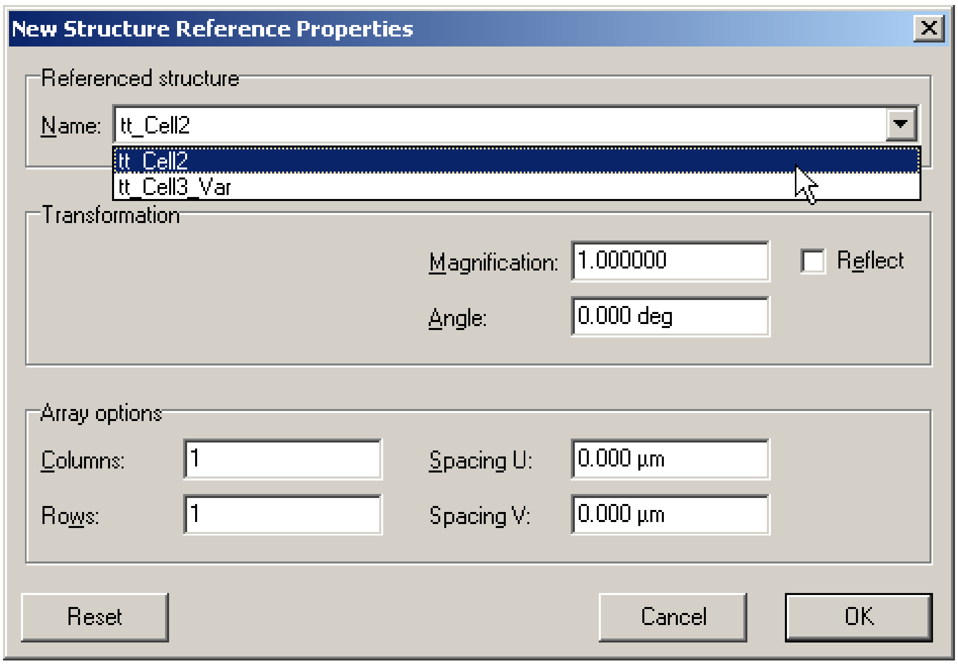


Figure 5-37: **New Structure Reference Properties** dialog

▶ In the **Name** combo box select a structure you want to add.

▶ Make settings for transformation and size of the structure reference:

- Enter the scaling factor for the referenced structure within the field **Magnification**. A factor 1 includes the referenced structure in its original dimensions and for example a factor 0.1 includes it with 1/10 of the original size.
- Select **Reflect** to mirror the element vertically (about the U axis) before angular rotation. For an array reference, the entire array is reflected, with the individual array members rigidly attached.

Magnification operation is applied to the selected structure only, but it is not applied to **Spacing U** and **Spacing V**. Therefore the values for the spacing have to be calculated from the magnified structure

---

- Enter the rotation angle for the entire structure matrix within the field **Angle**. The rotational center is the lower left corner of the first referenced structure, i.e. the lower structure, within the matrix.

The entire matrix is rotated, not the single matrix elements.

- Enter the number of repetitions of the referenced structure along the rotated horizontal structure axis within the field **Columns** and along the rotated vertical structure axis within the field **Rows**.
- Enter the distance in µm between equivalent points of adjacent, magnified structures along the rotated horizontal structure axis within the field **Spacing U** and along the rotated vertical structure axis within the field **Spacing V**.

▶ Choose **OK** to confirm the entries.

The resulting structure reference is indicated as a rectangle, attached to the mouse cursor.

▶ Move the mouse cursor to the target position and confirm by pressing the left mouse button or **Enter**.

The cursor indicates the rotational center of the structure matrix. Hence, this is not equal to the lower left corner of the enclosure, if a rotation angle is applied.

By default each structure reference is represented as a red-framed rectangle indicating the location and size of the field enclosing all referenced structures. The name of the referenced structure and its matrix repetition, i.e. the numbers of rows and columns, is displayed in the center of this rectangle:

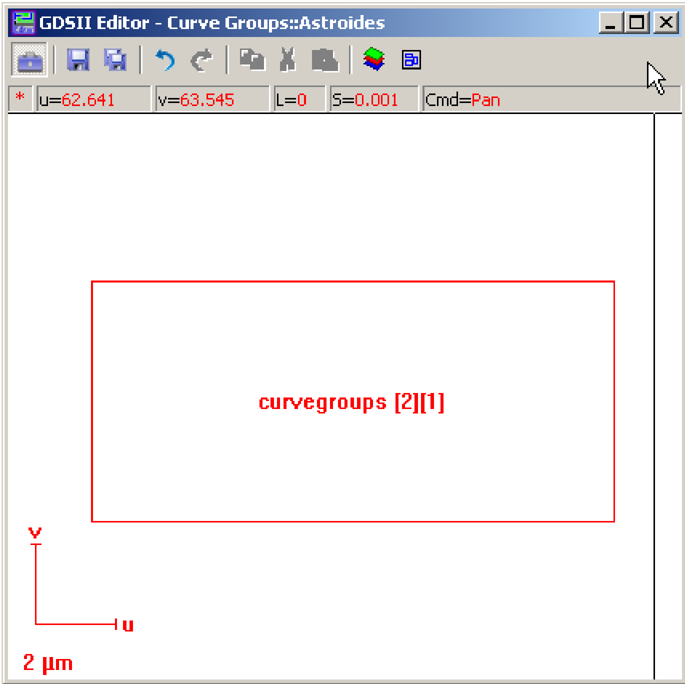


Figure 5-38: Structure reference, indicated by a red frame

Depending on the size of the referenced structure and the zoom factor of the UV window, the red-framed rectangular will be too small to carry the name and matrix repetition. In this case, zoom in the structure as long as this information will be displayed.

▶ For a detailed inspection through the hierarchical levels, choose the menu command **View** ➡ **Hierarchy** (⇒ *View menu* on page 5-94).

The placement of a structure reference can also be done digitally:

– After confirming the entries in the dialog **New Structure Reference Properties** dialog, press any digit key (0 through 9) to call up the dialog box for digital input. The input fields are preset with the coordinates of the current cursor position.
– Modify the target coordinates of the rotational center of the structure matrix.
– Choose **OK** to complete placement.

11 Adding Mark Scans

In the GDSII module it is possible to add and define 1- and 2-dimensional mark scans to a structure in order to align the writefield to the sample during patterning. The mark scans can be performed in a manual or automatic fashion. Adding and modifying of mark scans is only possible in the **GDSII Editor** dialog.

– For more details about writefield alignment and mark scans (⇒ *Writefield Alignment dialog* on page 13-18)
– How to perform a mark scan (⇒ *Markscan Options* on page 11-53)

11.1 Adding Manual Mark Scans – Imagescans

A manual mark scan is a 2-dimensional mark scan, i.e. an imagescan, which can be used for aligning the writefield during a patterning in a semi-automatic fashion. All imgescans are managed in layer **063: Manual marks**.

Proceed as follows to add a manual mark scan to a structure:

▶ In the main menu bar choose **Add** ➡ **Manual mark scan**.

▶ Move the cursor to the desired position within the structure and click the left mouse button.

The **Manual Mark Scan Properties** dialog will be opened. The input fields are preset with the coordinates of the current cursor position.

▶ Modify the corresponding parameters for mark and global settings:

- **Center** defines the position of the mark in U- and V-direction.
- Enter **Size** to modify the area which will be scanned around the center position.
- The number of points used for the scan mark can be defined using **Scan points**.
- A point-by-point averaging is defined by modifying **Average**.

▶ Confirm with **OK**.

The Imagescan will automatically be added to layer **063: Manual marks**.

11.2 Adding Automatic Mark Scans – Linescans

An automatic mark scan is a line scan, which can be used for aligning the writefield during a patterning in an automatic fashion. All linescans are managed in layer **061: Automatic marks**.

Proceed as follows to add an automatic mark scan to a design:

▶ In the main menu bar choose **Add** ➡ **Auto mark scan**.

▶ Move the cursor to the desired position within the structure and click the left mouse button.

The **Auto Mark Scan Properties** dialog will be opened. The input fields are preset with the coordinates of the current cursor position.

▶ Modify the corresponding parameters for mark and global settings:

- **Center** defines the position of the mark in U- and V-direction.
- Select the **Scan direction** to define the direction of the Linescan.
- Enter **Size** to modify the length of the scan.
- **Average size** modifies the width used for averaging.
- The number of points used along the length of the scan can be defined as well as how often scans were recorded for averaging using **Scan points** and **Average points**.
- A point-by-point averaging is defined by modifying **Average**.
- **Options** shows the preset patterning mode.

▶ Confirm with **OK**.

The Linescan will automatically be added to layer **061: Automatic marks.**

11.3 Automatic Mark Scan – Imagescans/Adding Mark Scans from Scan Manager

An automatic imagescan is a 2-dimensional mark scan, which can be used for aligning the writefield during a patterning in an automatic fashion.

Automatic image scans will be added to layer **063: Manual marks**.

Additionally to the imagescans and line scans generated in the GDSII module, you can add mark scans from **Writefield Control** module ➡ **Scan Manager** dialog ➡ **Writefield Alignment Procedures**.

The following table shows the

Table 5-12: Adding mark scans from **Scan Manager**

Scan type	GDSII target layer
GDSII Writefield Mark Scans	
Automatic with images	62
Automatic with Linescans	61
Manual	63
GDSII Layer Based Mark Scans	
Automatic with Linescans	61
Manual with Images	63

For a detailed description of the functions and handling of the different scan types (⇒ *Scan Manager* on page 11-1).

12 Working with Layers

Normally a structure is build during several process steps and the idea is to assign all the structure elements used in one process step to one layer. There are 256 layers available.

The layers **063: Manual marks** and **061: Automatic marks** are predefined for scanning procedures of manual mark scans and automatic mark scans (⇒ *Adding Mark Scans* on page 5-65).

To display only the layer of interest in the **GDSII Editor** or **Viewer** dialog the **GDSII Layer** add-on window can be opened.

To distinguish between the structure elements of various layers it is very convenient to assign each layer different properties like colors and filling patterns. This is done via the **Layer Properties** subdialog. The layer properties are stored in a separate file with extension .LAY. Path and file name are the same as of the corresponding GDSII database file.

12.1 Displaying Layers and selecting structure elements

- In the **GDSII Database** tree view select a structure.
- In the **GDSII Editor/Viewer** dialog, click on .

The **GDSII Layer** add-on window will be opened:

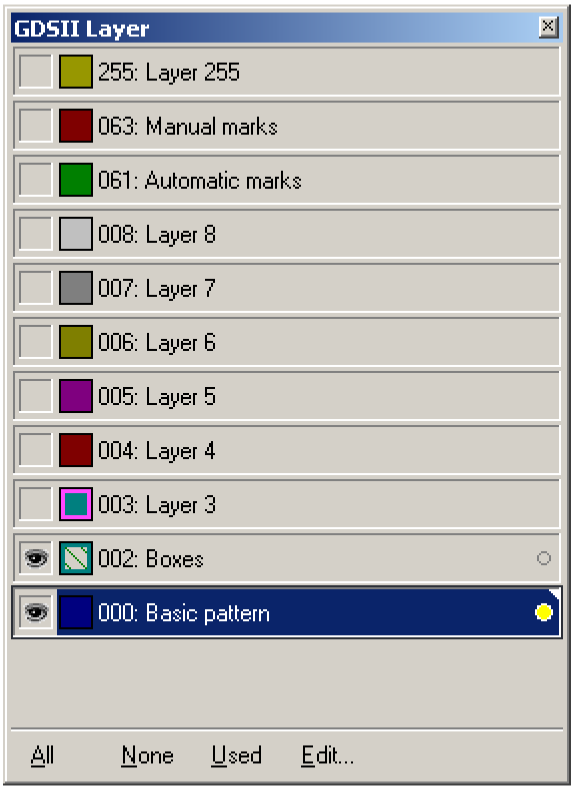


Figure 5-39: **GDSII Layer** add-on window

The dialog shows a list of all defined layers.

- Click on the squares in front of the list entries to display or blind out a layer. The button indicates that the structure elements of the selected layers are displayed.
- Click on a list entry to activate a layer. If a layer is activated, you can create and edit structure elements in it.

Creating, editing and copying of structure elements is only possible in the **GDSII Editor** dialog.

- Click on to select all structure elements of a layer – e.g. to copy them into another layer. The button turns to yellow if one or more structure elements are selected.

For a detailed description of all control elements of the add-on window refer to (⇒ *GDSII Layer add-on window* on page 5-145).

12.2 Defining Layer properties

- In the **GDSII Layer** add-on window choose **Edit**.

  The **Layer Properties** subdialog will be opened:

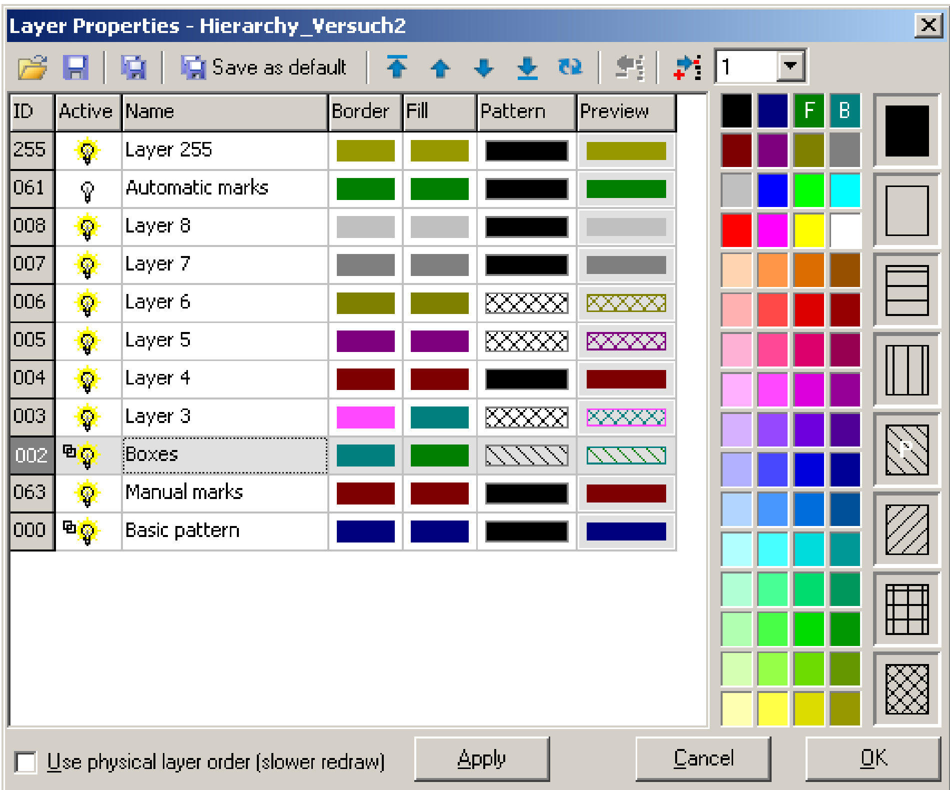


Figure 5-40: **Layer properties** subdialog

- Select a layer from the list.
- Place the cursor in the **Name** input field, type in a proper name and confirm with **ENTER**.
- Assign a **Border** color by clicking in the color table with the left mouse button.

- Assign a **Fill** color by clicking in the color table with the right mouse button.
- Assign a **Pattern** by clicking in the pattern table.

In the **Preview** column a preview of the layout of the structure elements will be displayed.

12.3 Adding and removing Layers

To aid the management of these LAY files, the following commands are available.

- Choose the arrow icons to change the order of the different layers within the list. It is now interesting to interpret the order in the list also a physical order.

Physical order means that a layer at the end of the list is processed first while a layer at the top of the list is processed later. Hence the order is interpreted as a sequence of process steps. To display also the layers in this physical order, i.e. later processed layers overlay previous processed layers, select **Use Physical layer order**.

- Choose to remove the selected layer and choose to add the layer with the number from the list box.
- Click on the bulb in the **Active** column of the list to activate or deactivate the corresponding layer from the **GDSII Layer** add-on window. Two small boxes in the **Active** column indicate if corresponding layer contains GDSII elements.

For a detailed description of all control elements of the subdialog refer to (⇒ *Layer Properties subdialog* on page 5-147).

13 Using Working areas

Normally not the whole GDSII structure is of interest for example to inspect it or to expose it. To expose only a fraction of the whole GDSII structure it is possible to define several areas of interest, so called working areas.

These working areas can be defined for each structure individually. The working areas for all structures are stored in a separate file with the extension .WOR. Path and file name are the same as of the corresponding GDSII database file.

Proceed as follows to create a working area:

- In the **GDSII Database** tree view select a structure.

- In the **GDSII Editor/Viewer** dialog, click on .

  The **Working Areas** add-on window will be opened and the currently defined working areas will be displayed:

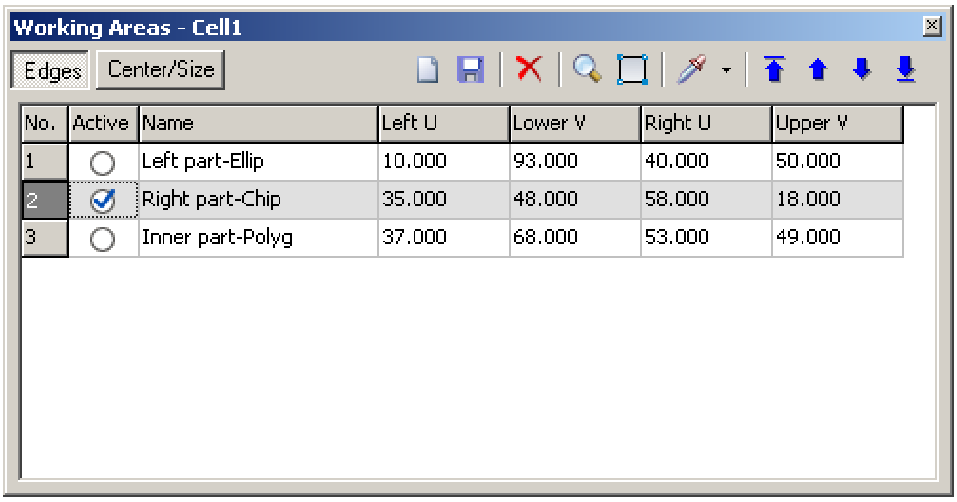


Figure 5-41: **Working Areas** add-on window

- Click on to add a new working area. To change the properties like the **Name** of a working area place cursor in corresponding field to modify values.
- Click into the
- To define the working areas over their edges select button **Edges**, to define working area over its center and size select **Center/Size**.

When selecting a new working area the corresponding area flashes several times within viewer or editor.

- Save changes with the save button. The current active working area can be deleted using the delete button . The selected area can be viewed using the button. To move only the center of the viewpoint choose .
- Choose the pipette button to take over the editor or viewer window coordinates into the currently selected working area. Use the arrow icons to change the order of the defined areas.
- To define a working area as the active one click into the **Active** field and a tick is placed in this field.
- In the menu bar choose **View** ➡ **Working area** to display the working area in the UV window.

  The working area will be indicated by a gray frame.

14 Proximity correction

How to perform a proximity correction refer to the NanoPECS Software description (⇒ *The proximity effect in electron beam lithography* on page 6-2).

15 Working with patterning recipes

Depending on the resist being used and the aim of patterning, the parameters affecting the patterning can vary highly. Therefore, it is helpful to set up datasets with experienced values and store them as so called patterning recipe.

The set up, storing and management of patterning recipes takes place independently from GDSII databases. Therefore, you can set up patterning recipes with experienced or calculated patterning parameter values and assign them to any structure element of a design. The management takes place via the **Recipe Manager** dialog.

To find the right settings, it is important to have some background information about how the GDSII design is transferred into a pattern and how the various patterning parameters affect the patterning process.
For more information about the individual patterning parameters refer to (⇒ *Patterning* on page 14-1).

15.1 Setting up patterning recipes

STEP 1: Creating a new patterning recipe

- In the **Recipe Manager** dialog choose to add a new recipe group to the tree view.
- Name the recipe group significantly: Select the recipe group, press the **F2** key and type in a proper name.
- Confirm with **Enter**.
- Choose to add a new recipe to the selected recipe group.

If you do not select a recipe group, the new recipe will be added to the recipe group you opened last.

The new pattern recipe is set up and will be displayed in the **Recipe Manager** dialog tree view.

STEP 2: Defining patterning attributes

- In the **Recipe Manager** tree view double click on the recipe you want to edit.

The **Edit Patterning Recipe** dialog will be opened:

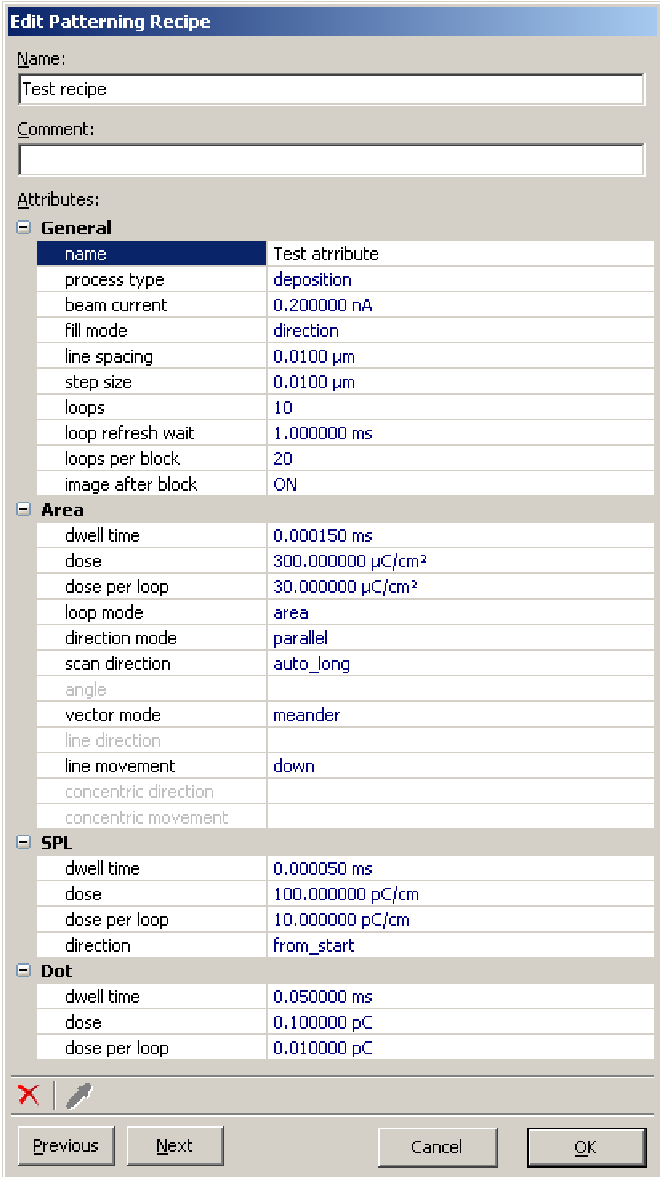


Figure 5-42: **Edit Patterning Recipe** dialog

- In the input fields **Name:** and **Comment:** enter a proper name for the recipe and a further comment if necessary.

- In the **Attributes** list, place the cursor in the field you want to edit.

▶ Depending on the parameter and the input field options

- type in a value or
- select a predefined parameter from a combo box or
- open the **Patterning Attribute Calculator** to calculate the value for a parameter – e.g. for **dwell time** and **dose**. If the calculator is attainable, the ... button will be displayed on the right hand side of the input field.

▶ Press **Enter** or place the cursor into another input field.

The last setting will be saved automatically.

For a detailed description of the parameters refer to (⇒ *Edit Patterning Recipe dialog* on page 5-126).

Alternatively, you can define the patterning parameters of a recipe via the **Patterning Attributes** add-on window. The add-on window displays the patterning parameters assigned to a selected structure (⇒ *Patterning Attributes add-on window (option)* on page 5-138).
Via the pipette button in the **Edit Patterning Recipe** dialog, you can read the settings from this add-on window.

▶ Confirm your settings with **OK**.

STEP 3: Assigning patterning recipes to structure elements

▶ In the **Recipe Manager** dialog tree view select a recipe.

▶ Move the recipe per drag and drop onto a structure element in the UV window of the **GDSII Editor**.

The patterning recipe will be assigned. On the element an arrow button will be displayed indicating the U and V writing direction:

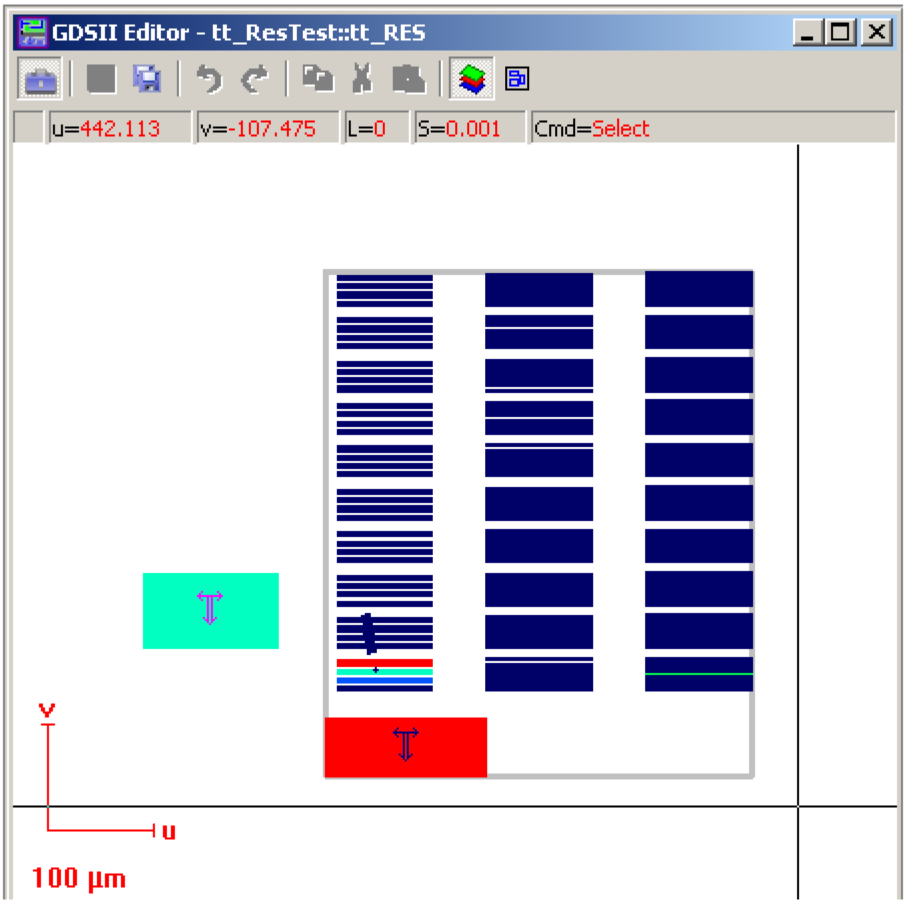


Figure 5-43: Two structure elements with assigned patterning recipes (turquoise, red), directional mode of scanning (**fill mode**)

The arrow button corresponds to the settings made for **fill mode** and the corresponding parameters in the **Edit patterning recipe** dialog.

You can inspect and modify the patterning parameters via the **Patterning Attributes** add-on window (⇒ *Patterning Attributes add-on window (option)* on page 5-138).

STEP 4: Inspecting and modifying patterning attributes

You have two possibilities to inspect and modify a patterning recipe:

- Via the **Edit Patterning Recipe** dialog.

  This dialog serves mainly to set up and assign a patterning recipe. You call up the dialog via the **Recipe Manager** dialog tree view by double clicking a patterning recipe. A modification of the patterning attributes affects all structure elements this recipe is assigned to.

- Via the **Patterning Attributes** add-on window.

  This add-on window serves mainly to inspect and modify single patterning attributes assigned to one structure element. A modification affect only the corresponding structure element but does not affect the patterning recipe itself. You call up the add-on window as follows:

- Select the structure element in the **GDSII Editor**.
- In the toolbox choose to open the Patterning Attributes add-on window.
- Modify one or more parameters and confirm with **Enter**.

New parameters will be applied when you press Enter or select other row in the list view.

The settings overwrite the settings for the patterning recipe assigned to the structure element. If you want to reassign the pattern recipe settings, select a recipe in the Recipe Manager tree view and drag and drop it on the wished structure element in the **GDSII Editor**.

If the structure will be saved after finishing work on it, the individual patterning attributes settings will be saved as well.

C Software Reference

1 GDSII Database

The GDSII Database consists of the following dialogs:

1.1 GDSII Database dialog

In the **GDSII Database** dialog, you can

- call up a database file,
- set up new structures,
- sort structures in the tree view,
- modify structures via a number of GDSII tools.

You open the dialog by clicking on  in the command bar of the NANOSUITE software.

The dialog is divided into two subsets:

- Tool bar with commands and displays to set up, edit and view databases and structures (⇒ *GDSII Database dialog* on page 5-77).
- **GDSII Database** tree view to display the structures and their hierarchical order (⇒ *GDSII Database dialog – tree view* on page 5-78).

1.1.1 GDSII Database dialog

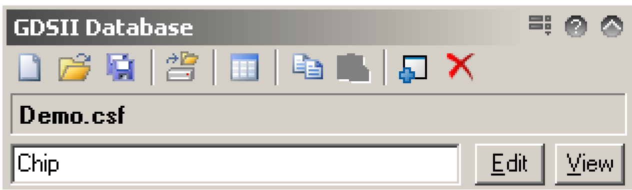


Figure 5-44: **GDSII Database** dialog – tool bar

The control elements have the following functions:

Control element	Function
	Calls up the **GDSII Database** menu (⇒ *GDSII Database menu* on page 80).
	Calls up a file select box to set up an empty GDSII database file. Standard extensions are *.CSF for Calma Stream Format and *.GDS (⇒ *Setting up a GDSII Database* on page 7).

Control element	Function
	Calls up a file select box to open an existing GDSII database file (⇒ *Opening a GDSII Database* on page 7).
	Calls up the **Save GDS II database** dialog to store the currently opened database file under a free selectable name.
	Calls up the **Build Hierarchy** dialog to scan the hierarchical structure of the opened database and to create a corresponding hierarchy file (*.HIR). The file will be stored in your local **<User>** directory ➡ **GDSII**, but it will not be displayed in the tree view. **Note:** It is useful to create a *.HIR file if the design may edit in a third party application not supporting the hierarchy functionality.
	Calls up the **Database Properties** dialog with some statistical and general information about the database – like size, used layers and number of structures.
	Choose this command to copy the selected structure to the Windows clipboard.
	Choose this command to paste a structure from the Windows clipboard into the database.
	Choose this command to create a new structure. After assigning a structure name, the editor will be opened automatically.
	Choose this command to delete the selected structure. **Note:** Only structures that are not referenced within any other structure can be deleted
Edit	Calls up the **GDSII Editor** dialog for editing the selected structure (⇒ *GDSII Editor and Viewer dialogs* on page 89).
View	Calls up the **GDSII Viewer** dialog for viewing the selected structure (⇒ *GDSII Editor and Viewer dialogs* on page 89).

1.1.2 GDSII Database dialog – tree view

The **GDSII database** tree view serves to display the structures and their hierarchical order. The symbols in front of the structures are indicating the function and the status of the structure.

Within the tree view the following symbols could be displayed:

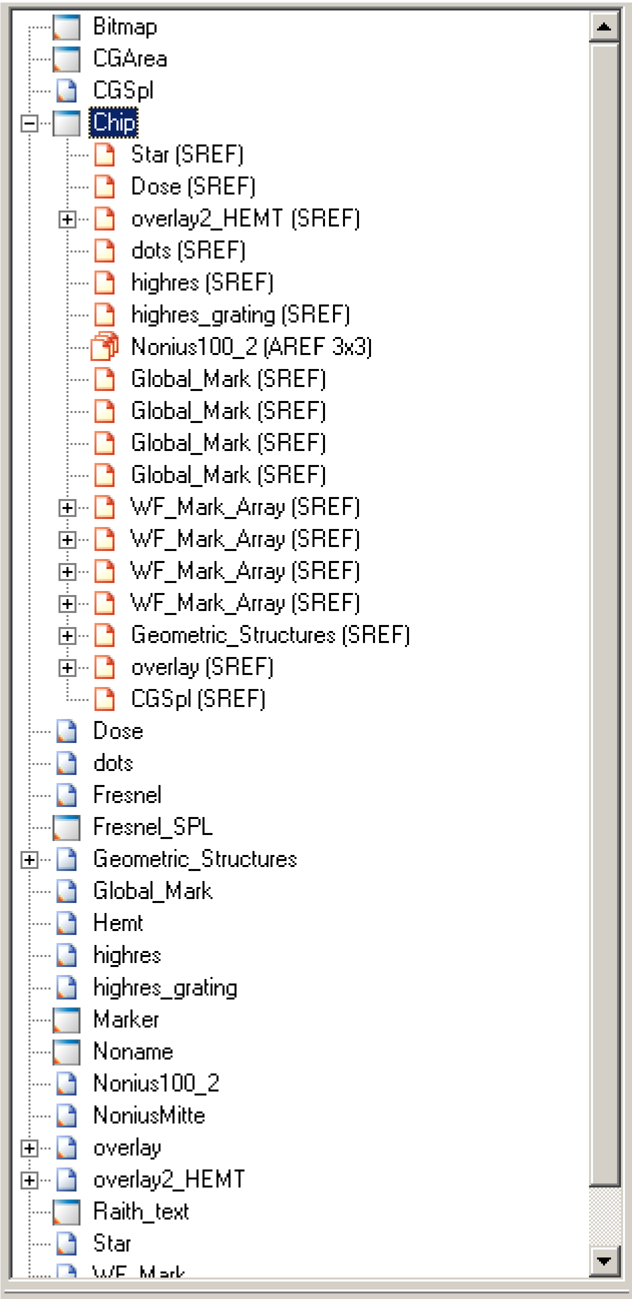


Figure 5-45: **GDSII Database** dialog – tree view

The symbols have the following meaning:

Control element	Function
	Main structure which is not referenced within any other structure.
	Single structure which is referenced within another structure.

Control element	Function
	Single structure referenced in the main structure tree.
	Area structure referenced in the main structure tree.
	Structure hold in computer memory.
	Structure opened in the **GDSII Editor/Viewer** dialog.

1.2 GDSII Database menu

The GDSII Database menu consists of commands and functions to

- administrate the database files,
- edit single structures or structure elements,
- sort the structures in the GDSII database list area.

You open the menu via **GDSII Database** dialog ➡ .

The Menu contains the following Submenus and items:

GDSII Database menu – Submenus	Function
File	Calls up a submenu to administrator the database files and import foreign file formats (⟹ *GDSII Database – File submenu* on page 81).
Tools	Calls up a submenu with commands to optimize single structures and structure elements before patterning (⟹ *GDSII Database – Tools submenu* on page 82).
Edit	Calls up a submenu with commands to edit and view the structure (⟹ *GDSII Database – Edit submenu* on page 83).
Sort structures	Calls up a submenu to sort the structures in between the **GDSII database** tree view (⟹ *GDSII Database – Sort structures submenu* on page 84).

1.3 GDSII Database – File submenu

You open the menu via **GDSII Database** dialog ➡ ➡ **File**.

The Menu contains the following Submenus and entries:

GDSII Database menu – Submenu File	**Function**
New...	Calls up a file select box to set up an empty GDSII database file. Standard extensions are *.CSF for Calma Stream Format and *.GDS (⇒ *Setting up a GDSII Database* on page 7).
Open...	Calls up a file select box to open an existing GDSII database file (⇒ *Opening a GDSII Database* on page 7).
Save as...	Calls up the **Save GDS II database** dialog to store the currently opened database file under a free selectable name.
Build Hierarchy	Calls up the **Build Hierarchy** dialog to scan the hierarchical structure of the opened database and to create a corresponding hierarchy file (*.HIR). The file will be stored in your local **<User>** directory ➡ **GDSII**, but it will not be displayed in the tree view. **Note:** It is useful to create a *.HIR file, if the design may edit in a third party application not supporting the hierarchy functionality.
Close	Closes the currently opened database file.
Database Properties…	Calls up the **Database Properties** dialog with some statistical and general information about the structure.
Load *.ELM	Calls up the **Load Proxy Element File** dialog to open/ import a PROXY Data Exchange File.
Load *.ASC	Calls up the **Load ASCII Element File** dialog to open/ import an ASCII Polygon Data File (⇒ *Importing files in ASCII format* on page 8).
Save *.ASC	Calls up the **Save ASCII Element File** dialog to save an ASCII Polygon Data File (⇒ *Saving files in ASCII format* on page 13).

GDSII Database menu – Submenu File	**Function**
Load *.DXF	Calls up the **Load AutoCAD DXF File** dialog to open/ import an AutoCAD Data Exchange File (⇒ *Importing files in DXF format* on page 13).
Load *.CIF	Calls up the **Load Caltech Intermediate Format File** dialog to open/import a Caltech Intermediate Format (⇒ *Importing files in CIF format* on page 17).

1.4 GDSII Database – Tools submenu

The **Tools** submenu contains commands to optimize single structures and structure elements for patterning.

You open the menu via **GDSII Database** dialog ➡ ➡ **Tools**.

All commands in this submenu overwrite the original structure. Therefore, work on a copy to maintain the original.

The commands have the following functions:

GDSII Database menu – Submenu Tools	**Function**
Overlap removal	Removes overlapping elements within the selected structure (⇒ *Removing overlaps* on page 21). The command checks if there are overlapping elements. If so, then one of the two elements will be cut to size so that it just contacts the other one. **Note:** This command is not removing overlaps within different structure references.
Layer removal	Deletes complete layers.
Sort elements	Calls up the **Sort elements** subdialog to change the order of all elements within the selected structure (⇒ *Sorting elements* on page 23). **Note:** This command is not sorting within different structure references.

GDSII Database menu – Submenu Tools	**Function**
Unify structures	Resolves all references within the selected structure in order to get one large flat structure (⇒ *Removing GDSII hierarchy* on page 19). The result is stored in a new GDSII database. **Note:** Not all GDSII programs from other vendors support hierarchical files. Use this command to import GDSII files edited in the NANOSUITE software into those programs.
Convert userdefs	Converts Raith specific elements like text, circles etc. to common CAD elements (⇒ *Converting to standard GDSII* on page 17).
Merge elements	Calls up the **Merge elements in selected structure** dialog to remove gaps between structure elements (⇒ *Merging GDSII elements* on page 25). **Note:** This command is not merging elements within different structure references.
Join sorted elements	Calls up the **Join sorted elements in selected structure** dialog to join in the minimum two GDSII elements to one larger GDSII element (⇒ *Joining GDSII elements together* on page 27). For this the elements have to be sorted by location before.
Boolean operations	Calls up the **GDSII Boolean Operations** dialog to create complex structure elements by applying Boolean operations (⇒ *Using Boolean operations* on page 40).
Set dose factor	Calls up the **Set dose factor** dialog to assign a consistent dose factor to all structure elements of the selected or all structures of the GDSII database file.

1.5 GDSII Database – Edit submenu

The **Edit** submenu contains commands to edit and view the structure elements.

You open the submenu via **GDSII Database** dialog ➡ ➡ **Edit**.

The commands have the following functions:

GDSII Database menu – Submenu Edit	**Function**
View	Calls up the **GDSII Viewer** dialog. This viewer is used for inspection purposes, especially to resolve hierarchical structures.
Edit	Calls up the **GDSII Editor** dialog. This editor is used to design new structures and to modify existing structures. It includes the functions of the **GDSII Viewer** dialog.
Delete...	To delete the selected structure. **Note:** Only structures can be deleted, that are not referenced within any other structure.
Duplicate...	To make a copy of the selected structure within the same database. The copy will be named **Copy_of_**<**name**> automatically, where <**name**> is the name of the original structure. **Note:** It is also possible to copy a structure in between the tree view via drag and drop.
Rename	To change the name of the selected structure.
Properties...	Calls up the **GDSII Structure Properties** dialog to display some statistical and general information about a selected structure – like stream file size, boundaries, kind and number of integrated structure elements.
New...	To create a new structure. After assigning a structure name, the **GDSII Editor** dialog will be opened automatically.
Copy	To copy the selected structure to the clipboard. Afterwards it can be pasted within another database by choosing the command **Edit** ➡ **Paste**.
Paste	Choose this command to paste the copied structure within a database.
Clear clipboard	Choose this command to clear the clipboard and to free the memory after storing large structures in the clipboard.

1.6 GDSII Database – Sort structures submenu

The **Sort structures** submenu contains commands to define the order in which the structures will be displayed in the **GDSII Database** tree view.

You open the submenu via **GDSII Database** dialog ➡ ➡ **Sort structures**.

The commands have the following functions:

GDSII Database menu – Submenu Sort structures	**Function**
by type and name	Sorts structures first by the type – i.e. main structures first – and alphabetically.
by name	Sorts structures alphabetically.
by file order	Shows structures in the order they are stored in the database.

1.7 DXF Import Options subdialog

In the **DXF Import Options** subdialog you can set parameters to control the import process of DXF files (⇒ *Importing files in DXF format* on page 5-13).

You open the submenu via
GDSII Database dialog ➡ ➡ **File** ➡ **Load *.DXF**.

How to import a DXF file refer to (⇒ *Importing files in DXF format* on page 13).

The import filter offers the possibility to

- transfer a hierarchical structure created in the origin AutoCAD file,
- import defined layers and assign them to a GDSII layer,
- define settings for some individual structure elements.

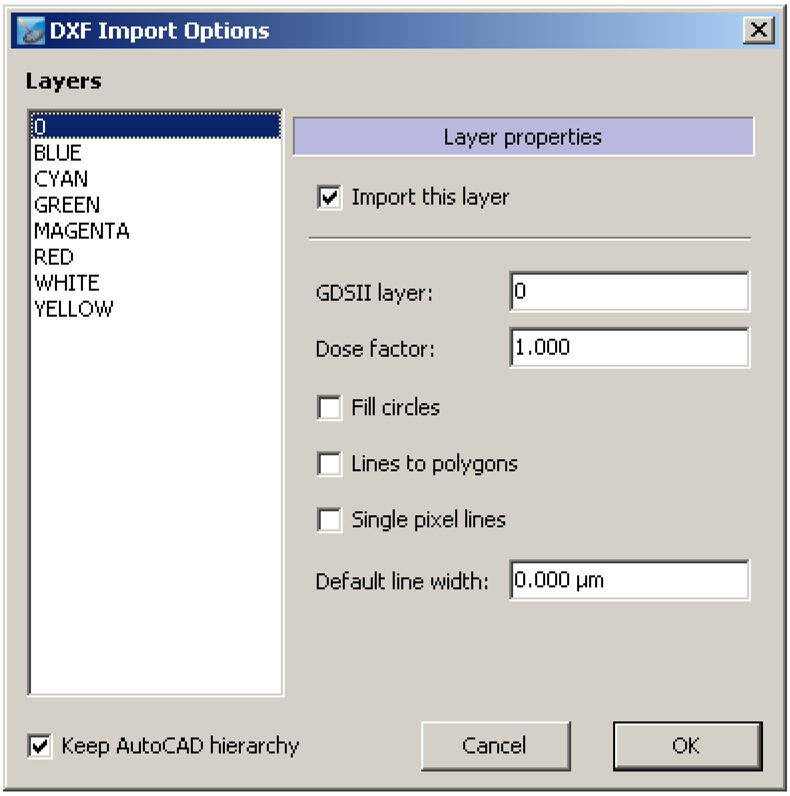


Figure 5-46: **DXF Import Options** subdialog

The control elements have the following functions:

Field/Button	Function
Layers	To select the names of those CAD layers import options have been defined for. **Note:** If the CAD file contains several layer definitions, these may be assigned to different GDSII layers. In addition, the options under Layer properties – e.g. the dose factor – can vary for each DXF layer.
Import this layer	To select if the selected layer is to be imported.
GDSII layer:	To enter the GDSII destination layer for the selected CAD layer.
Dose factor	To enter the common dose for all elements of the selected CAD layer. **Recommendation:** Enter 1.000 = 100% of clearing dose.
Fill circles	To select, if circles are to be imported as filled polygons or lines, i.e. borders only.
Lines to polygons	To toggle, if polylines are to be imported as filled polygons or lines only.
Single pixel lines	To toggle, if the line width of all imported lines is set to 0 or to the line width defined in the field **Default line width:**.
Default line width	To enter the line width, valid for all lines, which have no line width definition in the DXF format. **Note:** If line width is included in the CAD definition, this parameter is not applied. If **Single pixel lines** are selected, this input field is disabled and all imported lines are set to 0.
Keep AutoCAD hierarchy	To keep the hierarchical structure of the origin AutoCAD file.

1.8 Overlaps out subdialog

In the **Overlaps out** subdialog, you can remove overlapping areas of structure elements in between one structure and create a backup copy of the GDSII database file.

You call up the dialog via GDSII Database dialog ➡ ➡ **Tools** ➡ **Overlap removal**.

How to remove overlaps refer to (⇒ *Removing overlaps* on page 5-21).

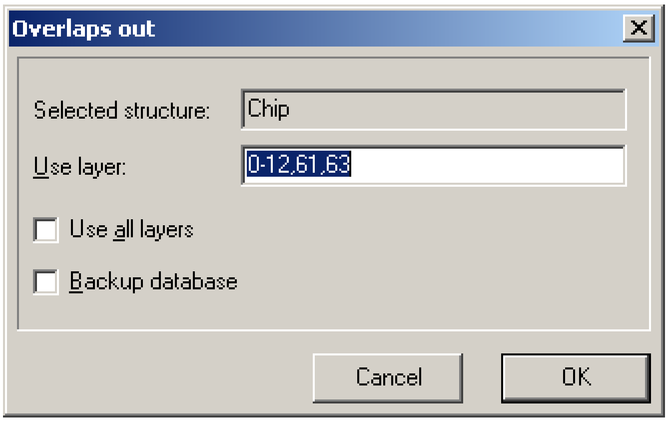


Figure 5-47: **Overlaps out** subdialog

The fields, elements and buttons have the following functions:

Field/Element/ Button	Function
Selected structure	Displays the currently selected structure.
Use layer	To enter the layers, in which overlays of structure elements are to be removed.
Use all layers	To activate, if overlays of structure elements are to be removed in all layers of the selected structure.
Backup database	Creates a backup copy of the currently selected database file. **Note:** The backup copy will be indicated with the following syntax: **<filename>_$$1.csf** If you create more than one copy of the same database file, the extension will be count up: _$$1.csf, _$$2.csf, _$$3.csf etc.

2 GDSII Editor and Viewer

The GDSII Editor and Viewer dialogs will be explained in the following sections.

2.1 GDSII Editor and Viewer dialogs

In the **GDSII Editor**/**Viewer** dialog, you can

- inspect the structures and structure elements of a design,
- create and modify structure elements,
- add structure references to set up a hierarchical GDSII database.

You call up the dialogs via the **GDSII Database** dialog ➡ **Edit**/**View**.

The dialogs are divided into three subsets:

- Tool bar with commands for saving, basic handling of structure elements, and add-on window call up (⇒ *GDSII Editor and Viewer dialog – Tool bar* on page 5-89).
- Status bar with information about the current UV position of the cursor, active layer, grid, and currently active command (⇒ *GDSII Editor and Viewer dialog – Status bar* on page 5-90).
- UV window with the UV coordinate system, in which the design is displayed (⇒ *UV windows* on page 4-4).

2.1.1 GDSII Editor and Viewer dialog – Tool bar

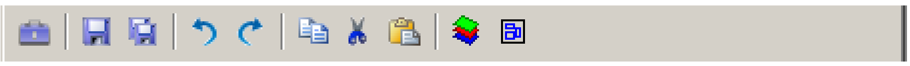


Figure 5-48: **GDSII Editor/Viewer** – Tool bar

This tool bar contains general tools which are common to all UV windows of the NANOSUITE software as well as context specific tools. The following tools are specific to the **GDSII Editor**/**Viewer** dialogs:

Button	Function
	Opens the toolbox (⇒ *Toolbox* on page 135).
	Opens the **GDSII Layer** dialog (⇒ *GDSII Layer add-on window* on page 145).
	Opens the **Working Areas** dialog (⇒ *Working areas add-on window* on page 143).

The tools which are common to all UV windows of the NANOSUITE software are described in (⇒ *UV windows* on page 4-4).

2.1.2 GDSII Editor and Viewer dialog – Status bar

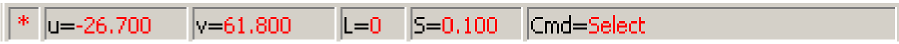


Figure 5-49: **GDSII Editor/Viewer** – Status bar

By default, the following information is displayed in this status bar. Additional information, e.g. rotation angle, are displayed when using specific commands.

Control element	Function
*	Indicates that the last modifications are currently not saved.
u=/v=	Indicates the actual U and V positions of the design cursor within the UV window. The values are given in µm with a resolution of 1 nm.
L=	Indicates the number of the layer currently activated for editing (editor only). To change the layer for editing, click the layer indicator to open the **Preset Layer** dialog.
S=	Indicates the currently used design grid. Within the editor the design grid defines a grid, which is used to place elements. To change its step size, click the step size indicator to open the **Edit Design Grid** dialog. Minimum step size is 0.001 µm, i.e. 1 nm, maximum step size is 100 µm.
Cmd=	Indicates the currently active command – e.g. **Add rectangle.**

2.2 GDSII Editor and Viewer menus

The GDSII editor and viewer are controlled via the main menu bar of the NANOSUITE software. The commands in the menus are context-sensitive depending, if the editor or viewer is currently active.

In the following an (E) or a (V) indicates that this command is only available in the **GDSII Editor** or in the **GDSII Viewer** dialog respectively.

2.2.1 File menu

In the following an (E) or a (V) indicates that this command is only available in the **GDSII Editor** or in the **GDSII Viewer** dialog respectively.

The menu items have the following functions:

File menu item	**Function**
Close	– Closes the **GDSII Editor** dialog. – Closes an opened and currently active **GDSII Viewer** dialog.
Close all (V)	Closes all opened **GDSII Viewer** dialogs.
Save (E)	Saves the current structure without closing the **GDSII Editor**.
Save and close (E)	Closes the **GDSII Editor** and saves the current structure automatically. **Note:** If changes have been made in referenced structures, the hierarchy information of the database will be automatically scanned and updated.
Save as ... (E)	To save a structure under a different name.
Save viewer image as... (V)	To save the actually viewed area in Tagged Image File Format (*.TIFF) or in Bitmap format (*.BMP).
Print...	To print out the actually viewed area of the **GDSII Editor/ Viewer** dialog.

2.2.2 Edit menu

In the following an (E) or a (V) indicates that this command is only available in the **GDSII Editor** or in the **GDSII Viewer** dialog respectively.

The menu items have the following functions:

Edit menu item	**Function**
Undo, Redo (E)	Choose these commands to undo or redo the last action. Internally a list of all changes is managed meaning that the number of actions stored is limited only by memory.

Edit menu item		Function
Cut (E)		Choose this command to cut out the selected elements and copy them into the Microsoft Windows clipboard for further usage, especially to paste them into another structure or database.
Copy (E)		Choose this command to copy the selected elements into the Microsoft Windows clipboard for further usage, especially to paste them into another structure or database.
Paste (E)		Choose this command to paste the elements actually located in the Microsoft Windows clipboard into the actually opened structure.
Delete (E)		Choose this command to delete a selected structure element.
Select (E)		To select one or more structure elements within a structure. The selection of at least one element within a structure is necessary for most of the Edit commands. All currently selected elements are characterized by having yellow points at the corners. The number of selected elements will be displayed in the status bar on bottom of the Program Desktop. After choosing the Select command, a cascading menu appears offering additional functions:
	All	To select all elements of the structure irrespective of their layer assignments.
	Dose factor...	To select elements within the specified range of dose factors.

Edit menu item		Function
	In	To select all elements within the active layer located inside a box to be dragged. After choosing the command, point to any corner of the area covering all elements to be selected and click the left mouse button. Then point to the opposite corner of that area and click the left mouse button again. Pressing the **Shift** key while doing so, the first point defines the center and not the first corner of the rectangle. Pressing the **Control** key while doing so, the field of selection will become quadratic. **Note:** As a shortcut, do not choose this command, but point directly to any corner of the area covering all elements to be selected, press and hold the **Shift** key and click the left mouse button. Then proceed as described.
	Layer	To select all elements within the active layer.
	New	To select all elements within the active layer, which have been added after saving, respectively after opening the **GDSII Editor** dialog.
	Next	To select the next element within the active layer of the structure. The sequence for selecting next elements is the sequence of these elements in the design. The operation is cyclic. Starting element is the last element that has been selected before choosing this command.
	Previous	To select the previous element within the active layer of the structure. The sequence for selecting previous elements is the inverse sequence of these elements in the design. The operation is cyclic. Starting element is the last element that has been selected before choosing this command.
	Structure reference	To select a structure reference by one of the following selection modes: – **All** selects all structures. – **At** selects all structures, which cover the point defined by a subsequent mouse click. – **In** selects all structures, which are located inside a box to be dragged.

Edit menu item	Function
Structure reference by name	To select structure references specified by the name of a referenced structure. After choosing one of the following cascading commands, a structure can be specified by means of a dialog showing all existing structures. – **All** selects all structure references containing the specified structure. – **At** selects a structure reference containing the specified structure by a subsequent mouse click. – **In** selects all structure references containing the specified structure are selected by a box to be dragged.
Unselect (E)	Consists of the same set of commands as the **Select** command. The target is remove element's selection for desired elements or substructures.
Adjustment (E/V)	Command common to all UV windows and described in (⇒ *UV windows* on page 4-4)
Measure (E/V)	Command common to all UV windows and described in (⇒ *UV windows* on page 4-4)

2.2.3 View menu

In the following an (E) or a (V) indicates that this command is only available in the **GDSII Editor** or in the **GDSII Viewer** dialog respectively.

The menu items have the following functions:

View menu item	Function
Hierarchy	Choose this command to resolve hierarchy levels of the opened GDSII structure. After choosing this command, a drop-down menu appears offering additional commands. A checkmark indicates the resolved hierarchy level.
Increase	To increase the hierarchy by one level.
Decrease	To decrease the hierarchy by one level.
0 – 8	To resolve the hierarchy to the specified level, where **0** is the highest level.

View menu item		Function
	Auto	To resolve the hierarchy as a function of the current viewed area.
	Max	To always resolve to the highest hierarchy level. **Note:** Sometimes, the highest hierarchy level may contain so many elements that it would take minutes to build up the entire structure. In this case you may cancel this operation by cancel the redraw which can easily be achieved by pressing the **Esc** key on the keyboard.
Redraw		Command common to all UV windows (⇒ *UV windows* on page 4-4).
Zoom		A cascading menu opens with additional commands from which the most are common to all UV windows (⇒ *UV windows* on page 4-4). Additionally, there is the following command attainable: **Active working area** to display the currently defined working area.
Design grid		Choose this option to display the design grid as small light gray dots, which are related to the actually chosen cursor step size. **Note:** Creation of new elements or modification of elements is possible only within this design grid, irrespectively whether it is displayed or not. Depending on the relation between displayed area and chosen step size, it may happen that not all of the grid dots will be displayed.
Patterning grid		Choose this option to display the patterning grid, i.e. the division of the active working area into stitch fields. **Note:** This option is only available when having the corresponding license.
Working area		Choose this option to display the active working area, i.e. the rectangle used for patterning.
Layers		Choose this command to display the layer selection add-on window. It will stay on top of all activated windows until it is closed. It allows the selection of layers to be displayed.
Overlays		Command common to all UV windows (⇒ *UV windows* on page 4-4).

View menu item	**Function**
Patterning attribute editor (E)	Calls up the **Patterning Attributes** add-on window (⇒ *Patterning Attributes add-on window (option)* on page 138)
Toolbox	Calls up the toolbox with the most important commands for creating and modifying structure elements (⇒ *Toolbox* on page 135).
Transformation (E)	Calls up the **Transform** add-on window to move, scale and rotate a structure element (⇒ *Transform add-on window* on page 140).
Align and distribute (E)	Calls up the **Align and distribute** add-on window to align and distribute selected structure elements (⇒ *Align and distribute add-on window* on page 141).
Working area editor	Calls up the **GDSII Working** area add-on window to define fractions of a structure which are to be exposed (⇒ *Working areas add-on window* on page 143, ⇒ *Using Working areas* on page 70).

2.2.4 Options menu

In the following an (E) or a (V) indicates that this command is only available in the **GDSII Editor** or in the **GDSII Viewer** dialog respectively.

The menu items have the following functions:

Options menu item	**Function**
Background	Choose this command to select a background color for the UV window of the **GDSII Editor/Viewer** dialog. A checkmark indicates the currently selected color.
Backside view (V)	Choose this option to mirror the layout horizontally and to display the layers in reverse order. This becomes necessary, if the sample is viewed from the bottom side.

Options menu item	**Function**
Fill	Choose this option to indicate filled structure elements, i.e. boxes, polygons, etc. A checkmark shows whether this option is in effect or not. To remove the checkmark, choose this option once again. If this option is not in effect, elements are indicated by frame, only. The fill color can be defined for each layer individually. **Note:** If an element is covered completely by another element, it is no longer visible after choosing this option. To overcome this effect, switch the fill mode in the **GDSII Viewer/Editor** dialog via main menu command **Options** ➡ **Fill**. After closing and opening the editor or viewer window once again, choose **Fill**. Hatching will now indicate the filled structure elements.
Physical layer order	Choose this command to display the different layers in their physical order – i.e. layers which are deeper are displayed first. The layer order is defined in the **Layer Properties** subdialog (⇒ *Layer Properties subdialog* on page 147).
Same aspect ratio	Command is common to all UV windows (⇒ *UV windows* on page 4-4).
MBMS Periodic Cells	Choose this option to show the base pattern in MBMS elements.
Layer editor…	Calls up the **Layer Properties** subdialog for defining the layer properties (⇒ *Layer Properties subdialog* on page 147).
Design grid …(E)	Calls up the **Edit Design Grid** dialog to display the design grid and define the stepsize of the grid. The design grid will be displayed as small light gray dots. **Note:** The creation of new or modification of selected elements is only possible within the design grid, irrespectively whether it is displayed or not. Depending on the relation between the displayed area and the chosen step size of the grid, it may happen that not all of the dots will be displayed.
Show dose	Choose this option to fill the elements with a color indicating the dose factor. The layer assignment is indicated by the corresponding frame colors.

Options menu item	**Function**
Dose colors	Choose this command to open the **Dose Colors** dialog for defining the color assignment of the dose scale. **Note:** This command is only available if the dose distribution is displayed via the command **Show dose**. The relation between colors and dose scale can be defined by editing the related input fields. Use the pipette button to insert dose values from the design into the dose range.
Show patterning attributes	Choose this command to display special symbols on structure elements, representing which kind of patterning attributes or patterning recipe are assigned to these elements. **Note:** The symbols can only be displayed, if patterning attributes or a patterning recipe are assigned to one or more structure elements (⇒ *Working with patterning recipes* on page 72).
Show video	Choose this option to show the currently opened images simultaneously with the GDSII structure. **Note:** In order to show the entire background image, the viewer/editor area coordinates have to be fit to the image coordinates (⇒ *UV windows* on page 4-4, ⇒ *Options menu* on page 4-21).
UV display…	Command is common to all UV windows (⇒ *UV windows* on page 4-4).

2.2.5 Annotate menu

The **Annotate** menu is only available in the **GDSII Viewer** dialog.

The **Annotate** menu serves to add and handle comments and markers. They are stored within a separate file (*.ann) and do not influence the related GDSII database file.

The menu items have the following functions:

Item	Function
Show annotations	Choose this option to activate the annotation functionality. After choosing, the annotation location is indicated by a red triangle or a red frame. To delete an annotation, select it and press the **Del** key.
Text	Choose this command to create an annotation text. After choosing the command, move the mouse to the point of interest and click the left mouse button or press the **Space bar**. The **Edit Annotation Text** dialog with additional options will be opened. Use input fields **Position U/V** to define the coordinates of the annotation location. Use **Creation angle** to enter the angle for rotating the annotation location around the coordinate origin (U/V = 0/0). To modify content, U/V position and alignment of a text annotation, double click on the red triangle next to the text. The **Edit Annotation Text** dialog will be opened and you can modify the settings.
Rectangle	Choose this command to create an annotation box. After choosing the command, point to any corner of the area to be annotated and click the left mouse button or press the **Space bar**. Then point to the opposite corner of that area, a rectangular frame will be drawn, and click the left mouse button again or press the **Space bar**. Pressing the Shift key while dragging, the first point defines the center and not the first corner of the rectangle. Pressing the Control key the rectangle will become quadratic. The annotation box will be displayed with a red frame. **Note:** To modify U/V position and alignment of the annotation box, double click on it. The **Edit Annotation Rectangle** dialog will be opened and you can modify the settings.

Item	Function
Ellipse	Choose this command to create an annotation ellipse. The handling is analog to **Annotation** ➡ **Box**.
Delete	Choose this command to delete a selected annotation.
Move	Choose this command to move a selected annotation. The movement vector has to be defined by two subsequent mouse clicks: First mouse click defines the starting point of the movement vector; second click defines the end point of the movement vector. For digital control, the resulting shift values in both axes (UV) are displayed within the status bar. To perform a combined move and copy operation, press and hold the **Control** key while defining the movement vector. **Note:** With this command it is only possible to move one annotation. If you want to move several annotations, click on the **Select** tool in the toolbox after the selected annotation has been moved to the right place and select **Annotation** ➡ **Move** again to move another annotation.

2.2.6 Modify menu

The **Modify** menu is only available in the **GDSII Editor** dialog.

The menu items have the following functions:

Modify menu item		**Function**
Properties...		Calls up a dialog for modifying a selected element digitally. The dialog contains all parameters that are specific for the selected element, i.e. the type of dialog is automatically related to the currently selected element. The dialog is also available by double clicking with the left mouse button on a structure element. **Note:** If more than one element is selected, modification is possible only for the element with the lowest serial number.
Cut		Choose this command to cut the selected structure element(s). After cutting, the structure elements will be converted as follows: – Filled text strings, circles and ellipses to polygons – SPL as well as unfilled text strings, circles and ellipses with zero width to paths **Note:** This command is not available for dots or structure references. Two different possibilities for cutting exist:
	Horizontal	Choose this command to cut a selected elements along a horizontal line. After choosing the command, define the cutter line by two subsequent mouse clicks: First mouse click defines the starting point. Then move the mouse pointer, while a horizontal line will be drawn. Second mouse click defines the end point. All selected elements will be cut along that line if they are touched.
	Vertical	Choose this command to cut the selected elements along a vertical line. Handling is analog to horizontal cutting.
Dose factor		Choose this command to assign another dose to selected structure elements. Two different possibilities exist:

Modify menu item	**Function**
Set	Calls up the **Change Dose Factor** dialog to enter a dose factor for all selected elements.
Scale	Calls up the **Scale Dose Factor** dialog to enter a scale factor for the doses of all selected elements
Duplicate	Choose this command to duplicate selected structure elements. Two different possibilities exist:
Single	Choose this command to make a copy of selected elements. This command copies the selected elements and activates the command **Modify** ➡ **Move**. **Note:** Annotations will not be duplicated.
Matrix	Calls up the Duplicate Elements dialog to copy selected elements in multiple columns and rows with dose scaling. You can duplicate the elements using cartesian or polar coordinates (⟹ *Duplicate Elements dialog* on page 121).
Layer...	Calls up the **Modify Layer** dialog to assign another layer to selected structure elements in Within the dialog, select the layer to be assigned. All layers in use within the opened structure are listed.
Line width...	Calls up the **Line Width** dialog to assign another line width to selected paths µm.
Mirror	Choose this command to perform a mirror operation for all selected structure elements. Three different possibilities exist:
Horizontal	Choose this command to perform a horizontal mirror operation of the selected structure elements with respect to a vertical mirror axis, i.e. V direction, which is automatically placed in the centre of the enclosure of the selected structure elements.
Vertical	Choose this command to perform a vertical mirror operation of the selected structure elements with respect to a horizontal mirror axis, i.e. U direction which is automatically placed in the centre of the enclosure of the selected structure elements.

Modify menu item	**Function**
Line	Choose this command to perform a mirror operation of the selected structure elements with respect to a mirror axis, which has to be defined by two subsequent mouse clicks.
Move	Choose this command to move the selected elements. The movement vector has to be defined by two subsequent mouse clicks: First mouse click defines the starting point of the movement vector, the second click defines the end point of the movement vector. For digital control, the resulting shift values in both axes are displayed within the status bar. To perform a movement along the U and V axis, press and hold the **Control** key while defining the movement vector. **Note:** Alternatively, you can move structure elements by distance via the **Transform** add-on window (⟹ *Transform add-on window* on page 140).
Rotate	Choose this command to rotate the selected structure elements. **Note:** This command is not available for structure references and text elements. Two different possibilities for rotation exist:
Center	Calls up the **Rotate Elements** dialog to rotate the selected structure elements digitally. The rotation centre is defined by the centre point of the enclosure of the selected elements. The dialog provides a range of most used rotation angles on buttons and in addition, an input field for digital input of the rotation angle. To perform a combined rotation and copy operation, select the **Copy** checkbox.
Free	Choose this command to rotate the selected elements by mouse control. The rotation procedure has to be defined by two subsequent mouse clicks: The first mouse click starts the procedure. Then move the mouse circularly around until the enclosure of the selected elements has reached the requested target position. Fix this target position with the second mouse click. For digital control, the resulting shift values in both axes and the rotation angle are displayed within the status bar.

Modify menu item		Function
Scale		Choose this command to scale the selected structure elements. **Note:** This command is not available for structure references and text elements. Two different possibilities for scaling exist:
	Factors	Calls up the **Rotate Elements** dialog to scale the selected structure elements digitally. The scaling centre is defined by the centre point of the enclosure of the selected elements. The dialog provides a range of most used scaling factors and in addition, input fields for digital input of scaling factors for both directions, separately. Negative scaling factors include an additional mirror operation. To perform a combined scale and copy operation, select the **Copy** checkbox.
	Free	Choose this command to scale the selected element(s) by mouse control. The scaling procedure has to be defined by two subsequent mouse clicks: The first mouse click starts the procedure, the second click defines the new size of the enclosure of one or more selected elements. In detail, the enclosure of the selected elements, i.e. old area, is transferred into a new area, which is defined in size and orientation by the two clicks. Consequently, this operation may involve zooming and mirroring actions, simultaneously. Proceed as follows: Select a target position of the upper right corner of the old area. Press the left mouse button and keep it pressed. Move the mouse as follows to define the target size of the element: – Move to the right hand side to right affect the expansion, – Move up to affect the height, – Move left to perform a horizontal mirroring, – Move down to perform a vertical mirroring. For digital control, the width and height, i.e. du and dv, of the new area are displayed within the status bar. Fix the target size with a second mouse click. To perform an equal scale in U and V direction, press and hold the **Control** key while defining the target size.
Group		Choose this command to group selected elements.

Modify menu item	**Function**
Ungroup	Choose this command to ungroup a selected group so that the GDSII elements can be treated independently.
Order	Choose this command to enable modification of the element sequence. The sequence may be essential for patterning routines. After choosing this command, the serial numbers of all elements are displayed. To change this order, select the first element of the new order, then the second one etc. Press **ESC** to interrupt this command. **Note:** If you call up the order command again, the serial numbers of the elements will start with 01 again by selecting the first element.
Raise to top	Brings a structure element into the foreground of the UV window.
Lower to bottom	Brings a structure element into the background of the UV window.
Align	To choose a command for aligning selected structure elements at each other along specified axis or an anchor point. **Note:** The commands correspond to the commands in the **Align and distribute** add-on window (⟹ *Align and distribute add-on window* on page 141)
Distribute	To choose a command for distributing selected structure elements at each other along a specified axis or an anchor point. **Note:** The commands correspond to the commands in the **Align and distribute** add-on window (⟹ *Align and distribute add-on window* on page 141)
Grow...	Calls up the **Grow elements** dialog to grow all selected elements by a defined amount in µm.
Overlaps out	Choose this command to remove overlaps of at least two selected elements to avoid an overexposure.
Shrink...	Calls up the **Shrink elements** dialog to shrink all selected elements by a defined amount in µm.

Modify menu item	**Function**
Snap to grid...	Choose this command to enlarge too small elements respectively too small gaps between elements. Before choosing this command, select the elements to be modified. Choose this command and click somewhere in the UV window. The **Snap To Grid** dialog will be opened. Modify the UV values for the **Base point** of the grid. The input fields are preset by the selected base point of the selected elements. Enter the used grid size. The grid size is the smallest allowed distance between two structures, respectively the smallest dimension of an element.
Boolean operations	Choose this command to perform Boolean operations over selected GDSII elements. The functions **AND**, **OR**, **SUB**, and **XOR** are available. **SUB** subtracts the element selected second from the one selected first. **XOR** is the "exclusive or" operation. How to apply the boolean operations refer to ⇒ *Using Boolean operations* on page 40.
Convert	Choose one of these commands to convert structure elements into another shape – e.g. circles into dots – via a related dialog box. Five possibilities exist:
to polygons/ paths...	To convert ellipses, circles and text strings to paths or polygons, depending on the line width (SPL or not) and the filling. The structure elements will be converted as follows: – Filled text strings, circles and ellipses into polygons – Unfilled SPL text strings, circles and ellipses with zero line width into paths
to FBMS paths...	To convert all but referenced structures to FBMS paths.
Circles to dots...	To convert circles to dots.
Polygons/ paths to circles...	To convert polygons into circles and paths to SPL circles. When converting a polygon into a circle the center of all points is calculated which becomes the center of the circle. The largest distance of all polygons' corners with respect to the center is used as the radius.

Modify menu item	Function
Polygons to paths...	To convert polygons into open paths.

2.2.7 Add menu

The Add menu is only available in the editor.
You can modify all added structure elements afterwards by means of the command **Modify** ➡ **Properties** or by double clicking the element. An element specific dialog will be opened offering possibilities for adjustment/settings.

How to add and edit structure elements refer to (⇒ *Creating structure elements* on page 5-28)

The menu items have the following functions:

Add menu item	Function
Preset	Choose this command to assign layer, dose and line width as a standard value for subsequent **Add** operations. After choosing this command, a cascading menu appears offering the following settings:
Dose factor	Calls up the **Preset Dose Factor** dialog to define a standard dose for all subsequently added basic elements. In the **Dose factor** input field, enter the dose to be preset as a factor (1 = 100% of clearing dose).
Layer	Calls up the **Preset Layer** dialog to define a standard layer for all subsequently added basic elements. All layers in use within the opened structure are listed. Select the layer to be preset. The preset layer is indicated within the status bar of the **Editor** or **Viewer** dialog. **Note:** To select a new layer, create this new layer first via the **Layer Properties** subdialog (⇒ *Layer Properties subdialog* on page 147).

Add menu item	Function
Line width	Calls up the Preset **Line Width** dialog to define a standard line width for all subsequently added paths. In the Line width input field, enter the line width to be preset in µm. **Note:** A value of zero leads to single pixel lines. These are no standard GDSII elements but useful for special ultra-high resolution applications.
Circle	Choose this command to add a circle or an ellipse, filled or border only, to the active layer (⟹ *Adding ellipses* on page 28).
Dot	Choose this command to add a dot to the active layer. Dots are single points with zero extension (⟹ *Adding dots* on page 29).
Open path	Choose this command to add a closed or open path to the active layer (⟹ *Adding open and closed paths* on page 30). **Note:** There is a difference between a closed path and a polygon: A polygon has an area whereas a path consists of a border only. This difference can be made visible by choosing the menu command **Options** ➡ **Fill.**
Polygon	Choose this command to add a polygon to the active layer (⟹ *Adding polygons* on page 31).
Rectangle	Choose this command to add a rectangular box to the active layer (⟹ *Adding rectangles* on page 32).
Structure reference	Choose this command to add a structure reference, i.e. to design hierarchical GDSII files.(⟹ *Creating hierarchical GDSII Databases* on page 62).
Text	Choose this command to add a text element to the active layer (⟹ *Adding text elements* on page 33).
Bitmap	Choose this command to add a bitmap (⟹ *Adding bitmaps* on page 37).
As reference	Choose this command to add the link to a black and white image which will be exposed in a raster mode.

Add menu item	Function
As bitmap	Choose this command to import an image so that each pixel of the image is a separate GDSII element. All GDSII elements are automatically grouped.
Group of curves	Choose this command to add a group of curves to the active layer. This command gives access to a generator for groups of curves, which can be specified by flexible mathematical formulas (⇒ *Creating a group of curves* on page 55).
FBMS	Choose this command to add a FBMS structure element to the active layer. For more details about FBMS structure elements refer to chapter (⇒ *FBMS (option)* on page 14-6).
Path	Adds a FBMS path to the active layer.
Circle	Adds a FBMS circle to the active layer.
MBMS Path	Choose this command to add a MBMS structure element to the active layer. For more details about FBMS structure elements refer to chapter (⇒ *MBMS (option)* on page 14-11).
Manual mark scan	Choose this command to add a 2-dimensional mark scan, i.e. an Imagescan, which can be used for aligning the writefield during a patterning in a semi-automatic fashion. After positioning the mark by using the mouse or digitally, this command opens the **Manual Mark Scan Properties** dialog which aids the definition of the mark scan. **Center** defines the position of the mark in U- and V-direction. Enter **Size** to modify the area which will be scanned around the center position. The number of points used for the scan mark can be defined using **Scan points**. A point-by-point averaging is defined by modifying **Average**. This functionality is described in (⇒ *Beam to Sample Alignment* on page 13-1) **Note:** Settings like **Scan points** and **Average** are global settings which are equal for all mark scans in the GDSII design. This parameter cannot be assigned for single mark scans.

Add menu item	**Function**
Auto mark scan	Choose this command to add a 1-dimensional mark scan, i.e. a Linescan, which can be used for aligning the write-field during a patterning in an automatic fashion. After positioning the mark by using the mouse or digitally, the command opens the **Auto Mark Scan Properties** dialog which aids the definition of the mark scan. **Center** defines the position of the mark in U- and V-direction. Select the **Scan direction** to define the direction of the Linescan. Enter **Size** to modify the length of the scan whereas the **Average size** modifies the width used for averaging. The number of points used along the length of the scan can be defined as well as how often scans were recorded for averaging using **Scan points** and **Average points**. A point-by-point averaging is defined by modifying **Average**. This functionality is described in (⇒ *Beam to Sample Alignment* on page 13-1) **Note:** Settings like **Scan points**, **Average Pointer** and **Average** are global settings which are equal for all mark scans in the GDSII design. This parameter cannot be assigned for single mark scans.

2.3 Edit Rectangle dialog

In the **Edit Rectangle** dialog you can modify position, energy dose and layer affiliation of a rectangle.

You call up the dialog by double clicking a rectangle in the **GDSII Editor** dialog.

How to add a rectangle to the design refer to (⇒ *Adding rectangles* on page 5-32).

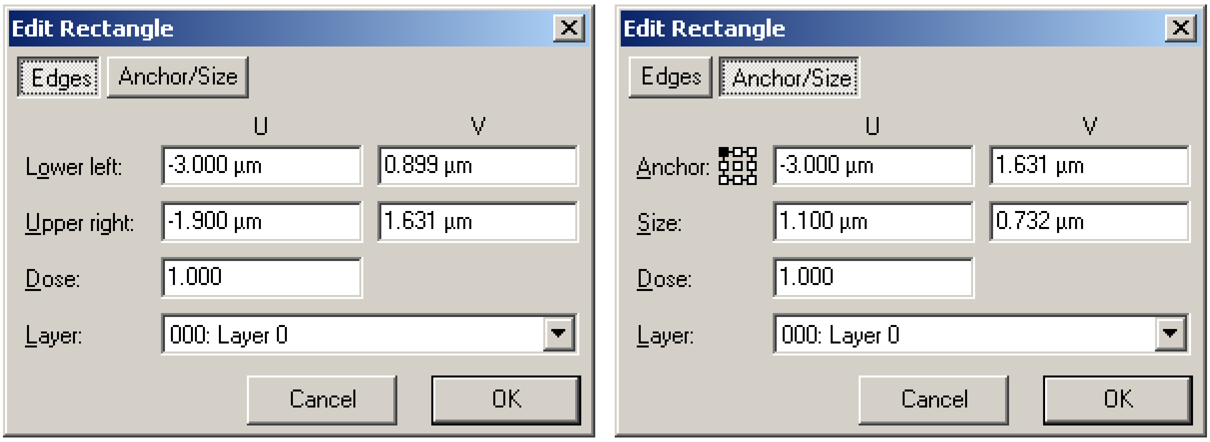
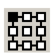


Figure 5-50: **Edit Rectangle** dialog – **Edges**, **Anchor/Size**

The control elements have the following functions:

Control element	Function
Edges	To define size and position of the rectangle via its UV edge coordinates.
Lower left:	To enter the lower left U and V coordinates in µm.
Upper right:	To enter the upper right U and V coordinates in µm.
Anchor/Size	To define size and position of the rectangle based on one of nine anchor points.
	To select an anchor point.
Anchor:	To enter the UV coordinates of the anchor point.
Size:	To define the size based on the selected anchor point in UV direction.

Control element	Function
Dose:	To set a dose factor which is multiplied to the dwell time.
Layer:	To define the layer.

2.4 Circle Properties dialog

In the **Circle Properties** dialog you can edit the properties of ellipses and circles.

You call up the dialog by double clicking an ellipse in the **GDSII Editor** dialog.

How to add an ellipse to the design refer to (⇒ *Adding ellipses* on page 28).

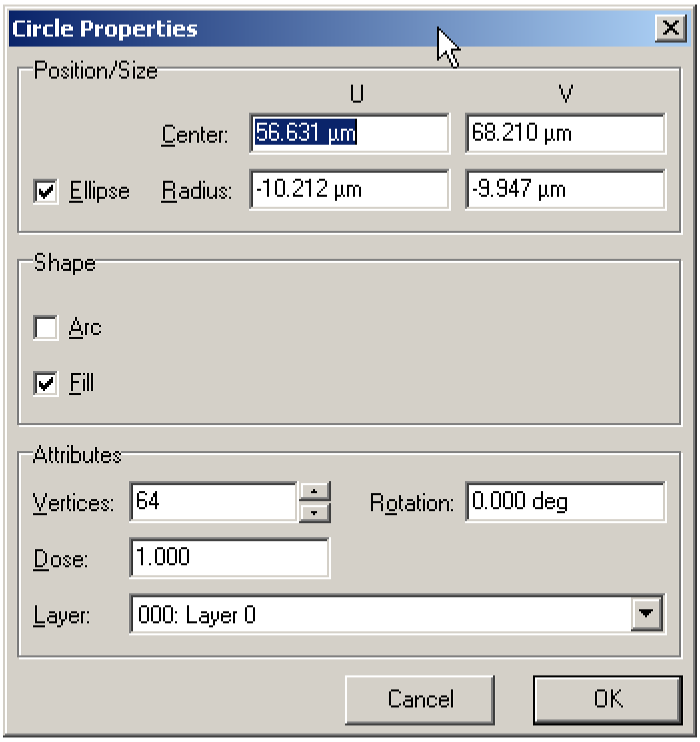


Figure 5-51: **Circle Properties** dialog

The control elements have the following functions:

Control element	Function
Center:	To enter the U and V centre coordinates of the circle or the ellipse in µm.

Control element	Function
Radius:	To enter the radius of the circle, respectively the U and V radii of the ellipse in µm.
Ellipse	To change the elements shape between circle and ellipse. If **Ellipse** is disabled, the element will be transformed into a circle with the **Radius: U** value as radius.
Arc	To change the elements style. If the checkbox is selected, two input fields appear to enter an **Angle Start** value and an **Angle End** value for the circular or elliptical arc to be created in the range of -360° through +360°. If the checkbox is cleared, a full size circle or ellipse is added.
Fill	To change the elements kind. If the checkbox is selected, a circle or ellipse will be filled and an arc will be closed to become a filled segment not a sector. If the checkbox is cleared, an input field appears to enter a **Width** value for the circular or elliptical ring respectively for the arc in µm.
Vertices:	To enter the number of corners for the element to be designed. This number must be in the range of 5 through 1024. The starting point is equal to the end point and this is counted twice.
Rotation:	To rotate the circle or ellipse counterclockwise.
Dose:	To set a factor which is multiplied to the dwell time.
Layer:	To define the layer.

2.5 Edit Polygon dialog

In the **Edit Polygon** dialog you can modify position, energy dose and layer affiliation of the polygon.

You call up the dialog by double clicking a polygon in the **GDSII Editor** dialog.

How to add a polygon to the design, refer to (⇒ *Adding polygons* on page 5-31).

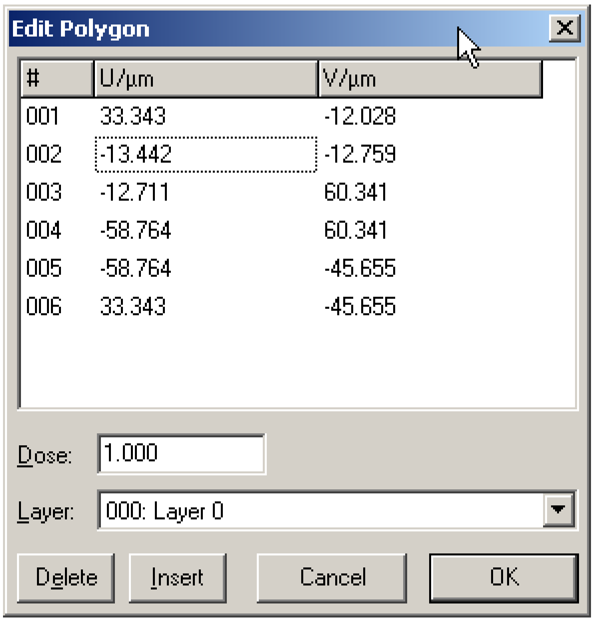


Figure 5-52: **Edit Polygon** dialog

The list in the upper part of the dialog shows the current position of all corners of the selected polygon as well as the current layer affiliation. With click on the entries in the columns **U/µm** and **V/µm**, you can edit the coordinate values.

The layer affiliation can be modified via the combo box in the lower part of the dialog.

The columns and control elements have the following functions:

Column/ Control element	Function
#	Indicates the current serial number of the corner.
U/µm	Indicates the current U position in µm.
V/µm	Indicates the current V position in µm
Dose:	To set a dose factor which is multiplied to the dwell time.
Layer:	To select a layer.
Delete	Deletes a selected corner of the polygon from the list. If a corner is selected, the corresponding corner will be highlighted red in the UV window. Confirm with **OK** to apply the command.

Column/ Control element	Function
Insert	Adds a new corner to the polygon. The new corner will be added depending on the corner currently selected in the list. With click on **Insert**, a new corner will be added to the list below the selected one, with the same coordinate values. The serial number of the following list entries will be counted further. Confirm with **OK** to apply the command.

2.6 Edit Path dialog

In the **Edit Path** dialog you can modify position, energy dose and layer affiliation of an open or closed path.

You call up the dialog by double clicking a path in the **GDSII Editor** dialog.

How to add a path to the design, refer to (⇒ *Adding open and closed paths* on page 5-30).

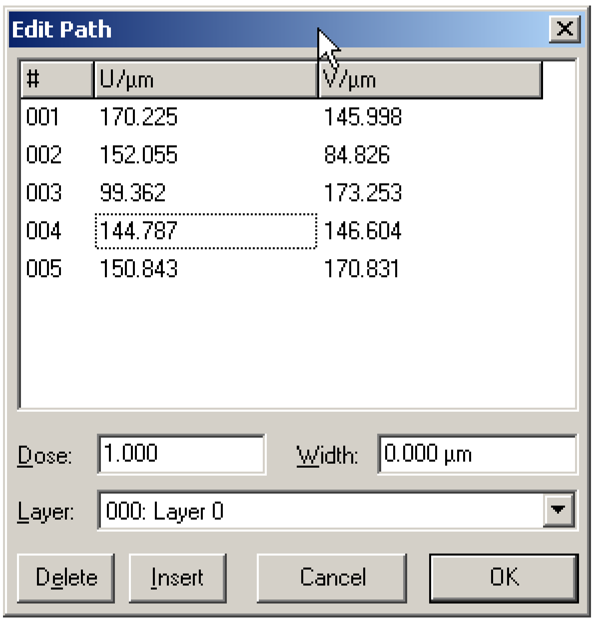


Figure 5-53: **Edit Path** dialog

The list in the upper part of the dialog shows the current position of all corners of the selected path as well as the current layer affiliation. With click on the entries in the columns **U/µm** and **V/µm**, you can edit the coordinate values.

The layer affiliation can be modified via the combo box in the lower part of the dialog.

The columns and control elements have the following functions:

Column/ Control element	Function
#	Indicates the current serial number of the corner.
U/µm	Indicates the current U position in µm.
V/µm	Indicates the current V position in µm
Dose	To set a dose factor which is multiplied to the dwell time.
Width	To set a stroke weight of the path. A value of 0 creates a single pixel line.
Layer	To select a layer.
Delete	Deletes a selected corner of the path from the list. If a corner is selected in the list above, the corresponding corner will be highlighted red in the UV window. Confirm with **OK** to apply the command.
Insert	Adds a new corner to the path. The new corner will be added depending on the corner currently selected in the list. With click on **Insert**, a new corner will be added to the list below the selected one, with the same coordinate values. The serial number of the following list entries will be counted further. Confirm with **OK** to apply the command.

2.7 Edit Dot dialog

In the **Edit Dot** dialog you can modify position, energy dose and layer affiliation of a dot.

You call up the dialog by double clicking a dot in the **GDSII Editor** dialog.

How to add a dot to the design refer to (⇒ *Adding dots* on page 5-29).

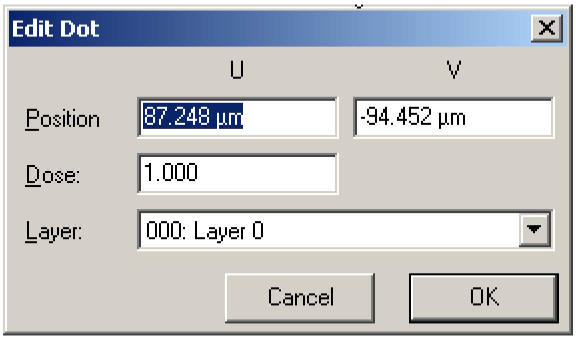


Figure 5-54: **Edit Dot** dialog

The control elements have the following functions:

Control element	Function
Position	To enter the U and V coordinates in µm.
Dose	To set a dose factor which is multiplied to the dwell time.
Layer	To define the layer.

2.8 Edit Structure Reference Properties dialog

In the **Edit Structure Reference Properties** dialog, you can select a structure you want to add as reference to the current selected structure and define settings for creating an array.

You call up the dialog by double clicking a structure reference in the **GDSII Editor** dialog.

How to add a structure reference to a structure refer to (⇒ *Creating hierarchical GDSII Databases* on page 5-62).

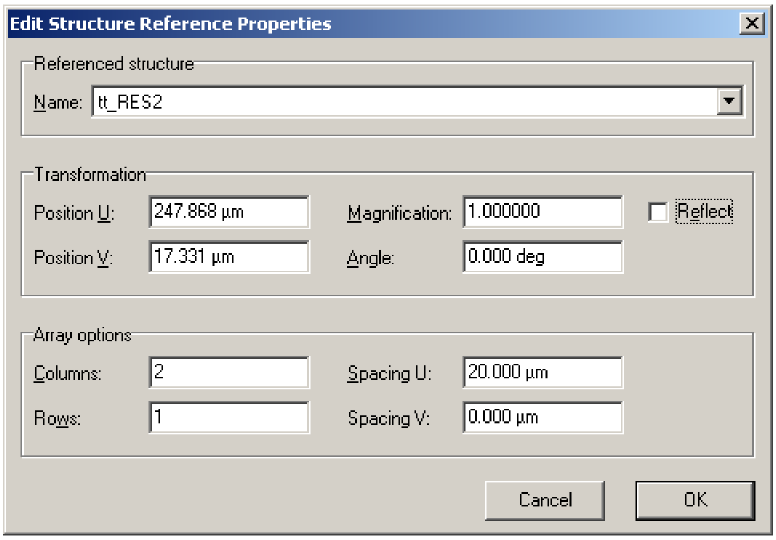


Figure 5-55: **Edit Structure Reference Properties** dialog

The control elements have the following functions:

Control element	Function
Name	To select a structure, which can be linked to the current structure.
Magnification	To enter a scaling factor for the referenced structure. For example a factor 1 includes the referenced structure in its original dimensions.
Reflect	To select, if the element is reflected about the U axis before angular rotation. **Note:** For an array reference, the entire array is reflected, with the individual array members rigidly attached.
Angle	To enter a rotation angle for the entire structure matrix. The rotational center is the lower left corner of the first referenced structure, i.e. the lower structure, within the matrix. **Note:** The entire matrix is rotated, not the single matrix elements.
Columns	To enter the number of repetitions of the referenced structure along the rotated horizontal structure axis.
Spacing U	To enter a distance between the columns on the U axis.
Rows	To enter the number of repetitions of the referenced structure along the rotated vertical structure axis.
Spacing V	To enter a distance between the columns on the V axis.

2.9 Text Properties dialog

In the **Text Properties** dialog, you can define content and properties of a text element.

You call up the dialog via the toolbox ➡ **Text** tool or by double clicking a text element in the **GDSII Editor** dialog.

How to add a text element to the design, refer to (⇒ *Adding text elements* on page 5-33).

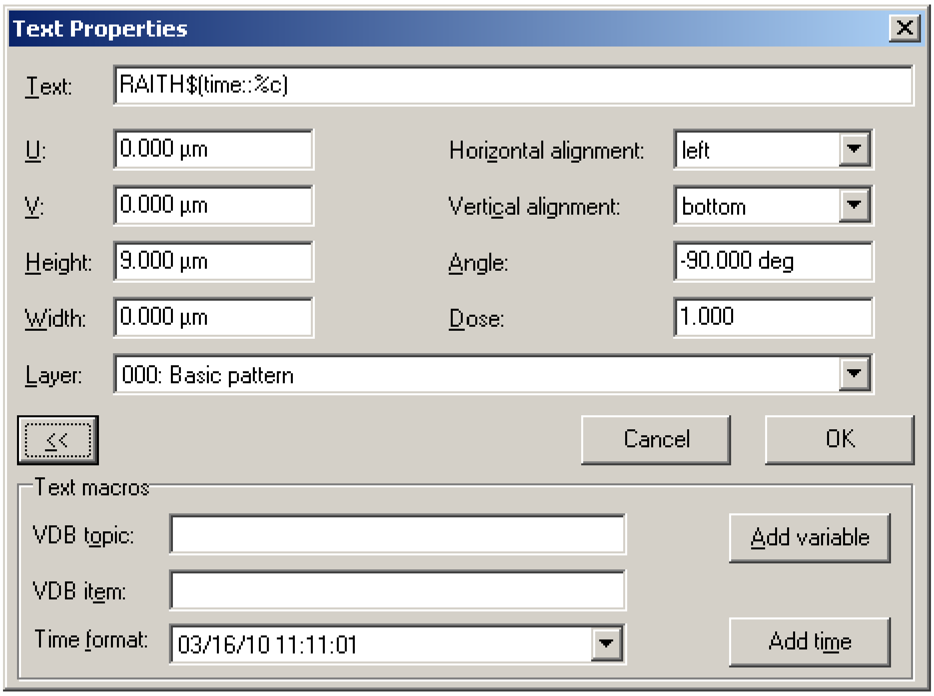


Figure 5-56: **Text Properties** dialog

The **Text Properties** dialog is divided into two subsets:

- Basic commands subset in the upper part of the dialog to define the basic parameters for a text string.
- **Text macros** subset in the lower part of the dialog to define date and time strings as well as contents of VDB variables. Via the **>>** button or **<<** button, you can expand and retract the subset.

2.9.1 Basic commands subset

The input fields have the following functions:

Input field	Function
Text	To enter a text string as clear text. **Note:** If you define date or time strings or VDB variables in the **Text macros** subset, they will be added to the text string. The syntax is as follows: $(vdb:<topic>.<variable>)
U:/V:	To enter the position of a reference point for the text string in µm.
Horizontal alignment:	To define the horizontal alignment of the text string with respect to its reference point.
Vertical alignment:	To define the vertical alignment of the text string with respect to its reference point.
Height:	To enter the height of the letters in µm.
Angle:	To enter the rotation angle of the text in °.
Width:	To enter the width of the letters in µm. **Note:** A value of zero will lead to letter consisting of single pixel lines.
Dose:	To enter the dose to be assigned as a factor with 1 = 100% of clearing dose.
Layer:	To select the Layer to be assigned to the text element.

2.9.2 Text macros subset

The control elements have the following functions:

Control element	Function
VDB section	To define the name of the topic and the name of the variable listed within the variables database file (VDB). Syntax: <section>
VDB item	To define the name of the variable listed under that topic within the variables database file (VDB). Syntax: <variable>

Control element	Function
Add variable	Adds the **VDB** variable to the text string in the **Text:** input field.
Time format	To select a time format.
Add time	Adds the time string to the text string in the **Text:** input field.

2.10 Duplicate Elements dialog

In the **Duplicate Elements** dialog, you can duplicate structure elements in order to create a matrix of them.

In the GDSII Editor, you call up the dialog via **Modify** ➡ **Duplicate** ➡ **Matrix**.

How to create a matrix of structure elements refer to (⇒ *MBMS paths with gratings* on page 5-49).

The **Duplicate Elements** dialog is divided into two subsets:

- Tabs subset to duplicate elements
  - horizontally and vertically using cartesian coordinates (⇒ *Duplicate Elements dialog – Cartesian Coordinates tab* on page 5-122),
  - around their center point using polar coordinates (⇒ *Duplicate Elements subdialog – Polar Coordinates tab* on page 5-123).
- **Dose scaling** subset to define the dose assignment for the duplicated elements (⇒ *Duplicate Elements subdialog – Dose scaling subset* on page 5-124).

2.10.1 Duplicate Elements dialog – Cartesian Coordinates tab

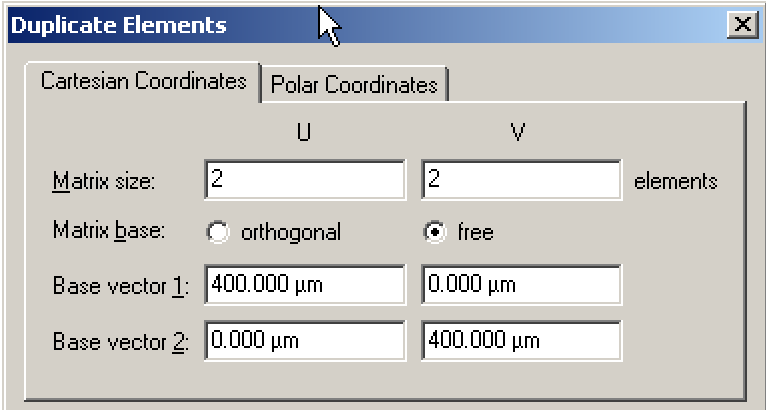


Figure 5-57: **Duplicate Elements** dialog – **Cartesian Coordinates** tab

The control elements have the following functions:

Control element	Function
Matrix size:	To define the size of the matrix in U and V, i.e. the number of columns and rows.
Matrix base:	To select, how the duplicated elements should be arranged: – **orthogonal**: The columns and rows will be arranged horizontally and vertically to each other. – **free**: The course of the columns and rows in U and V direction can be defined individually via the **Base vector 1** and **Base vector 2** parameters.
Base vector 1:	To define the distance between the columns in U and V direction, measured from the lower left point of each element respectively. **Note:** If **orthogonal** is selected, the corresponding V input field will be disabled.
Base vector 2:	To define the distance between the rows in U and V direction, measured from the lower left point of each element respectively. **Note:** If **orthogonal** is selected, the corresponding U input field will be disabled.

2.10.2 Duplicate Elements subdialog – Polar Coordinates tab

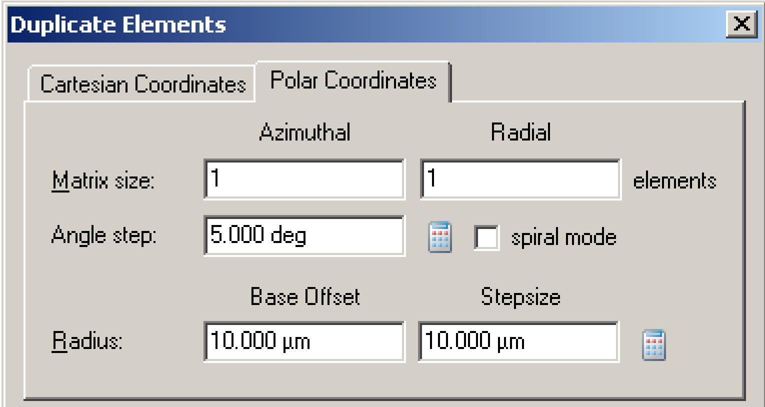


Figure 5-58: **Duplicate Elements** dialog – **Polar Coordinates** tab

The control elements have the following functions:

Control element	Function
Matrix size:	To define the size of the matrix, i.e. the number elements in azimuthal and radial direction.
Angle step:	To define the distance between the elements in azimuthal direction.
	To calculate the minimum angle step on the basis of azimuthal matrix step size and predefined angle step.
spiral mode	To select, if the elements are to be arranged spirally around their center point or not.
Radius	To define the distance of the elements to the center point and to each other: – **Base Offset:** Distance of the matrix from the center point. – **Stepsize:** Distance of the elements to each other, measured from the lower left point respectively.
	To calculate the minimum base offset and Stepsize on the basis of azimuthal matrix step size and structure dimension.

2.10.3 Duplicate Elements subdialog – Dose scaling subset

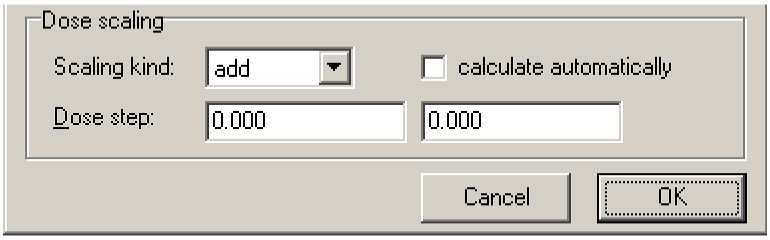


Figure 5-59: **Duplicate Elements** dialog – **Dose scaling** subset

The control elements have the following functions:

Control element	Function
Scaling kind:	To define, how the dose will be calculated for the duplicated elements – **add:** Dose value will be added to the previous element of the matrix. – **multiply:** Dose value will be multiplied to the previous element of the matrix.
calculate automatically	To select, if the dose will be calculated automatically in V direction.
Dose step	In addition, the dose can be varied by entering dose scaling values for each direction.
Cancel	Closes the dialog.
OK	Saves the settings and closes the dialog.

3 Recipe Manager dialog (option)

In the **Recipe Manager** dialog, you can set up, store and manage datasets with attributes for special patterning strategies. This dataset is called patterning recipe. A patterning recipe can be assigned to all structure elements, structure references and groups of a design.

Single patterning recipes are assigned to patterning recipe groups. Recipes and recipe groups are displayed in the **Recipe Manager** tree view.

You call up the dialog via control bar .

- How to work with patterning recipes refer to (⇒ *Working with patterning recipes* on page 5-72).
- For tips how to find the right patterning parameter values and strategies for your application refer to (⇒ *Patterning* on page 14-1).

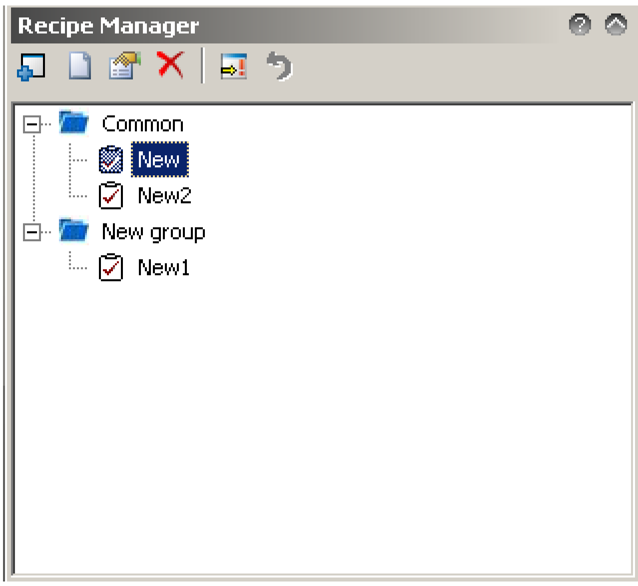


Figure 5-60: **Recipe Manager** dialog

The control elements have the following functions:

Control element	Function
	Creates a new recipe group in the **Recipe Manager** tree view. A recipe group serves as directory for single recipes.
	Creates a new recipe in a selected recipe group. **Note:** If you do not select a recipe group, the new recipe will be added to the recipe group, which was last active.

Control element	Function
	Calls up the **Edit Patterning Recipe** dialog (⇒ *Edit Patterning Recipe dialog* on page 126).
	Deletes selected patterning recipes.

3.1 Edit Patterning Recipe dialog

In the **Edit Patterning Recipe** dialog, you can set up a patterning recipe.

The dialog offers a list of the most important patterning parameters for the patterning of area structure elements like circle and polygons as well as for Single Pixel Lines and dots. You can define the corresponding values and assign the recipe to a structure element.

The list behaves context sensitive depending

- on the parameter currently selected: Only those parameters will be enabled, which are related to each other.
- on the type of parameter currently selected. Some parameter can be set by predefined subparameters – e.g. **fillmode**: **direction**, **concentric**, or **raster**. You can select the parameters via a combo box.

  To appraise the parameters **beam current**, **dwell time**, **dose**, and **dose per loops** a calculator is available respectively.

You call up the dialog via **Recipe Manager** dialog .

For a detailed description of the patterning parameters as well as for tips how to find the right patterning parameter values and strategies for your application, refer to (⇒ *Patterning* on page 14-1).

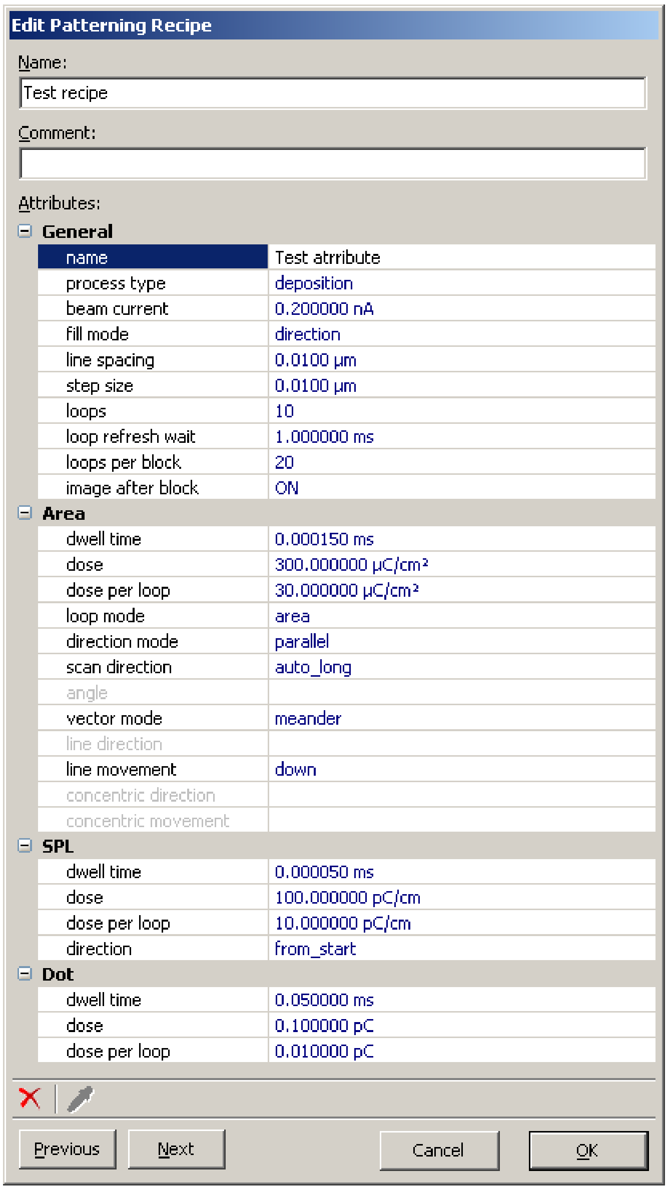


Figure 5-61: **Edit Patterning Recipe** dialog

The control elements and list entries have the following functions:

Control element/ List entry	Function
Name	To enter a name for the recipe.
Comment	To add some additional information to the recipe.

Control element/ List entry	**Function**
Attributes	List to define attributes for special patterning strategies. The **Attributes** list contains the following sublists: – **General** for general settings of the recipe (⇒ *General sublist* on page 129). – **Area** with patterning parameters for rectangles, circles and polygons (⇒ *Area sublist* on page 130). – **SPL** with patterning parameters for Single Pixel Lines (⇒ *SPL sublist* on page 132). – **Dot** with patterning parameters for dots (⇒ *Dot sublist* on page 132).
	Clears up all input fields.
	Reads the patterning parameter settings made in the **Patterning Attributes** add-on window (⇒ *Patterning Attributes add-on window (option)* on page 138).
Previous	Scrolls to the previous recipe parameter set in the **Recipe Manager** tree view and indicates the corresponding settings. **Note:** **Previous** scrolls through the whole tree view upwards, independent of the group the recipe is assigned to.
Next	Scrolls to the next recipe parameter set in the **Recipe Manager** tree view and indicates the corresponding settings. **Note**: **Next** scrolls through the whole tree view downwards, independent of the group the recipe is assigned to.
Cancel	Resets all entries and closes the dialog.
OK	Adopts all entries and closes the dialog.

3.1.1 General sublist

The following parameters and subparameters are available:

Parameter/ Subparameter	Function
name	Optional name, which helps the user to assign the patterning attributes to a special application or GDSII structure.
Process type	To select the type of patterning process:
standard	To perform a patterning predefined from your lithography system.
exposure	To perform a pattern by exposing a resist using the electron beam lithography principle (e.g. E_LINE, RAITH150^{TWO}).
deposition	To perform patterning by depositing material on a resist – e.g. using the Gas Injection System (GIS).
milling	To perform patterning by etching material from a resist using the ion beam lithography principle (e.g. IONLINE).
beam current	Opens the **Beam Current** subdialog to read the beam current from a current meter (⇒ *Beam Current subdialog* on page 133).
fill mode	To select a mode of patterning:
direction	The element will be patterned line-by-line – either from the same direction or in alternating directions.
concentric	The element will be patterned in concentric circles in the following modes: – clockwise or counterclockwise – inwards or outwards
raster	The element will be patterned in raster mode. That means, a whole working area will be patterned and the beam blanker will only be turned on, when the beam passes areas with to exposed structure elements.
line spacing	To enter a value for the spacing between the lines in µm.
step size	To enter a value for the spacing between patterning spots within a line in µm.

Parameter/ Subparameter	Function
loops	To enter, how often a patterning is to be repeated.
loop refresh wait	To enter a delay time, before the next loop will be started in ms.
loops per block	To enter, how many patterning repetitions are to be performed before an image scan is initiated.
image after block	To choose, if an image shall be performed after a loop block is finished. **Note:** To generate an image after block, the **Image** window must be opened before the patterning process is started.
ON	To enable the function.
OFF	To disable the function.

3.1.2 Area sublist

In the **Area** sublist you can define patterning parameters for area structure elements – like rectangles, circles, and polygons.

With click on ..., the **Patterning Attributes Calculator** dialog will be opened to aid calculating the parameters for the dwell time, dose, and dose per loops respectively (⇒ *Patterning Attributes Calculator subdialog* on page 5-134).

The following parameters and subparameters are available:

Parameter/ Subparameter	Function
dwell time	To enter or calculate a value for the dwell time.
dose	To enter or calculate a value for the dose.
dose per loop	To enter or calculate a value for the dose per loop.
loop mode	To define, how the patterning of a structure will be repeated:
area	Repeats the patterning area-by-area.
line	Repeats the patterning line-by-line.

Parameter/ Subparameter	Function
direction mode	To define, in which direction an area structure element is to be patterned line-by-line.
parallel	To perform a patterning in direction of the U or V axis. **Note:** If the structure has no edge parallel to either U or V, it is recommended to use the longitudinal or transverse mode for the patterning, resulting in a structure with a cleaner, or better defined, edge.
angle	To select an angle, relative to the U axis of beam coordinate system for the direction of the patterning lines. All lines will be described at this defined angle.
longitudinal mode	The longest edge of a feature will become the first line in the patterning of the feature and all other lines will be parallel to this first line, separated by the defined line spacing.
transversal mode	The first exposed line will be perpendicular to the longest edge and all other lines will be parallel to this first line, separated by the defined line spacing.
scan direction	To select a patterning mode for **direction mode** – **parallel**:
U	To perform a patterning in direction of the U axis.
V	To perform a patterning in direction of the V axis.
auto_long	The software will select the scan direction automatically dependant on the dimension of the desired structure element: if the element's extension in V is larger than in U direction, **scan direction** – **V** is applied.
auto_short	The software will select the scan direction automatically dependant on the dimension of the desired structure element: if the element's extension in V is shorter than in U direction, **scan direction** – **V** is applied.
angle	To enter an angle for **direction mode** – **angle** in °.
vector mode	To define, how area structure elements are to be patterned:
line	The pattern area structure elements line-by-line. Each line will be patterned from the same direction.

Parameter/ Subparameter	Function
meander	To pattern area structure element line-by-line. Each line will be patterned in alternating directions from line to line.
line direction	To define the horizontal patterning direction for **vector mode – line**:
left->right	Starts the line-by-line patterning always from left to right.
right->left	Starts the line-by-line patterning always from right to left.
line movement	To define the vertical patterning direction for **vector mode – line**:
down	Patterns the elements starting from the most upper line downwards.
up	Patterns the elements from the most lower line upwards.

3.1.3 SPL sublist

In the **SPL** sublist you can define patterning parameters for Single Pixel Lines.

With click on ..., the **Patterning Attributes Calculator** dialog will be opened to aid calculating the parameters respectively (⇒ *Patterning Attributes Calculator subdialog* on page 5-134).

The following parameters and subparameters are available:

Parameter/ Subparameter	Function
dwell time	To enter or calculate a value for the dwell time.
dose	To enter or calculate a value for the dose.
dose per loop	To enter or calculate a value for the dose per loop.

3.1.4 Dot sublist

In the **Dot** sublist you can define patterning parameters for dots.

With click on ..., the **Patterning Attributes Calculator** dialog will be opened to aid calculating the parameters respectively (⇒ *Patterning Attributes Calculator subdialog* on page 5-134).

The following parameters and subparameters are available:

Parameter/ Subparameter	Function
dwell time	To enter or calculate a value for the dwell time.
dose	To enter or calculate a value for the dose.
dose per loop	To enter or calculate a value for the dose per loop.

3.2 Beam Current subdialog

In the **Beam Current** subdialog you can define the position and intensity of the beam current – e.g. for measuring beam current at a faraday cup.

You call up the dialog via **Edit Patterning Recipe** subdialog or **Patterning Attributes** add-on window ➡ **General** ➡ **beam current** ➡ ... .

For more information how to find the right beam current value refer to (⇒ *Beam Current subdialog* on page 5-133).

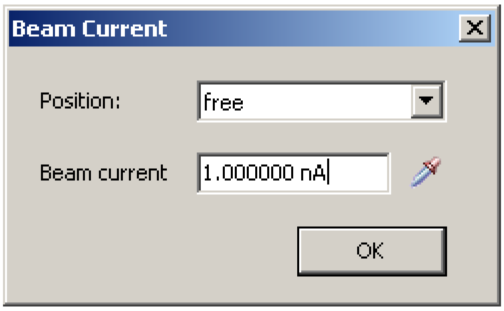


Figure 5-62: **Beam Current** dialog

The control elements have the following functions:

Control element	Function
Position	To select, where the beam current measurement is to be applied.
Beam Current	To enter a value for the beam current in nA.
	To pick up a value for the beam current at the selected **Position**.

3.3 Patterning Attributes Calculator subdialog

With the help of the **Patterning Attributes Calculator** subdialog you can calculate the patterning parameters dwell time, dose, and dose per loop for area, SPL, and dot structure elements.

You call up the dialog via

- **Edit Patterning Recipe** dialog,
- **Patterning Attributes** add-on window.

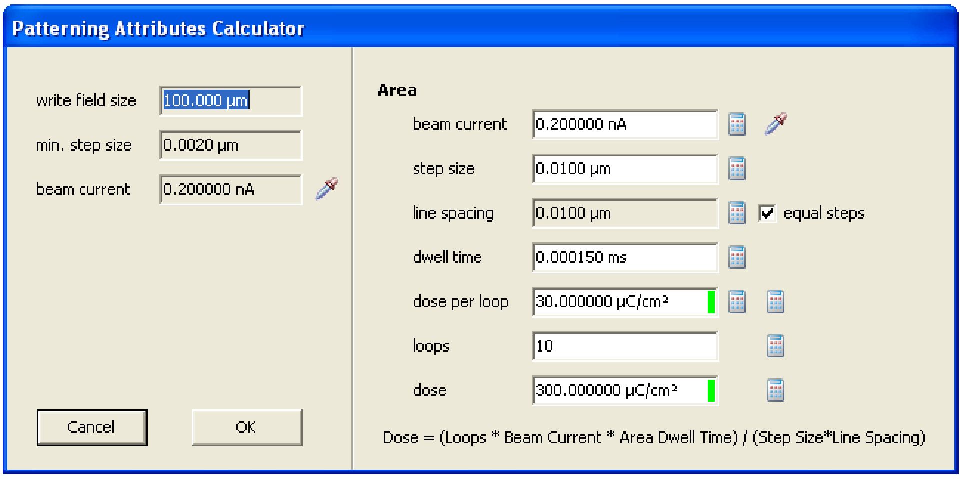


Figure 5-63: **Patterning Attributes Calculator** dialog – **Area** elements

The calculation is based on a formula, which considers various patterning parameters depending on the kind of structure elements. Dwell time, dose, and dose per loop for e.g. SPLs will be calculated via the formula **Dose = (Loops * Beam Current * Line Dwell Time) / (Line Step Size)**.

Basis for the calculation of these three parameters is the number of loops. Enter the number in the loops input field and calculate the values for **dwell time**, **dose**, and **dose per loops** with click on correspondingly.

The calculator will then perform a plausibility check: If one of the three values are implausible after calculation, the corresponding input field will marked with a red bar. In that case, you have to recalculate the value with click on correspondingly.

If the plausibility test is positive, the corresponding input field will be marked with a green bar (⇒ Figure 5-63 on page 5-134).

For a detailed description of all functionalities and the handling of the calculator refer to (⇒ *Patterning Parameter calculation* on page 14-22).

---

4 Add-on windows

This section describes the control elements and commands of the add-on windows of the GDSII module. Add-on windows stay on top of all activated windows until they are closed and manage several commands.

4.1 Toolbox

The toolbox offers a fast access to often used commands for creating and modifying structure elements.

You call up the toolbox via **GDSII Editor/GDSII Viewer** dialog.

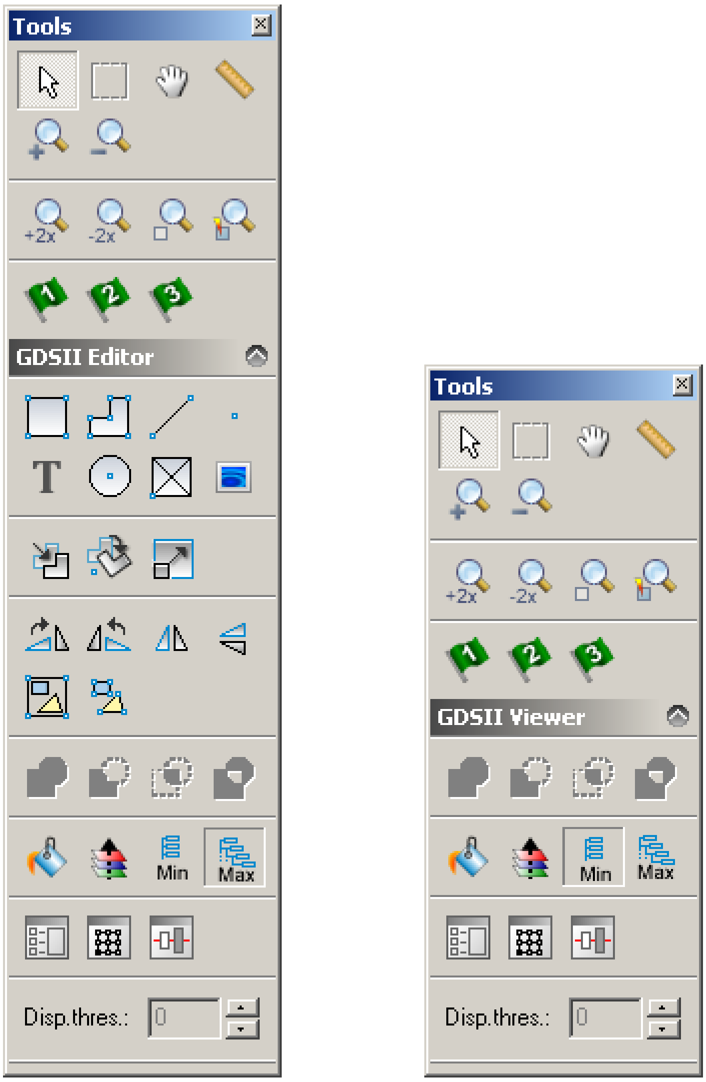


Figure 5-64: Toolbox for editor (left) and viewer (right)

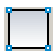
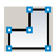
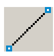
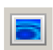
The toolbox is divided into two subsets:

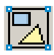
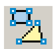
- **Tools** subset with general tools for selecting elements, viewing, measuring, zooming and placing flags. These commands correspond to the general commands in all UV windows and are described in (⇒ *UV windows* on page 4-4).
- **GDSII Editor**/**GDSII Viewer** subset with tools for creating and modifying structure elements respectively. These tools are also available via the **Add**, **Modify**, **Options**, and **View** menus of the main menu bar.

For a detailed description of the tools refer to:

(⇒ *View menu* on page 5-94)
(⇒ *Options menu* on page 5-96)
(⇒ *Modify menu* on page 5-101)
(⇒ *Add menu* on page 5-107)

The tools in the toolbox correspond to the following main menu commands:

Tools	Main menu command
	Editor command **Add** ➡ **Rectangle**.
	Editor command **Add** ➡ **Polygon**.
	Editor command **Add** ➡ **Open path**.
	Editor command **Add** ➡ **Dot**.
	Editor command **Add** ➡ **Text**.
	Editor command **Add** ➡ **Circle**.
	Editor command **Add** ➡ **Structure reference**.
	Editor command **Add** ➡ **Bitmap** ➡ **As reference**.
	Editor command **Modify** ➡ **Move**.
	Editor command **Modify** ➡ **Rotate** ➡ **Free**.

Tools	Main menu command
	Editor command **Modify** ➡ **Scale** ➡ **Free**.
	Rotate selected element 90° clockwise.
	Rotate selected element 90° counter clockwise.
	Editor command **Modify** ➡ **Mirror** ➡ **Horizontal**.
	Editor command **Modify** ➡ **Mirror** ➡ **Vertical**.
	Editor command **Modify** ➡ **Group**.
	Editor command **Modify** ➡ **Ungroup**.
Boolean OR	Viewer and editor command **Modify** ➡ **Boolean operations** ➡ **Union (OR)**.
Boolean SUB	Viewer and editor command **Modify** ➡ **Boolean operations** ➡ **Difference (SUB)**.
Boolean AND	Viewer and editor command **Modify** ➡ **Boolean operations** ➡ **Union (AND)**.
Boolean XOR	Viewer and editor command **Modify** ➡ **Boolean operations** ➡ **Union (XOR)**.
	Viewer and editor command **Options** ➡ **Fill**.
	Viewer and editor command **Options** ➡ **Physical layer order**.
	Viewer and editor command **View** ➡ **Hierarchy** ➡ **0**.
	Viewer and editor command **View** ➡ **Hierarchy** ➡ **Max**.

Tools	Main menu command
	Viewer and editor command **View** ➡ **Patterning attribute editor**.
	Viewer and editor command **View** ➡ **Transformation**.
	Viewer and editor command **View** ➡ **Align and distribute**.
Disp. thres.:	To select a minimum window pixel size for structure elements to be displayed.

4.2 Patterning Attributes add-on window (option)

In the **Patterning Attributes** add-on window, you can inspect and edit the patterning attributes for a selected structure element. You can save the settings to a patterning recipe for future use via the **Edit Patterning Recipe** dialog. The attributes and settings correspond to those in the **Edit Patterning Recipe dialog** (⇒ *Edit Patterning Recipe dialog* on page 5-126).

You call up the add-on window via the toolbox .

For more information about working with patterning attributes refer to (⇒ *Working with patterning recipes* on page 5-72).

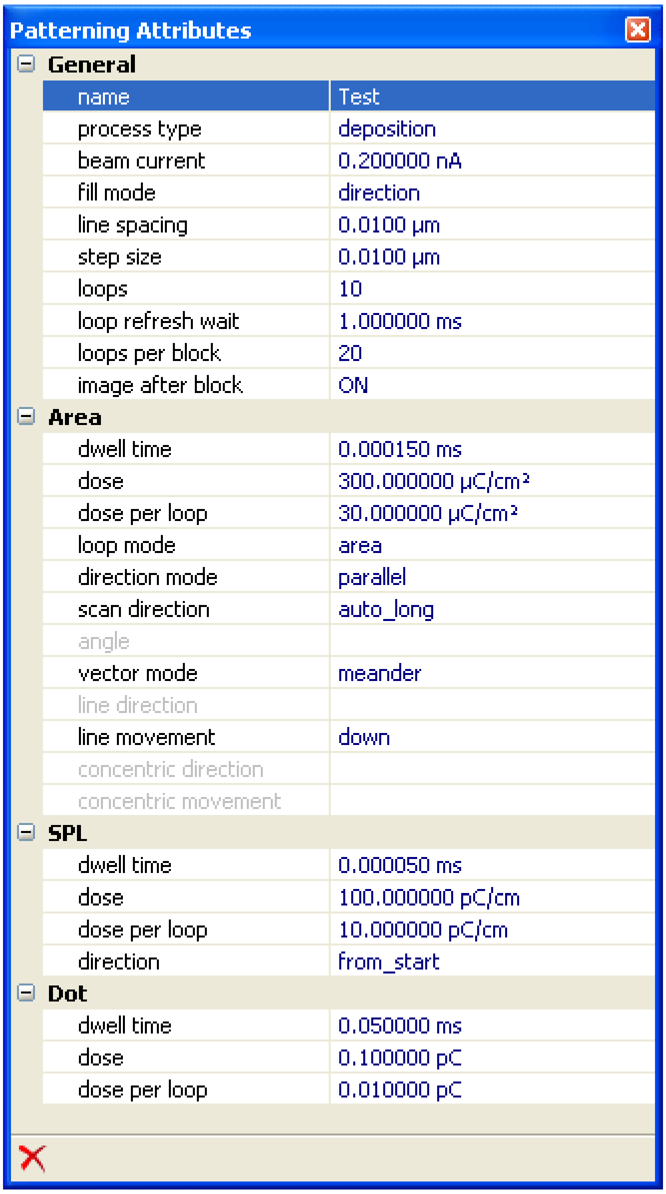


Figure 5-65: **Patterning Attributes** add-on window

4.3 Transform add-on window

In the **Transform** add-on window you can move, scale and rotate a structure element.

You call up the window via the toolbox .

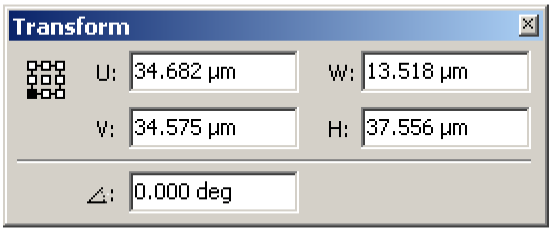
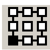


Figure 5-66: **Transform** add-on window

The control elements have the following functions:

Control element	Function
	To select the starting point of the element for movement or rotation. **Example:** If you select the lower left square, the absolute UV coordinate values are applied to that point – e.g. the element will be moved about 2 µm further right on the U axis. If you enter a value for the rotation angle, the element will be rotated around its lower left point.
U:/V:	To enter new absolute coordinates in U and V direction of the position to that the structure element is to be moved.
W:/H:	To enter new absolute values for width and height for the structure element.
	To enter a value in ° by which a structure element is to be rotated. **Note:** Positive values lead to clockwise rotation, negative values to counter clockwise rotation.

4.4 Align and distribute add-on window

In the **Align and distribute** add-on window you can align or distribute selected structure elements along the axis you specify. You can use either the elements edges or anchor points as the reference point.

You call up the window via the toolbox .

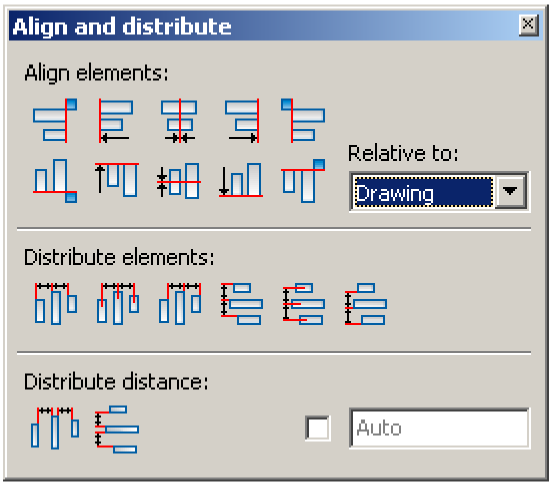
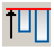
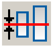
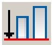
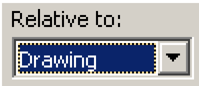
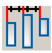
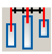
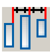
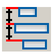
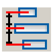
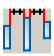
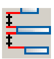


Figure 5-67: **Align and distribute** add-on window

The control elements have the following functions:

Control element	Function
	Aligns the right side of one or more elements at the left side of an anchor. **Note:** As anchor always serves the first selected element.
	Aligns selected elements at their left sides.
	Centers selected elements horizontally.
	Aligns selected elements at their right sides.
	Aligns the left side of one or more elements at the right side of an anchor. **Note:** As anchor always serves the first selected element.
	Aligns the bottom of one or more elements at the top of an anchor. Note: As anchor always serves the first selected element.

Control element	Function
	Aligns selected elements at their top.
	Centers selected elements vertically.
	Aligns selected elements at their bottom.
	Aligns the top of one or more elements at the bottom of an anchor. **Note:** As anchor always serves the first selected element.
	To select, relative to which object the selected elements are to be aligned: – **Selection** – **First selected** – **Drawing**
	Distributes selected elements at the left sides equidistantly.
	Distributes the horizontal centers of selected elements equidistantly.
	Distributes selected elements at their right sides equidistantly.
	Distributes the tops of selected elements equidistantly.
	Distributes the vertical center of selected elements equidistantly.
	Distributes the bottoms of selected elements equidistantly.
	Equalizes the horizontal distances between selected elements.
	Equalizes the vertical distances between selected elements.
	To select, if the size of the gaps between the elements will be set automatically or manual. If you select the checkbox, you can type in the gap width in µm.

4.5 Working areas add-on window

In the **Working areas** add-on window you can define fractions of a whole structure which are of interest for inspecting or patterning purposes.

You call up the add-on window via **GDSII Editor/Viewer** dialog.

How to create and use working areas refer to (⇒ *Using Working areas* on page 5-70).

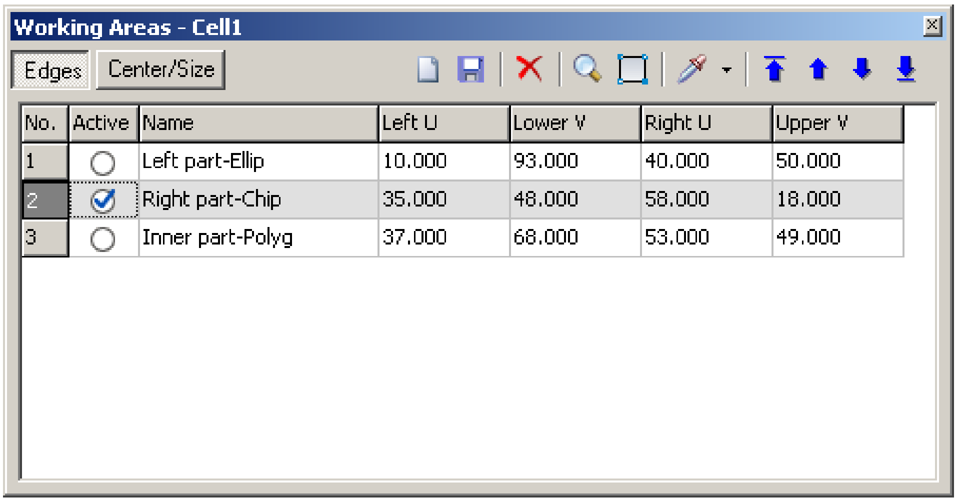


Figure 5-68: **Working areas** add-on window

The **Working Areas** add-on window consists of two subsets:

Subset	Function
Tool bar	To create and manage the working areas (⇒ *Tool bar* on page 143).
Working area list	To define and display the working areas (⇒ *Working area list* on page 144).

4.5.1 Tool bar

The buttons in the tool bar have the following functions:

Button	Function
Edges	To define size and position of a working area via its UV edge coordinates. After choosing this command, the column heads of the working area list will be changed correspondingly.

Button	Function
Center/Size	To define size and position of a working area via its center point. After choosing this command, the column heads of the working area list will be changed correspondingly.
	Adds a new working area to the list.
	Saves changes of the currently active working area.
	Deletes the currently selected working area.
	Shows the whole selected working area.
	Moves the viewport to the center of the selected working area.
	To take over the editor or viewer coordinates into the currently selected working area. There are two commands:
Read boundaries	Takes over the boundaries of the currently selected structure.
Read viewport	Takes over the currently viewable section of the UV window.
	To jump to the first list entry.
	To scroll upwards in the list.
	To scroll downwards in the list.
	To jump to the last list entry.

4.5.2 Working area list

The columns of the working area list have the following meanings:

Column	Function
No.	Indicates the serial number of the working areas.

Column	Function
Active	To activate a working area for patterning. The activated working area will be initiated in the design field with a gray border. **Note:** The gray border will only be indicated, if you have chosen the command **View ➡ Working area** in the main menu bar.
Name	To enter a name for the selected working area.
Left U/ Center U	To define the – U position of the left border, – Center point of the working area on the U axis.
Lower V/ Center V	To define the – V position of the lower border, – Center point of the working area on the V axis.
Right U/ Size U	To define the – U position of the right border, – Dimension of the working area on the U axis.
Right V/ Size V	To define the – V position of the left point of the working area, – Dimension of the working area on the V axis.

4.6 GDSII Layer add-on window

In the **GDSII Layer** add-on window you can

- display and activate a layer for editing in the UV window of the **GDSII Editor/Viewer** dialogs,
- select all structure elements placed on one layer.

You call up the add-on window via **GDSII Viewer/Editor** .

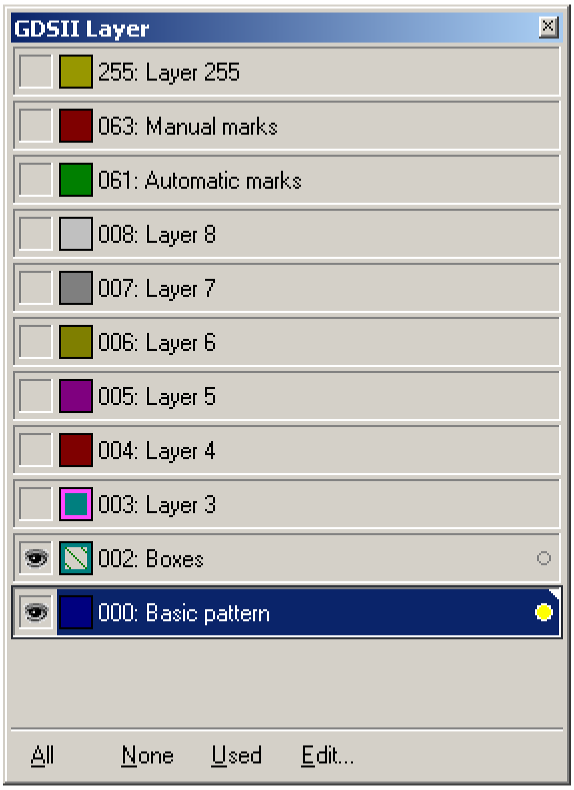


Figure 5-69: GDSII Layer dialog

The control elements have the following functions:

Control element	Function
	– Indicates that the selected layer is displayed. – Blends the selected layer in or out.
	Indicates the layer properties for border and filling of the structure elements in the layer.
000: XXX	Name of the layer.
	– Indicates that the layer contains structure elements. – Indicates yellow, that one or more structure elements of the layer are selected. **Note:** With click on the button you can select all structure elements of the layer – e.g. to copy and paste them into another layer.
All	Selects all available layers.
Used	Selects only layers with content.
None	Cancels the selection.

Control element	Function
Edit	Calls up the **Layer properties** subdialog (⇒ *Layer Properties subdialog* on page 147).

4.7 Layer Properties subdialog

In the **Layer Properties** subdialog, you can create new layers and define name, color, and filling patterns for all layers individually.

You call up the subdialog via **GDSII Layer** add-on window ➡ **Edit**.

How to work with layers refer to (⇒ *Working with Layers* on page 67).

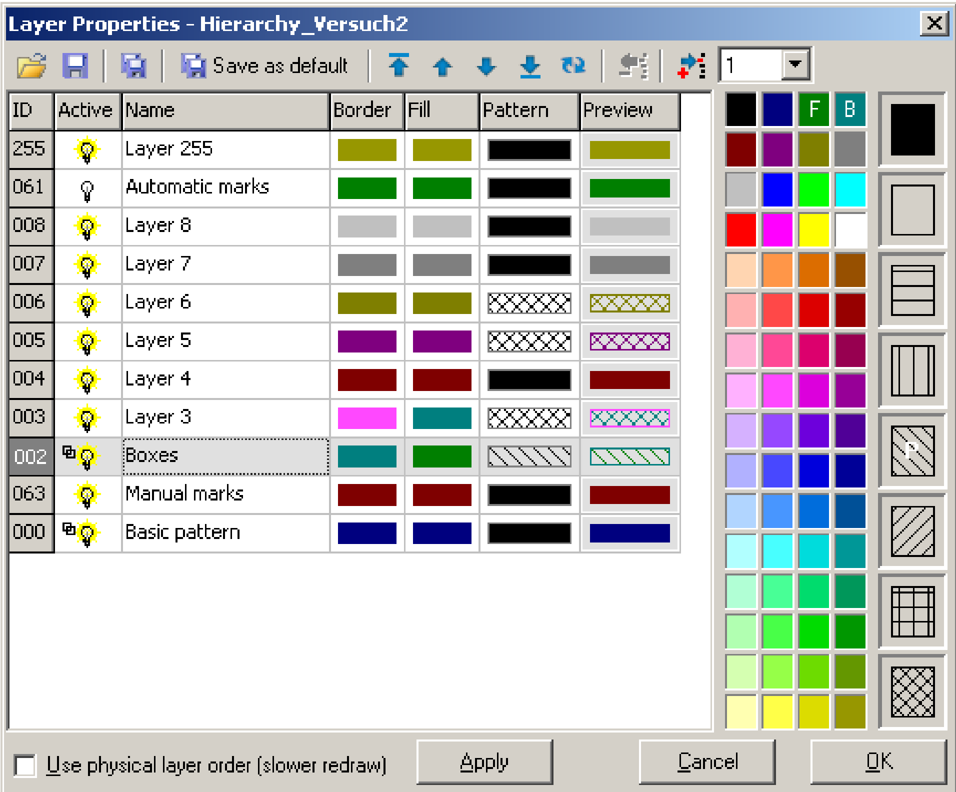


Figure 5-70: **Layer properties** subdialog

The dialog is divided into four subsets:

- Tool bar in the upper part of the dialog to open, save, add, and delete layers (⇒ *Layer Properties subdialog – Tool bar* on page 5-148).

- Layer list in the middle part of the dialog to define number and name of the layers as well as the layout of the structure elements placed in the layer (⇒ *Layer Properties dialog – Layer list* on page 5-149).
- Color and pattern bar to choose a color and pattern for border and filling of the structure elements,
- Command bar to apply the settings and define the patterning order (⇒ *Layer Properties dialog – Command bar* on page 5-149).

4.7.1 Layer Properties subdialog – Tool bar

The control elements have the following functions:

Control element	**Function**
	To open a LAY file from another GDSII database – e.g to take over these properties.
	To store the current properties to the opened LAY file.
	To store current properties to another LAY file, for example to attach current properties to another GDSII database.
 **Save as default**	To use current properties for all new created GDSII databases.
 	To place a selected layer at the top or the end of the list.
 	To place a selected layer over the next or under the following layer in the list.
	To add a layer with the number of the adjacent list box.
	To remove a selected layer.

4.7.2 Layer Properties dialog – Layer list

The columns and symbols have the following functions:

Column/button	Function
ID	Indicates the number of the layer.
Active	To activate or deactivate the corresponding layer from the **GDSII Layer** add-on window. The two small boxes button indicates, if the corresponding layer contains GDSII elements.
Border	To define a border color for the structure elements placed in the corresponding layer.
Fill	To define a fill color for the structure elements placed in the corresponding layer.
Pattern	To define a fill pattern for the structure elements placed in the corresponding layer
Preview	Displays the final layout of the structure elements in the corresponding layer.

4.7.3 Layer Properties dialog – Command bar

The commands have the following functions:

Command	Function
Use physical layer order	To display the layers in their physical order, i.e. later processed layers overlay previous processed layers. Physical order means that a layer at the end of the list is processed first while a layer at the top of the list is processed later. Hence the order is interpreted as a sequence of process steps.
Apply	To store the current properties to LAY file.
Cancel	To close the dialog without overtaking new settings.
OK	To close the dialog and apply new settings.

5 Mouse and keyboard commands

Within the editor selecting and unselecting of single elements is also possible by using the mouse. For selection, click anywhere within the element area. Selection of another element by mouse click will unselect the already selected elements. To unselect all elements, click within a free area. If a single element is selected, information about type, layer and dose is shown within the status bar.

Selecting and unselecting of multiple elements by using the mouse is possible in two ways. Point directly to any corner of the area covering all elements to be selected, press and hold the Shift key and click the left mouse button. Releasing the Shift key lets the user define now the second corner of an area, holding it down the first point defines the center of an area. Or select one element after another while pressing and holding the Control key. In addition, one may use any combination of mouse click selection and Select respectively, Unselect commands. If multiple elements are selected, the status bar shows the number of selected elements.

Within the editor and the viewer, zooming can be done using the wheel of a corresponding mouse.

If the crosshair is used as mouse pointer its movement is also done on the meshes defined by the design grid. If the default mouse pointer is used, the mouse can be placed also in between. The placement of the GDSII elements is not effected.

The following tables give a summary of all available mouse and keyboard commands within the GDSII database, the viewer as well as the editor window.

5.1 GDSII Database

The GDSII Database will be explained in the following sections.

5.1.1 Mouse commands in GDSII Database dialog

Table 5-13: Mouse commands in GDSII database dialog

Action	Key(s) + Mouse Button	Functionality
Singe click	none + left	Select a structure.
Singe click	none + right	Open the right mouse menu.
Double click	none + left	Open the viewer window.
Double click	**Control** + left	Open the editor window.

5.2 Context menu commands in GDSII Database dialog

The commands of the context menu are mainly listed under the **Edit** menu. There is only one additional command:

Command	Function
Mark/unmark favorite	Highlights a structure in the GDSII Database with an additional star button. **Note:** The command could not be apply to referenced structures.

5.2.1 Keyboard shortcuts in GDSII Database dialog

Table 5-14: Available GDSII database keyboard shortcuts.

Key	Functionality
F2	To rename a structure.
Control + C	To copy a structure.
Control + N	To create a new structure within a GDSII database.
Control + O	To open a new GDSII file.
Control + V	To paste a structure into GDSII database.
Control + Shift + N	To create a new GDSII database.
Control + Shift + S	To save GDSII database into a new file.

5.3 GDSII Viewer

The GDSII Viewer will be explained in the following sections.

5.3.1 Mouse commands in viewer window

Table 5-15: Mouse commands in viewer window.

Action	Key(s) + Mouse Button	Functionality
Singe click	none + left	Repeat last command.

Table 5-15: Mouse commands in viewer window.

Action	Key(s) + Mouse Button	Functionality
Singe click	Control + right	Drive to location.

5.3.2 Keyboard shortcuts in viewer window

Table 5-16: Available GDSII viewer keyboard shortcuts.

Key	Functionality
+	To zoom in by a factor of 2.
-	To zoom out by a factor of 2.
=	To view all structure elements.
*****	To increase the design grid by factor 2.
/	To decrease the design grid by factor 2.
a	To display GDSII element properties.
e	To measure dimensions in editor or viewer.
f	To fill out the elements, toggle mode.
g	To show / hide the design grid, toggle mode.
h	To move viewed area.
l	To open layer selection box, toggle mode.
t	To open toolbox , toggle mode.
w	To open working area editor, toggle mode.
z	To zoom in and out, toggle mode.
Shift + arrow keys	To scroll the display into direction of arrow.
Alt + i	To increase hierarchy by one step.

Table 5-16: Available GDSII viewer keyboard shortcuts.

Key	Functionality
Alt + d	To decrease hierarchy by one step.
Alt + o	To show overlays, toggle mode.
Alt + r	To redraw the display.
Del	To delete an annotation.

5.4 GDSII Editor

The GDSII Editor will be explained in the following sections.

5.4.1 Mouse commands in editor window

Table 5-17: Mouse commands in editor window.

<format>	Result	Description
Singe click	none + left	On an unselected structure element: Select that element and unselect all other elements Anywhere else: Unselect all elements Related menu command: **Edit ➡ Unselect ➡ All**
Singe click	Control + left	On an unselected structure element: Select that element additionally (without unselecting all other elements) On a selected structure element: Unselect that element only (without unselecting all other elements)
Singe click	Shift + left	Select all elements additionally within a frame to be dragged (without unselecting all other elements - but only for active layer) Related menu command: **Edit ➡ Select ➡ In**
Singe click	none + right	Repeat the last command
Singe click	Control + right	Drive to location
Double click	none + left	On a structure element: select that element, unselect all other elements and open a dialog box to modify the attributes of that element Related menu command: **Modify ➡ Attributes**

Table 5-17: Mouse commands in editor window.

<format>	Result	Description
Drag over the area you want to select	none + left	Selecting structures inside the field of selection
	Control + left	Hold down the Control key as you drag to constrain the selection marquee to a square.
	Shift + left	To drag a marquee from the center, hold down Shift after you begin dragging.
	Alt + left	Holding Alt key on starting a selection frame keeps the currently selected elements. Keeping Alt key pressed on ending the selection frame removes the surrounded elements from current selection. Releasing Alt key before ending the selection frame adds the surrounded elements to the selection.

5.4.2 Keyboard shortcuts in editor window

Table 5-18: Available GDSII editor keyboard shortcuts.

Key	Functionality
+	To zoom in by a factor of 2.
-	To zoom out by a factor of 2.
=	To view all.
c	To add a circle.
d	To duplicate the selected elements.
e	To measure dimensions in editor or viewer.
f	To fill out the elements, toggle mode.
g	To show / hide the design grid, toggle mode.
h	To move viewed area.

Table 5-18: Available GDSII editor keyboard shortcuts.

Key	Functionality
l	To open the layer selection box, toggle mode.
m	To move the selected elements.
o	To modify the order of elements.
p	To add a polygon.
r	To add a rectangle.
t	To open the toolbox, toggle mode.
w	To open working area editor, toggle mode.
z	To zoom in and out, toggle mode.
Del	To delete the selected elements.
Tab	To select the next element.
Shift + Tab	To select the previous element.
Shift + arrow keys	To scroll the display into direction of arrow.
Control + a	To select all elements.
Control + b	To show the base pattern in MBMS elements.
Control + c	To copy the selected elements.
Control + s	To save the structure.
Control + v	To paste the previously copied elements.
Control + x	To cut the selected elements.
Control + z	To undo last editing command.
Control + y	To redo last undo command.
Alt + i	To increase hierarchy by one step.
Alt + d	To decrease hierarchy by one step.

Table 5-18: Available GDSII editor keyboard shortcuts.

Key	Functionality
Alt + o	To show overlays, toggle mode.
Alt + r	To redraw the display.
ESC	To cancel a selected tool mode – e.g. rectangle tool.

6 Drag and drop

A summary of drag and drop operations involving GDSII structures are listed below.

Table 5-19: GDSII drag and drop functionality.

Source	Object	Target	Action
GDSII Database	Structure	Image	Place structure in Image and generate complete data line in related Positionlist
GDSII Database	Structure	Positionlist	Generate data line in Positionlist; read and use actual sample coordinates as object location (origin).
GDSII Database	Structure	Wafermap	Place structure in Wafermap and generate complete data line in related Positionlist for automated patterning routines.
GDSII Database	Structure	Program desktop	Delete structure.
GDSII Database	Structure	GDSII Database	Copy selected structure to current database. The original structure will not be overwritten.

Chapter 6
NanoPECS (option)

This chapter contains

- A collection of self-interaction formulae for circles, lines and rectangles (⇒ *Self-interaction formulae* on page 6-17). You can use the self-interaction formulae to:
- Derive values for the width and diameter of lines and points in order to counteract their self-interaction during a patterning (intraspecific proximity effects) (⇒ *Dot + SPL Setup subdialog* on page 6-72),
- View the correction functions for rectangular elements on which the function **Use Error Functions for Boxes** in the **Proximity Function** subdialog is based (⇒ *Proximity Effect Function subdialog* on page 6-70).

An example of a contrast curve list (⇒ *Example of contrast curve list* on page 6-21). You can import a contrast curve as a text file into the **EDS - Cross Section** subdialog and use it to calculate the resist profile to be expected during the energy density simulation (⇒ *EDS - Cross Section subdialog* on page 6-88).

A Functional Description

This chapter gives you an overview of how the proximity effect works in electron beam NANOSUITE and how to compensate for it with appropriate correction procedures.

1 The proximity effect in electron beam lithography

The proximity effect is a phenomenon which causes particularly strong interference in electron beam lithography if one wishes to generate uniform nanostructures or very tightly arranged structures of widely varying sizes. During a patterning, beam broadening due to electron scattering in the material causes overexposure and underexposure of individual structures. This is especially the case where high acceleration voltages (>10 kV) are involved. The scattering of the electrons of the electron beam is the result of interactions with the electron shells and with atoms of the resist or substrate.

(⇒ Figure 6-1 on page 6-2) shows the scattering processes within a sample if it is exposed to an electron beam:

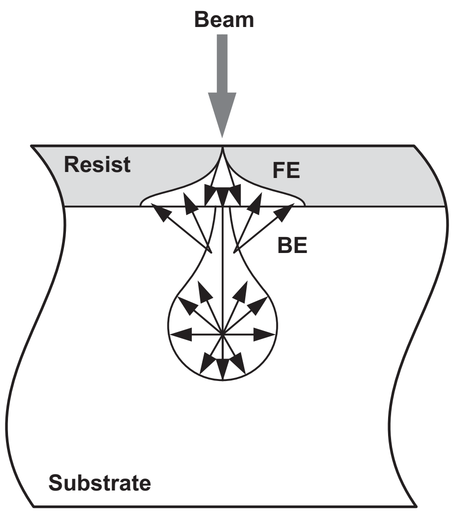


Figure 6-1: Scattering processes during electron beam patterning of a sample

When the primary electrons (FE) penetrate the resist, they are scattered forwards due to the interaction with the resist electrons. This causes the first beam broadening. When the electrons penetrate the substrate below it, they are scattered back to the lattice atoms of the substrate, broadening the beam again. These back-scattered electrons (BE) are responsible for most of the proximity effect.

Mathematically, the actual energy distribution by the primary and back-scattered electrons can be described approximately as the sum of two Gaussian distributions. The combination of these Gaussian functions is called a proximity function.

Depending on the design and the sample, the proximity effect may be intraspecific or interspecific:

- There is an intraspecific proximity effect if, for example, isolated structures of varying width are exposed to the same dose value when compared to larger elements. There is evidence of underexposure in the case of smaller structures as there is less backscatter here. This is particularly the case with spots.
- An interspecific proximity effect arises when the distance between neighboring structures is below a certain minimum. This causes overexposure due to mutual exposure of the neighboring elements. Experience has shown that elements positioned in the middle of a design are subject to more exposure than those at the edges due to this additional exposure.

The actual energy distribution after patterning thus deviates from the energy distribution prescribed in the design and assigned to the individual structures in the form of dose factors (target energy distribution).

This means that the proximity effect results in imprecise implementation of a design on a sample to be structured – edges designed to be steep are rounded-off. For example (⇒ Figure 6-2 on page 6-3) shows an example of the proximity effect on two element groups of a design after simulated patterning:

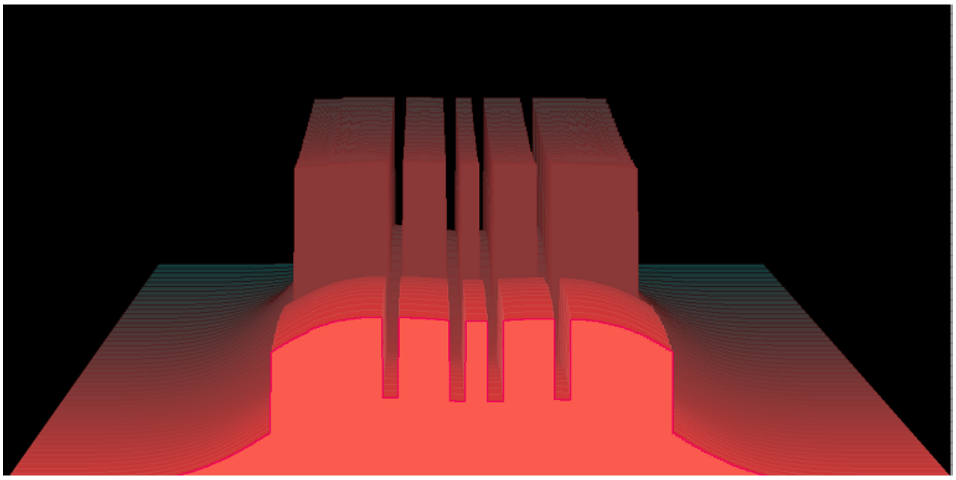


Figure 6-2: Cross-sectional view of proximity effects after simulated patterning

The rear element group shows the desired structure. The front element group shows the structuring result after the simulated patterning. The proximity effect has made the structure lower overall and has rounded it off at the edges.

If a design is to be implemented on a sample as precisely as possible, the proximity effect must be compensated before patterning by adjusting the energy distribution. This requires comprehensive and compute-intensive processing and corrections to the design and its dose values. This correction process is known as proximity effect correction (PEC).

2 Principle of proximity effect correction

The purpose of a proximity effect correction is to adjust the target energy distribution of a design as precisely as possible to the actual energy requirement of individual structures. The energy distribution resulting from this adjustment is referred to hereinafter as the corrected energy distribution.

For a proximity effect correction, it is necessary

- To simulate broadening of the electron beam in the concrete material situation, e.g. by means of a Monte Carlo simulation, and thus to determine the energy distribution being deposited along the electron trajectory.
- To determine the parameters of a proximity function. This may take place by means of fitting the simulated or measured energy distribution in resist.
- To divide the large structures of a design into smaller pieces to which separate corrected energy doses are assigned.
- To set the dose factors to individual parts in accordance with the actual energy requirement.

The comprehensive parameters needed for correction are stored in a special dataset called the proximity dataset and saved in a database called the proximity database.

The proximity effect correction for a design is carried out concretely by selecting individual elements or complete levels of a design in the GDSII editor of the NANOSUITE software and applying a corresponding proximity dataset to it. After the proximity effect correction, an energy density simulation (EDS) can be used to simulate additionally the patterning of the corrected design and to check the result of the correction.

The requirements for a proximity effect correction are thus:

- A design containing the draft of a structure to be exposed.
- A target energy distribution which has been assigned to the design in the form of dose factors.
- A definition of an electron beam which is to be used to carry out the patterning.
- A definition of the resist and substrate material.
- A proximity database with parameters for
  - the layer structure of the probe,
  - the description of the broadening of the electron beam in the material (proximity parameter),
  - the fineness of the correction required,
  - the identification and classification of the data set.

The parameters may refer to a real sample to be structured, or to a hypothetical sample which is to be used as a template for frequently structured types of sample.

(⇒ Figure 6-3 on page 6-5) shows an example of energy distribution in the case of both corrected and uncorrected element groups of a design:

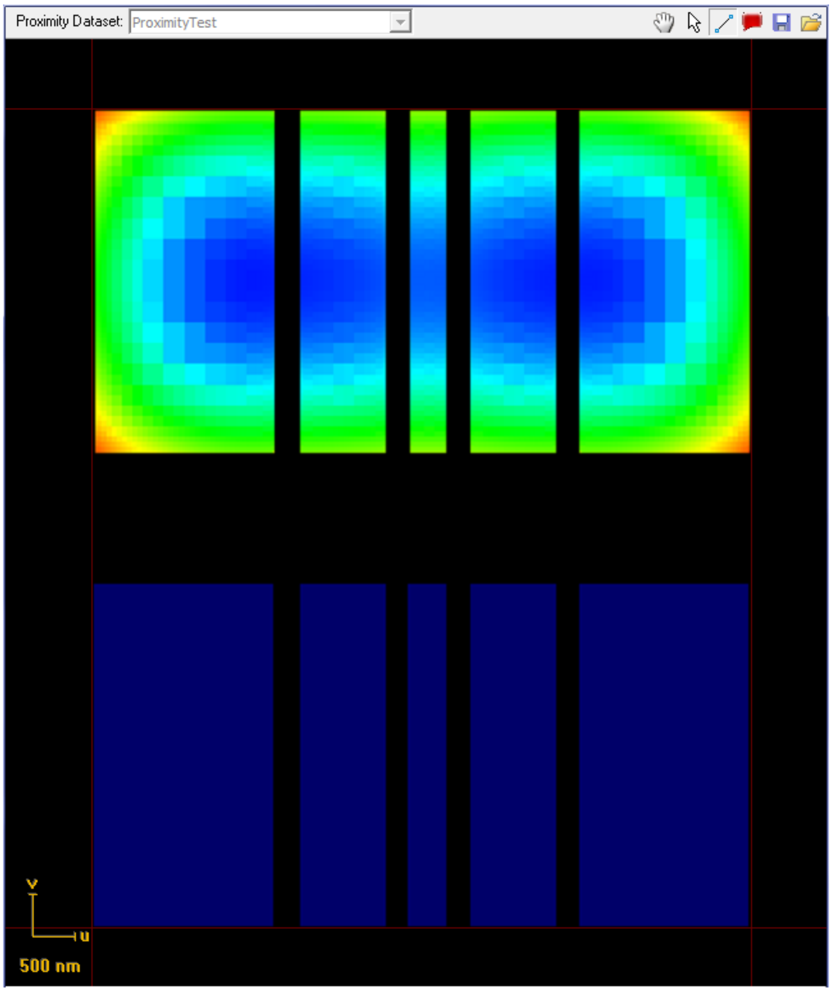


Figure 6-3: Energy distribution of corrected and uncorrected element groups of a design

Each element group is made up of five rectangles. The upper element group was corrected with NanoPECS™. The color tones indicate an unhomogenous input energy distribution across the element group. This distribution corresponds to the actual energy requirement: In the blue areas, the energy deposition is lowest due to the large proximity effect between individual elements. The energy deposition is highest in the red areas at the edge of the element group. The proximity effect is strongest here.

Furthermore, the upper element group shows that the five rectangles have been broken down into many small fracturing shapes to allow the required energy dose course needed to be assigned as precisely as possible.

The lower element group was not corrected. The even dark blue coloring indicates an homogenous distribution of energy across all elements of the element group. It corresponds to the target input energy distribution of the design.

3 Functional description for NanoPECS

In this section, you will learn how

- NanoPECS™ is integrated into the NANOSUITE software,
- to correct the proximity effect with NanoPECS™,
- the data needed for the proximity effect correction is managed.

3.1 Structure and integration

NanoPECS™ is a command line program which is integrated into the GDSII module of the NANOSUITE software.

NanoPECS™ is made up of a database and three dialogs. The database contains the proximity datasets (⇒ *Proximity database* on page 6-14). You
use the dialogs to edit the proximity datasets and to carry out the proximity effect correction and the energy density simulation (EDS).

(⇒ Figure 6-4 on page 6-7) shows an overview of the structure and the integration of NanoPECS™ as well as the interaction between individual components:

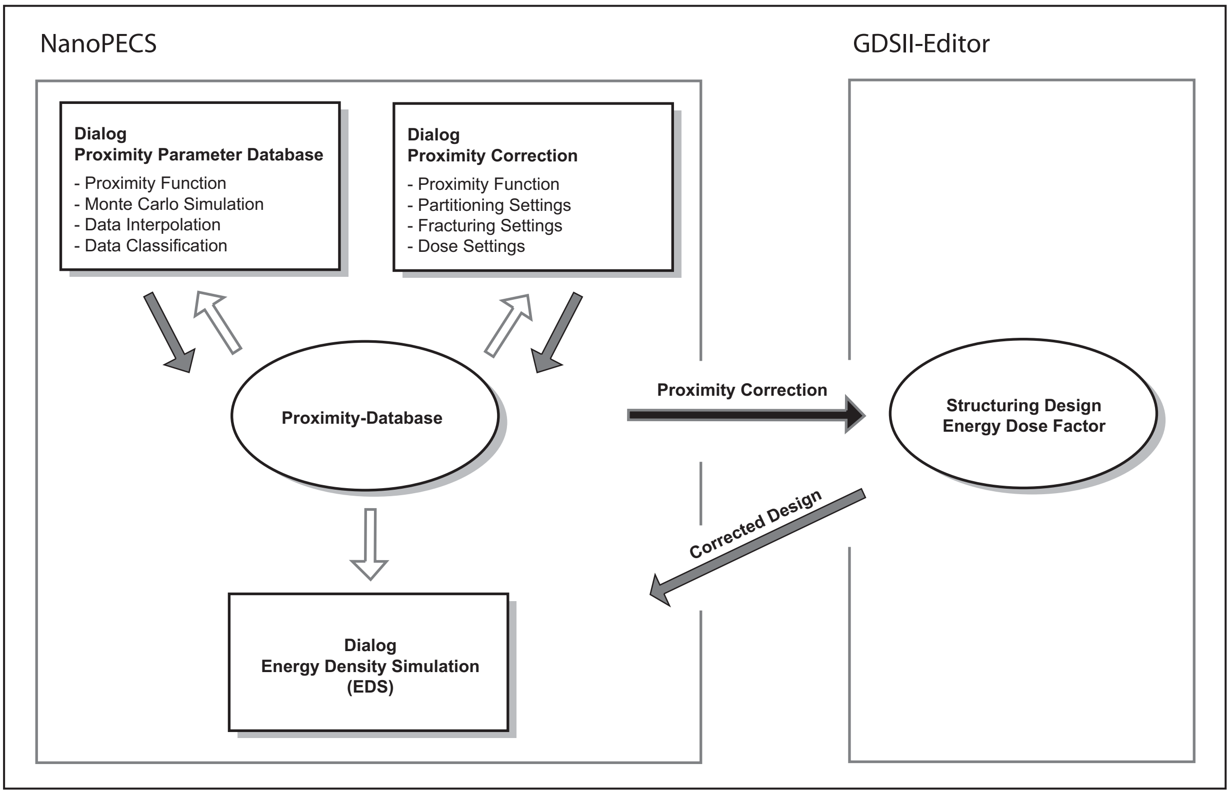


Figure 6-4: Structure and integration of NanoPECS™ into the *NanoSuite* software

You can find out how to work with the GDSII editor in the overall documentation for the relevant NANOSUITE system.

The NanoPECS™ dialogs have the following functions:

- **Proximity Parameter Database** dialog for
  - creating, editing and managing proximity datasets,
  - editing the proximity function,
  - executing simulations of the electron trajectories

  (⇒ *Proximity Parameter Database dialog* on page 6-30).
- **Proximity Effect Correction** dialog for
  - executing the proximity effect correction,
  - editing the proximity function,
  - creating and editing proximity datasets,
  - determining parameters for partitioning and fracturing the elements of the design

  (⇒ *Proximity Correction dialog* on page 6-62).
- **EDS - <Subdialog>** dialog (energy density simulation) for

– simulating the patterning of a corrected or uncorrected design,
– checking the patterning results,
– optimizing the selection of the resist

(⇒ *EDS - <Subdialog> dialog* on page 6-81).

You can use the dialogs **Proximity Parameter Database** and **Proximity Correction** to control the proximity database equally and to edit proximity datasets. Take these interactions into account when editing datasets.

You can call up the NanoPECS™ dialogs using the **Proximity** menu in the menu bar of the NANOSUITE software.

Table 6-1: Menu entries in the **Proximity** menu

Menu	Menu entry	Dialog
Proximity	**Proximity Parameter Database ...**	**Proximity Parameter Database**
	Correction ...	**Proximity Correction**
	Energy density simulation ...	**EDS - GDSII Design**

The commands in the **Proximity** menu are only active when you are within the GDSII module editor of the NANOSUITE software.

NanoPECS™ also contains interfaces to the following dialogs which you can use during a proximity correction:

- **Duplicate Elements** dialog to generate a copying matrix with the dose scaling for neighboring elements. You can call up the dialog by selecting
  Edit ➡ **Duplicate...** in the menu bar of the NANOSUITE software.
- The **Dose Colors** dialog is for assigning colors for the individual doses in the GDSII editor. You can call up the dialog by selecting **Options** ➡ **Dose colors** in the menu bar of the NANOSUITE software.

NanoPECS™ is a command line program and, as such, can be controlled independently of the dialogs using a command line interface. If you would like to use this operating option, contact Raith support for further information.

3.2 Proximity correction with NanoPECS

Proximity correction with NanoPECS™ consists of five steps. Figure below shows an overview of these steps:

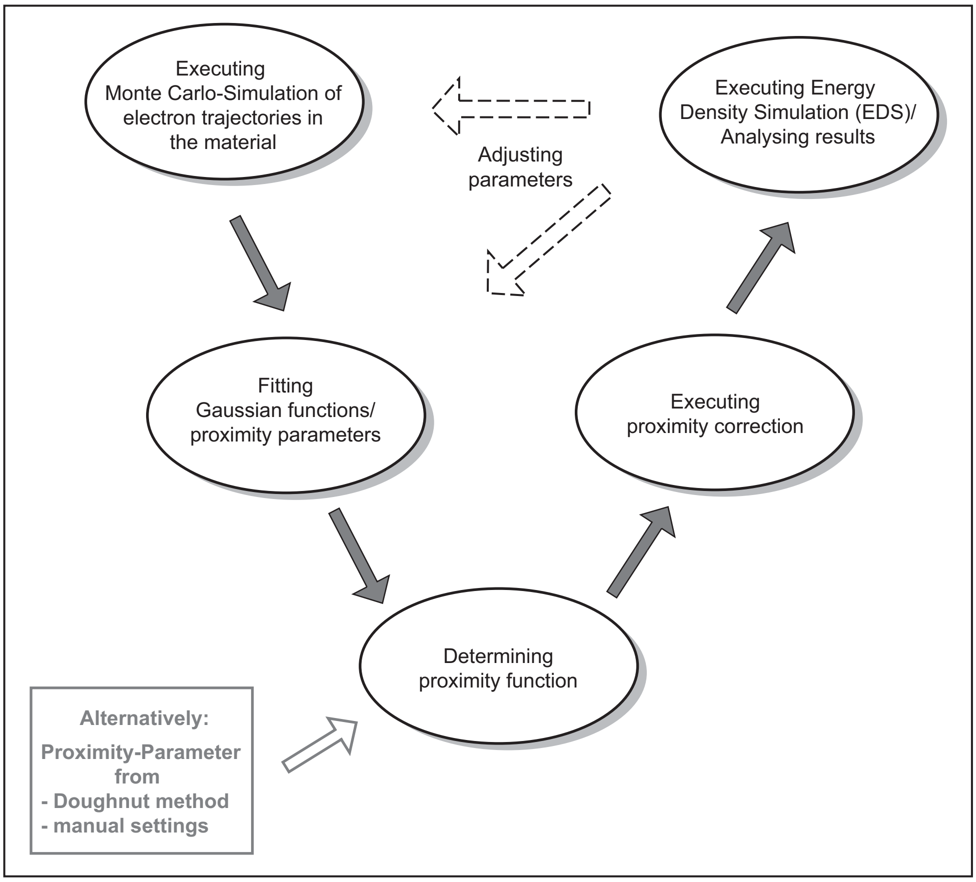


Figure 6-5: Proximity correction steps with NanoPECS

The individual steps are explained in more detail in the following sections:

3.2.1 Executing a Monte Carlo simulation of electron trajectories in the material

The Monte Carlo simulation (MC simulation) in general is a process used to calculate probabilities of individual events and is based on stochastic simulation algorithms. When correcting the proximity effect, it can be used to simulate the electron trajectories of the primary and backscattered electrons of a hypothetical electron beam in a given sample, thus enabling calculation of the distribution of energy deposited along the electron trajectories. One direct result of the MC simulation is thus the energy distribution which arises at a defined depth within the resist, depending on the distance from the center of the hypothetical electron beam.

You can fit this distribution using a spectrum of circular Gaussian and exponential functions. You can then adopt the fit parameters of the Gaussian and exponential functions into a proximity dataset directly as basic proximity parameters (⇒ *Determining a Proximity function* on page 6-10).

The MC simulation is, for the most part, executed based on the following parameters:

- Acceleration voltage and electron beam shape,
- Material and layer structure of the sample,
- Layer depth in the resist from which the energy distribution is determined as a basis for fitting the proximity parameters.

You can execute the MC simulation using the **Proximity Parameter Database** dialog – **Data Simulation** subdialog (⇒ *Data Simulation subdialog* on page 6-37).

You can execute an MC simulation based on existing proximity datasets. Alternatively, you can redefine these parameters as part of the simulation and execute the MC simulation.

3.2.2 Fitting Gaussian functions

At the end of the MC simulation, the parameters of four Gaussian functions are calculated as the basis of a proximity dataset:

- Double Gaussian function,
- Double Gaussian function with exponential term,
- Triple Gaussian function,
- Triple Gaussian function with exponential term.

You can monitor and analyze the quality of the simulated Gaussian functions visually. To this end, a graphic interface is available which you can use to check which of the four Gaussian functions reproduces the energy distribution in the resist most closely. You can use the parameters of the closest Gaussian functions to define the proximity function.

You can carry out the graphic analysis in the **Simulation Visualizer** (⇒ *Simulation Visualizer* on page 6-53).

3.2.3 Determining a Proximity function

The proximity function contains the important parameters for correcting the target energy distribution of the design. In its simplest form it represents a double Gaussian function and describes the distribution of primary and back-scattered electrons in a sample after an actual or simulated patterning. The following equation shows the proximity function as a double Gaussian function:

$$f = \frac{1}{\pi(1+\eta)}\left(\frac{1}{\alpha^2}e^{-\frac{r^2}{\alpha^2}} + \frac{\eta}{\beta^2}e^{-\frac{r^2}{\beta^2}}\right)$$

The parameters α, β and η depend on the acceleration voltage, the material and the layer thickness of the resist, as well as the material of the substrate. They can be determined through experimentation. The first addend of the double Gaussian function describes beam broadening caused by falling primary electrons, the second addend that caused by backscattered electrons.

Depending on the target energy distribution of the design, you can add an exponential term and/or a third Gaussian term to the Gaussian function. You can adopt the parameters of the extended Gaussian functions from the MC simulation or define them manually. The following equation shows the proximity function as an expanded Gaussian function:

$$f = \frac{1}{\pi(1+\eta)}\left(\frac{1}{\alpha^2}e^{-\frac{r^2}{\alpha^2}} + \frac{\eta}{\beta^2}e^{-\frac{r^2}{\beta^2}} + \frac{\nu}{2\gamma^2}e^{-\frac{r}{\gamma}} + \frac{\nu_2}{\gamma_2^2}e^{-\frac{r^2}{\nu_2^2}}\right)$$

You can adopt the parameters of the simulated Gaussian functions into the proximity function and create a new proximity dataset accordingly.

As an alternative to the MC simulation, you can define the proximity function directly

- by experimentation – the Doughnut method, for example,
- by entering the individual proximity parameter methods manually based on empirical values or values from specialist literature.

You can edit the proximity function using the following dialogs:

- **Proximity Parameter Dataset** dialog – **Edit Dataset** subdialog (⇒ *Edit Dataset subdialog* on page 6-33),
- **Proximity Correction** dialog – **Proximity Function** subdialog (⇒ *Proximity Effect Function subdialog* on page 70).

3.2.4 Executing a Proximity correction

You can carry out a proximity correction by selecting elements to be corrected or complete levels of a design in the GDSII editor and apply a proximity dataset to them. You can execute the proximity correction using the **Proximity Correction** dialog (⇒ *Proximity Correction dialog* on page 6-62).

During a proximity correction with NanoPECS™, a design is broken down step-by-step into fracturing shapes of different types and sizes. Then corrected dose factors are set to the individual fracturing shapes. NanoPECS™ does this in six steps:

- Partitioning of the design (design partitions),
- Fracturing of all elements of the partition,

- Repartitioning of the fractured elements using the main memory options (memory partitions),
- Setting up of a proximity effect matrix (PEM) for each partition,
- Calculating the corrected dose factors for the element splitters by inverting the PEM,
- Adopting the calculation results into the starting design.

Partitioning the design

During the partitioning, the design is divided into two types of partition:

Partition	Description
Design partitions	Design partitioning takes place based on the design and non-interactive clusters are created. Non-interactive clusters are areas between which no interaction occurs during patterning because they are beyond the scattering range of the secondary electrons.
Memory partitions	Memory partitioning divides the interactive clusters into partitions which are small enough to allow matrix inversion of this partition to be executed later in the main memory of the computer. Areas of overlap are created for the interactive clusters to take the interactions at the division boundaries into account.

You can make partitioning settings using the dialog **Proximity Correction** – Subdialog **Partitioning Setup** (⇒ *Partitioning Setup subdialog* on page 6-76).

Fracturing elements of the design

The partitions continue to be fractured into fracturing shapes of varying sizes while the elements are broken down. Their size is related to how great the need to correct elements of the design is. The fracturing process is based on a recursive deviding algorithm. A routine fractures the partitions using the expected energy dose variations within the elements into small shapes at the edges and rougher shapes in the middle of the element.

(⇒ Figure 6-6 on page 6-13) shows an example of unfractured and fractured elements of a design after a partial proximity correction with NanoPECS™:

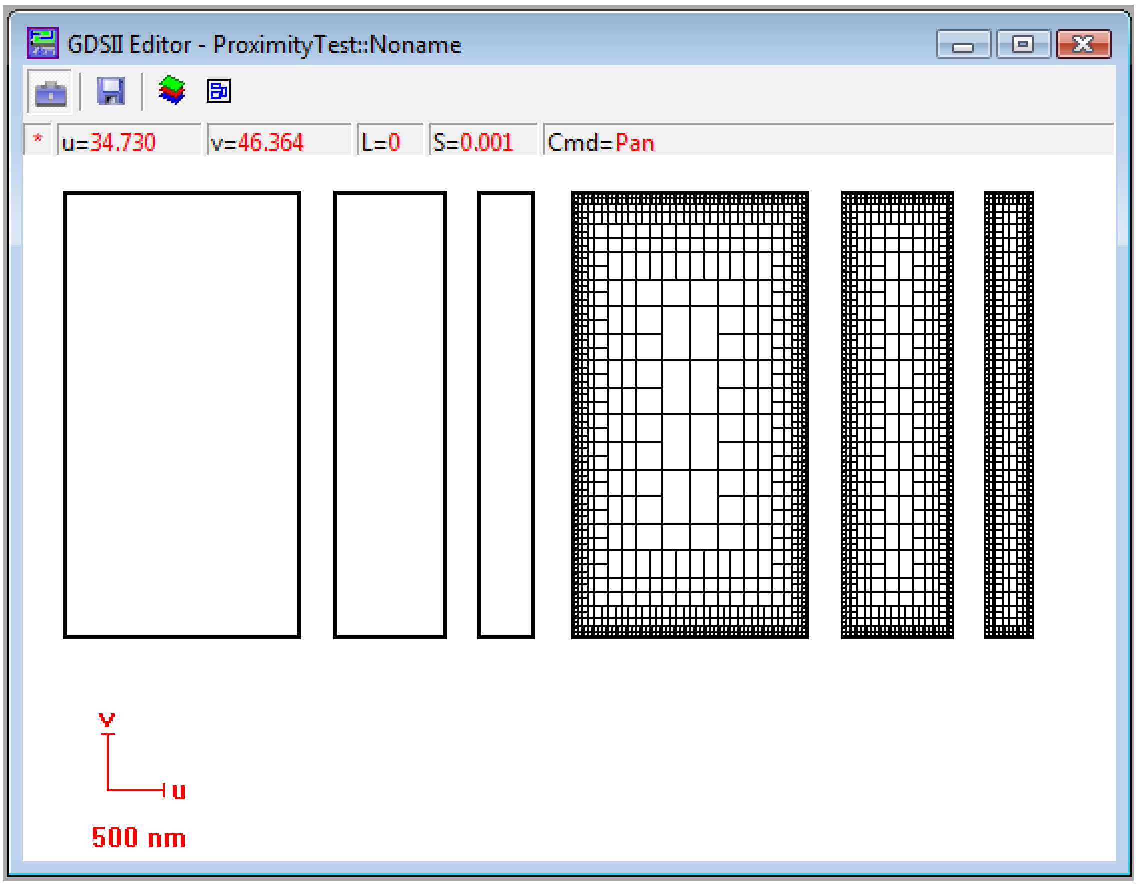


Figure 6-6: An example of unfractured (left) and fractured (right) elements of a design after a partial proximity correction with NanoPECS

Due to the stronger energy dose variations, there is finer fracturing at the edge than in the middle of the design. This keeps the overall number of fracturing shapes generated as low as possible and guarantees high correction precision, requiring a minimum of calculation and storage.

You can make settings for the type and degree of fracturing using the dialog **Proximity Correction** – Subdialog **Fracturing Settings** (⇒ *Fracturing Settings subdialog* on page 6-74).

Setting of dose factors

After partitioning and dispersing the elements of the design, the proximity effect matrix (PEM) is created. In a following step, the PEM is inverted using an iterative solver and the corrected dose factors are set to the individual fracturing shapes.

The following parameters are included in the calculations:

- Mutual and self-interaction of the elements (interspecific and intraspecific proximity effects),
- Lateral extension and initial dose factor for each element,

- Distance between the elements.

With regard to the precision of the proximity effect correction:

The finer the fracturing of the elements, the more precise the correction result will be. A too fine degree of scattering, however, results in a very high volume of data and requires a correspondingly high computing capability during inversion of the PEM. Furthermore, the patterning time for a design made up of many small fracturing shapes increases significantly when compared with a design with few elements.

Depending on the computer equipment of your NANOSUITE system, executing a proximity correction may necessitate memory requirements that are so great the the proximity correction is terminated.

3.2.5 Executing an Energy Density Simulation (EDS)

You can simulate the patterning of a corrected design using the energy density simulation (EDS). EDS allows you to

- Check and analyze the result of a proximity correction,
- Select a suitable resist,
- Decide whether a certain resolution is attainable experimentally (lower limit of resolution).

If the desired patterning result cannot be attained with the parameters of the underlying proximity dataset, you must change individual parameters of the proximity dataset – such as acceleration voltage, material composition and layer structure of the sample – and repeat the proximity correction.

An example of the execution of an EDS and the analysis of the results can be seen in section ⇒ *EDS – example investigation* on page 6-22.

You use the following dialogs to execute an EDS:

- **EDS - GDSII Design** subdialog
  (⇒ *EDS - GDSII Design subdialog* on page 6-81)
- **EDS - Cross Section** subdialog
  (⇒ *EDS - Cross Section subdialog* on page 6-88)
- **EDS - Energy Density Simulation** subdialog
  (⇒ *EDS - Energy Density Simulation subdialog* on page 6-86)

4 Proximity database

All proximity datasets which you use for the proximity correction are held in the proximity database. To access these datasets, you use the three NanoPECS™ dialogs to access the datasets.

4.1 Parameters of Proximity datasets

These are the most important parameters included in the proximity datasets:

- Material composition and layer structure of a sample,
- Acceleration voltage, profile and radius of the electron beam,
- Parameters of the proximity function,
- Parameters for partitioning and fracturing the elements of the design.

You can define the parameters in the

- **Proximity Parameter Database** dialog (⇒ *Proximity Parameter Database dialog* on page 6-30),
- **Proximity Correction** dialog (⇒ *Proximity Correction dialog* on page 6-62).

4.2 Classification and nomenclature of Proximity datasets

When you create a proximity dataset, you assign it a classification which identifies it uniquely. In particular, the classification comprises parameters which are needed for recognition of the purpose of this dataset – such as

- User
- Layer structure and material
- Sample name
- Acceleration voltage

These parameters are not required for the proximity effect correction itself. However, they are indispensable for targeted definition of the patterning conditions.

You can adopt the classification completely or in part into the dataset name by clicking onto the calculator button right to the input box of the Dataset name. You can also name a proximity dataset individually.

The standard procedure is that a specification is prefixed to the dataset name which identifies the source which was used to create the proximity dataset.

The following specifications are possible:

Table 6-2: Dataset types in the proximity database

Button	Specification	Description/source
	**<recommended default parameters>**	Standard datasets with predefined parameters

Table 6-2: Dataset types in the proximity database

Button	Specification	Description/source
	Levermann Test ...	Datasets which have been assigned a user-defined specification
	Advice for ...	Datasets which are available for selection in the NanoPECS™ Wizard Guide. These datasets each contain a predefined combination of voltage and material.
	Monte Carlo: ...	Datasets which were generated using the Monte Carlo simulation
	Interpolation: ...	Datasets which were generated using interpolation.

You can classify the datasets using the **Edit Proximity Dataset Classification** dialog (⇒ *Edit Proximity Dataset Classification dialog* on page 6-47).

4.3 Bubble-Help function

NanoPECS™ offers a bubble-help function to all of the dialogs and subdialogs. The bubble-help function is a context sensitive help, which gives you important information and input recommendations for values to the single input fields. The help is indicated if you move the cursor above the field.

(⇒ Figure 6-7 on page 6-16) shows an example of a bubble-help:

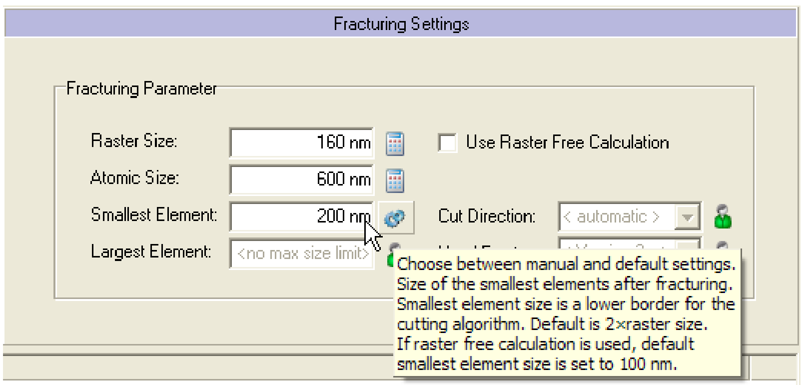


Figure 6-7: Bubble-help for field **Smallest Element** in **Proximity Correction** dialog - **Fracturing Settings** subdialog

5 Self-interaction formulae

Self-interaction for circles

Proximity function:

$$f(r)=\frac{1}{\pi\left(1+\eta+\nu+\nu_2\right)}\left[\frac{1}{\alpha^2}e^{-\frac{r^2}{\alpha^2}}+\frac{\eta}{\beta^2}e^{-\frac{r^2}{\beta^2}}+\frac{\nu_2}{\gamma_2^2}e^{-\frac{r^2}{\gamma_2^2}}+\frac{\nu}{2\gamma^2}e^{-\frac{d}{\gamma}}\right] \tag{1}$$

Self-interaction given by integration of Proximity function over circle radius Q:

$$SI_{\text{circle}}=\int_0^{2\pi}\int_0^{Q} f(r)\,rdrd\theta \tag{2}$$

$$SI_{\text{circle}}=2\pi\int_0^{Q} f(r)\,rdr \tag{3}$$

$$SI_{\text{circle}}=2\pi\int_0^{Q}\frac{1}{\pi\left(1+\eta+\nu+\nu_2\right)}\left[\frac{1}{\alpha^2}e^{-\frac{r^2}{\alpha^2}}+\frac{\eta}{\beta^2}e^{-\frac{r^2}{\beta^2}}+\frac{\nu_2}{\gamma_2^2}e^{-\frac{r^2}{\gamma_2^2}}+\frac{\nu}{2\gamma^2}e^{-\frac{r}{\gamma}}\right]rdr \tag{4}$$

$$SI_{\text{circle}}=\frac{2}{1+\eta+\nu+\nu_2}\left\{\frac{1}{\alpha^2}\int_0^{Q}e^{-\frac{r^2}{\alpha^2}}rdr+\frac{\eta}{\beta^2}\int_0^{Q}e^{-\frac{r^2}{\beta^2}}rdr+\frac{\nu_2}{\gamma_2^2}\int_0^{Q}e^{-\frac{r^2}{\gamma_2^2}}rdr+\frac{\nu}{2\gamma^2}\int_0^{Q}e^{-\frac{r}{\gamma}}rdr\right\} \tag{5}$$

Using:

$$\int_0^{Q} re^{-\frac{r^2}{a^2}}dr=\frac{a^2}{2}\left(1-e^{-\frac{Q^2}{a^2}}\right)\ \text{and}\ \int_0^{Q} re^{-\frac{r}{\gamma}}dr=\gamma\left(\gamma-\gamma\left(1+\frac{Q}{\gamma}\right)e^{-\frac{Q}{\gamma}}\right)$$

Eq. (5) yields:

$$SI_{\text{circle}}=\frac{1}{1+\eta+\nu}\left\{1-e^{-\frac{Q^2}{\alpha^2}}+\eta\left(1-e^{-\frac{Q^2}{\beta^2}}\right)+\nu_2\left(1-e^{-\frac{Q^2}{\gamma_2^2}}\right)+\nu\left[1-\left(1+\frac{Q}{\gamma}\right)e^{-\frac{Q}{\gamma}}\right]\right\} \tag{6}$$

Using $A=\pi\cdot Q^2$

$$\boxed{SI_{\text{circle}}=\frac{1}{1+\eta+\nu+\nu_2}\left\{1-e^{-\frac{A}{\pi\cdot\alpha^2}}+\eta\left(1-e^{-\frac{A}{\pi\cdot\beta^2}}\right)+\nu_2\left(1-e^{-\frac{A}{\pi\cdot\gamma_2^2}}\right)+\nu\left[1-\left(1+\frac{1}{\gamma}\sqrt{\frac{A}{\pi}}\right)\cdot e^{-\frac{1}{\gamma}\sqrt{\frac{A}{\pi}}}\right]\right\}} \tag{7}$$

Self-interaction for lines

Proximity function:

$$f_{\mathrm{SPL}}(x)=\frac{1}{\sqrt{\pi}\left(1+\eta+\nu+\nu_2\right)}\left[\frac{1}{\alpha}e^{-\frac{x^2}{\alpha^2}}+\frac{\eta}{\beta}e^{-\frac{x^2}{\beta^2}}+\frac{\nu_2}{\gamma_2}e^{-\frac{x^2}{\gamma_2^2}}+\frac{\nu_1\cdot\sqrt{\pi}}{2\gamma}e^{-\frac{|x|}{\gamma}}\right] \tag{8}$$

Self-interaction given by integration of Proximity function along a line of length L:

$$SI_{\mathrm{SPL}}=\int_{-\frac{L}{2}}^{\frac{L}{2}}f_{\mathrm{SPL}}(x)\,dx \tag{9}$$

$$SI_{\mathrm{SPL}}=\int_{-\frac{L}{2}}^{\frac{L}{2}}\frac{1}{\sqrt{\pi}\left(1+\eta+\nu+\nu_2\right)}\left[\frac{1}{\alpha}e^{-\frac{x^2}{\alpha^2}}+\frac{\eta}{\beta}e^{-\frac{x^2}{\beta^2}}+\frac{\nu_2}{\gamma_2}e^{-\frac{x^2}{\gamma_2^2}}+\frac{\nu\cdot\sqrt{\pi}}{2\gamma}e^{-\frac{|x|}{\gamma}}\right]dx \tag{10}$$

$$SI_{\mathrm{SPL}}=\frac{1}{\sqrt{\pi}\left(1+\eta+\nu+\nu_2\right)}\left\{\frac{1}{\alpha}\int_{-\frac{L}{2}}^{\frac{L}{2}}e^{-\frac{x^2}{\alpha^2}}dx+\frac{\eta}{\beta}\int_{-\frac{L}{2}}^{\frac{L}{2}}e^{-\frac{x^2}{\beta^2}}dx+\frac{\nu_2}{\gamma_2}\int_{-\frac{L}{2}}^{\frac{L}{2}}e^{-\frac{x^2}{\gamma_2^2}}dx+\frac{\nu\cdot\sqrt{\pi}}{2\gamma}\int_{-\frac{L}{2}}^{\frac{L}{2}}e^{-\frac{|x|}{\gamma}}dx\right\} \tag{11}$$

Using:

$$\int_{-\frac{L}{2}}^{\frac{L}{2}}e^{-\frac{r^2}{a^2}}dr=\sqrt{\pi}\cdot a\cdot\mathrm{erf}\left(\frac{L}{2a}\right)\text{ and } SI=\int_{-\frac{L}{2}}^{\frac{L}{2}}e^{-\frac{|x|}{\gamma}}dx=2\cdot\int_{0}^{\frac{L}{2}}e^{-\frac{x}{\gamma}}dx=2\gamma\left(1-e^{-\frac{L}{2\gamma}}\right)$$

Eq. (11) yields:

$$\boxed{SI_{\mathrm{SPL}}=\frac{1}{1+\eta+\nu+\nu_2}\left\{\mathrm{erf}\left(\frac{L}{2\alpha}\right)+\eta\cdot\mathrm{erf}\left(\frac{L}{2\beta}\right)+\nu_2\cdot\mathrm{erf}\left(\frac{L}{2\gamma_2}\right)+\nu\cdot\left(1-e^{-\frac{L}{2\gamma}}\right)\right\}} \tag{12}$$

Self-interaction for rectangle

Keep the exponential term as long as possible and throw it away when it becomes untractable.

Self-interaction: integration of proximity function over rectangle of width a and heigth b:

$$SI_{\mathrm{rectangle}}=\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}}f(x,y)\,dxdy \tag{13}$$

$$SI_{\mathrm{rectangle}}=\frac{1}{\pi\left(1+\eta+\nu+\nu_2\right)}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}}\left[\frac{1}{\alpha^2}e^{-\frac{x^2+y^2}{\alpha^2}}+\frac{\eta}{\beta^2}e^{-\frac{x^2+y^2}{\beta^2}}+\frac{\nu}{2\gamma^2}e^{-\frac{\sqrt{x^2+y^2}}{\gamma}}+\frac{\nu_2}{\gamma_2^2}e^{-\frac{x^2+y^2}{\gamma_2^2}}\right]dxdy \tag{14}$$

$$SI_{\mathrm{rectangle}}=\frac{1}{\pi\left(1+\eta+\nu+\nu_2\right)}\left[\frac{1}{\alpha^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}}e^{-\frac{x^2+y^2}{\alpha^2}}dxdy+\frac{\eta}{\beta^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}}e^{-\frac{x^2+y^2}{\beta^2}}dxdy+\cdots\right.$$

$$SI_{\text{rectangle}} = \frac{1}{\pi(1+\eta+\nu+\nu_2)}\left[\frac{1}{\alpha^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{x^2+y^2}{\alpha^2}}dxdy + \frac{\eta}{\beta^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{x^2+y^2}{\beta^2}}dxdy + \cdots\right.$$
$$\left.\cdots\frac{\nu_2}{\gamma_2^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{x^2+y^2}{\gamma_2^2}}dxdy + \frac{\nu}{2\gamma^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{\sqrt{x^2+y^2}}{\gamma}}dxdy\right] \tag{15}$$

$$SI_{\text{rectangle}} = \frac{1}{\pi(1+\eta+\nu+\nu_2)}\left[\frac{1}{\alpha^2}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{x^2}{\alpha^2}}dx\cdot\int_{-\frac{b}{2}}^{\frac{b}{2}} e^{-\frac{y^2}{\alpha^2}}dy + \frac{\eta}{\beta^2}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{x^2}{\beta^2}}dx\cdot\int_{-\frac{b}{2}}^{\frac{b}{2}} e^{-\frac{y^2}{\beta^2}}dy + \cdots\right.$$
$$\left.\cdots\frac{\nu_2}{\gamma_2^2}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{x^2}{\gamma_2^2}}dx\cdot\int_{-\frac{b}{2}}^{\frac{b}{2}} e^{-\frac{y^2}{\gamma_2^2}}dy + \frac{\nu}{2\gamma^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{\sqrt{x^2+y^2}}{\gamma}}dxdy\right] \tag{16}$$

Integral of Gaussian function:

$$\int_{-\frac{Q}{2}}^{\frac{Q}{2}} e^{-\frac{r^2}{a^2}}dr = a\cdot\sqrt{\pi}\,\text{erf}\left(\frac{Q}{2a}\right) \tag{17}$$

Eq. (21) yields:

$$SI_{\text{rectangle}} = \frac{1}{\pi(1+\eta+\nu+\nu_2)}\left[\frac{1}{\alpha^2}\alpha\sqrt{\pi}\,\text{erf}\left(\frac{a}{2\alpha}\right)\cdot\alpha\sqrt{\pi}\,\text{erf}\left(\frac{b}{2\alpha}\right) + \frac{\eta}{\beta^2}\beta\sqrt{\pi}\,\text{erf}\left(\frac{a}{2\beta}\right)\cdot\beta\sqrt{\pi}\,\text{erf}\left(\frac{b}{2\beta}\right) + \cdots\right.$$
$$\left.\cdots\frac{\nu_2}{\gamma_2^2}\gamma_2\sqrt{\pi}\,\text{erf}\left(\frac{a}{2\gamma_2}\right)\cdot\gamma_2\sqrt{\pi}\,\text{erf}\left(\frac{b}{2\gamma_2}\right) + \frac{\nu}{2\gamma^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{\sqrt{x^2+y^2}}{\gamma}}dxdy\right] \tag{18}$$

$$SI_{\text{rectangle}} = \frac{1}{\pi(1+\eta+\nu+\nu_2)}\left[\pi\,\text{erf}\left(\frac{a}{2\alpha}\right)\cdot\text{erf}\left(\frac{b}{2\alpha}\right) + \eta\pi\,\text{erf}\left(\frac{a}{2\beta}\right)\cdot\text{erf}\left(\frac{b}{2\beta}\right) + \cdots\right.$$
$$\left.\cdots\nu_2\pi\,\text{erf}\left(\frac{a}{2\gamma_2}\right)\cdot\text{erf}\left(\frac{b}{2\gamma_2}\right) + \frac{\nu}{2\gamma^2}\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\frac{\sqrt{x^2+y^2}}{\gamma}}dxdy\right] \tag{19}$$

Now the exponential term makes troubles. Let's throw it overboard…

$$\boxed{SI_{\text{rectangle}} = \frac{1}{1+\eta+\nu_2}\left[\text{erf}\left(\frac{a}{2\alpha}\right)\cdot\text{erf}\left(\frac{b}{2\alpha}\right) + \eta\,\text{erf}\left(\frac{a}{2\beta}\right)\cdot\text{erf}\left(\frac{b}{2\beta}\right) + \nu_2\,\text{erf}\left(\frac{a}{2\gamma_2}\right)\cdot\text{erf}\left(\frac{b}{2\gamma_2}\right)\right]} \tag{20}$$

Square: $a = b$

$$SI_{\text{rectangle}} = \frac{1}{1+\eta+\nu_2}\left[\text{erf}^2\left(\frac{a}{2\alpha}\right) + \eta\,\text{erf}^2\left(\frac{a}{2\beta}\right) + \nu_2\,\text{erf}^2\left(\frac{a}{2\gamma_2}\right)\right] \tag{21}$$

Self-interaction for square using circle approximation:

$$SI_{\text{circle}} = \frac{1}{1+\eta+\nu_2}\left\{1-e^{-\frac{a^2}{\pi\cdot\alpha^2}}+\eta\left(1-e^{-\frac{a^2}{\pi\cdot\beta^2}}\right)+\nu_2\left(1-e^{-\frac{a^2}{\pi\cdot\gamma_2^2}}\right)\right\},\ \nu=0 \tag{22}$$

Rectangle: $b=2a$

$$SI_{\text{rectangle}} = \frac{1}{1+\eta+\nu_2}\left[\operatorname{erf}\left(\frac{a}{2\alpha}\right)\cdot\operatorname{erf}\left(\frac{a}{\alpha}\right)+\eta\operatorname{erf}\left(\frac{a}{2\beta}\right)\cdot\operatorname{erf}\left(\frac{a}{\beta}\right)+\nu_2\operatorname{erf}\left(\frac{a}{2\gamma_2}\right)\cdot\operatorname{erf}\left(\frac{a}{\gamma_2}\right)\right] \tag{23}$$

With circle approximation:

$$SI_{\text{circle}} = \frac{1}{1+\eta+\nu_2}\left\{1-e^{-\frac{2a^2}{\pi\cdot\alpha^2}}+\eta\left(1-e^{-\frac{2a^2}{\pi\cdot\beta^2}}\right)+\nu_2\left(1-e^{-\frac{2a^2}{\pi\cdot\gamma_2^2}}\right)\right\},\ \nu=0 \tag{24}$$

Exponential term

$$SI_{\text{rectangle}} = \frac{\nu}{2\gamma^2}\left[\int_{-\frac{b}{2}}^{\frac{b}{2}}\int_{-\frac{a}{2}}^{\frac{a}{2}} e^{-\sqrt{\frac{x^2}{\gamma^2}+\frac{y^2}{\gamma^2}}}\,dxdy\right] \tag{25}$$

Exchange variables $\varphi(x)=\gamma\cdot x$

$$SI_{\text{rectangle}} = \frac{\nu}{2\gamma^2}\left[\gamma\int_{-\frac{b}{2\gamma}}^{\frac{b}{2\gamma}}\left(\gamma\int_{-\frac{a}{2\gamma}}^{\frac{a}{2\gamma}} e^{-\sqrt{x^2+y^2}}\,dx\right)dy\right] \tag{26}$$

$$SI_{\text{rectangle}} = \frac{\nu}{2}\left[\int_{-\frac{b}{2\gamma}}^{\frac{b}{2\gamma}}\int_{-\frac{a}{2\gamma}}^{\frac{a}{2\gamma}} e^{-\sqrt{x^2+y^2}}\,dxdy\right] \tag{27}$$

Mutual interaction

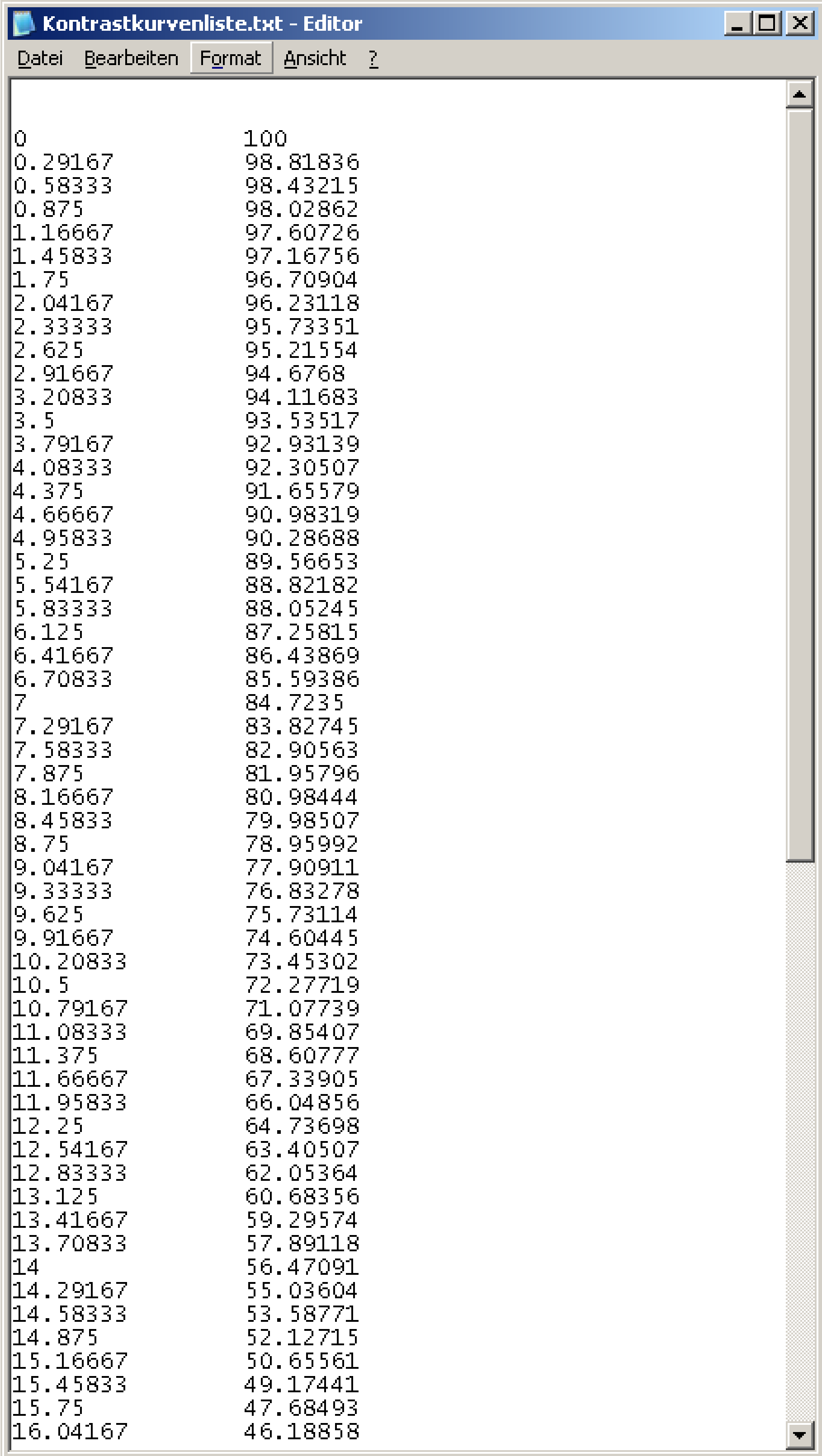
From polygon with area A:

$$\boxed{MI = \frac{A}{\pi\left(1+\eta+\nu+\nu_2\right)}\left(\frac{1}{\alpha^2}e^{-\frac{d^2}{\alpha^2}}+\frac{\eta}{\beta^2}e^{-\frac{d^2}{\beta^2}}+\frac{\nu}{2\gamma^2}e^{-\frac{d}{\gamma}}+\frac{\nu_2}{\gamma_2^2}e^{-\frac{d^2}{\gamma_2^2}}\right)} \tag{28}$$

From line with length L:

$$\boxed{MI_{\text{SPL}} = \frac{L}{\sqrt{\pi}\left(1+\eta+\nu+\nu_2\right)}\left[\frac{1}{\alpha}e^{-\frac{d^2}{\alpha^2}}+\frac{\eta}{\beta}e^{-\frac{d^2}{\beta^2}}+\frac{\nu_2}{\gamma_2}e^{-\frac{d^2}{\gamma_2^2}}+\frac{\nu\cdot\sqrt{\pi}}{2\gamma}e^{-\frac{d}{\gamma}}\right]} \tag{29}$$

6 Example of contrast curve list



Kontrastkurvenliste.txt - Editor

Datei Bearbeiten Format Ansicht ?

0	100
0.29167	98.81836
0.58333	98.43215
0.875	98.02862
1.16667	97.60726
1.45833	97.16756
1.75	96.70904
2.04167	96.23118
2.33333	95.73351
2.625	95.21554
2.91667	94.6768
3.20833	94.11683
3.5	93.53517
3.79167	92.93139
4.08333	92.30507
4.375	91.65579
4.66667	90.98319
4.95833	90.28688
5.25	89.56653
5.54167	88.82182
5.83333	88.05245
6.125	87.25815
6.41667	86.43869
6.70833	85.59386
7	84.7235
7.29167	83.82745
7.58333	82.90563
7.875	81.95796
8.16667	80.98444
8.45833	79.98507
8.75	78.95992
9.04167	77.90911
9.33333	76.83278
9.625	75.73114
9.91667	74.60445
10.20833	73.45302
10.5	72.27719
10.79167	71.07739
11.08333	69.85407
11.375	68.60777
11.66667	67.33905
11.95833	66.04856
12.25	64.73698
12.54167	63.40507
12.83333	62.05364
13.125	60.68356
13.41667	59.29574
13.70833	57.89118
14	56.47091
14.29167	55.03604
14.58333	53.58771
14.875	52.12715
15.16667	50.65561
15.45833	49.17441
15.75	47.68493
16.04167	46.18858

B Tasks

1 EDS – example investigation

The execution of an energy density simulation (EDS) and the analysis of the simulated patterning results will be presented in this section with an example. The analysis shows how you can examine an individual design by specifying a defined resist sensitivity to be able to balance the maximum attainable results with the requirements of the design.

1.1 Prerequisites and targets

For this example investigation, a design with five rectangular boxes of different shapes with defined columns between them was drawn up in the GDSII editor. The five boxes were duplicated, meaning that the design is made up of two identical element groups, each of five boxes. A target dose factor of 1.0 was assigned to the boxes.

After the simulated patterning, the boxes' edges should be as steep and unrounded as possible, and the boxes should have an actual dose factor of 1.0. Furthermore, it is important to calculate the maximum aspect ratio which can be attained for the smallest structures with the available resist.

1.2 Executing the EDS

First of all the proximity dataset “Proximity Test” was created based on an MC simulation, and one of the two element groups of the design with these dataset was corrected.

The dataset has the following underlying proximity function:

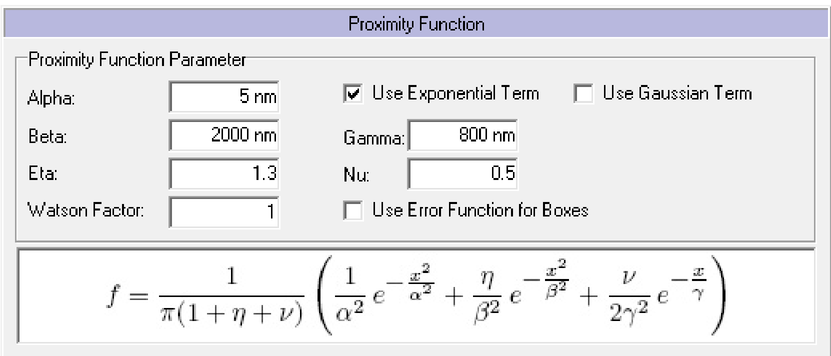


Figure 6-8: Proximity function of the example investigation, displayed in the **Proximity Correction** dialog

(⇒ Figure 6-9 on page 6-23) shows the result of the proximity effect correction in the **EDS - GDSII Design** dialog:

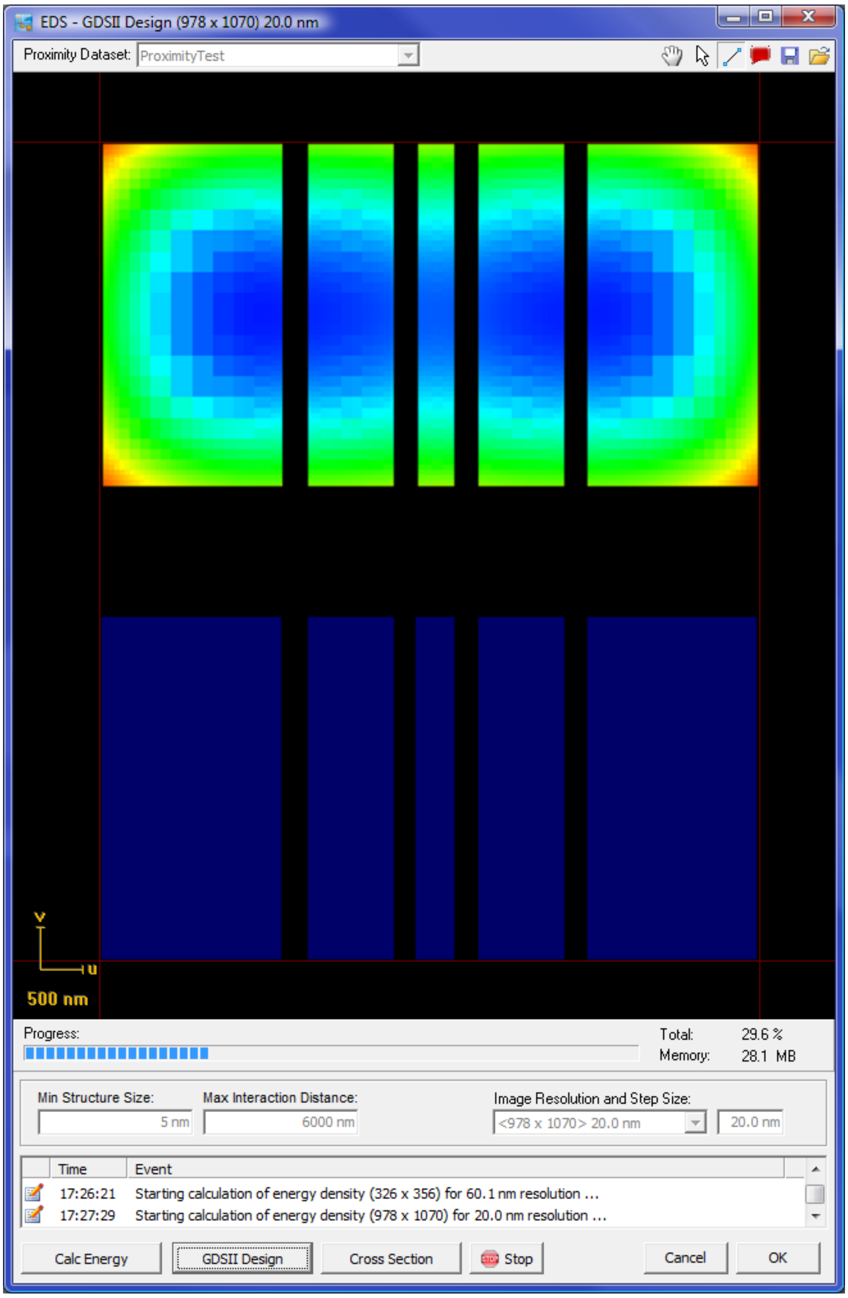


Figure 6-9: Corrected (above) and uncorrected (below) element groups of the example design; displayed in the **EDS - GDSII Design** subdialog

The lower element group was not corrected. The even dark blue coloring indicates an homogenous energy deposition across all elements of the element group.

The upper element group was corrected with the proximity dataset “Proximity Test”. The color tones indicate an inhomogeneous energy deposition across the elements of the element group: In the blue areas, the energy deposition is lowest due to the large proximity effect between individual elements. The energy deposition is highest in the red areas at the edge of the individual elements. The influence of the proximity effect is strongest here.

The partitioning and scattering of the elements of the design which occurred during the proximity effect correction can be seen in the pixelated appearance of the upper element group.

After the proximity effect correction, an EDS was executed for the complete design by clicking on **Calc Energy**.

1.3 Analysis of results

(⇒ Figure 6-10 on page 6-24) shows the result of the energy density simulation in the **EDS - Energy Density Simulation** dialog:

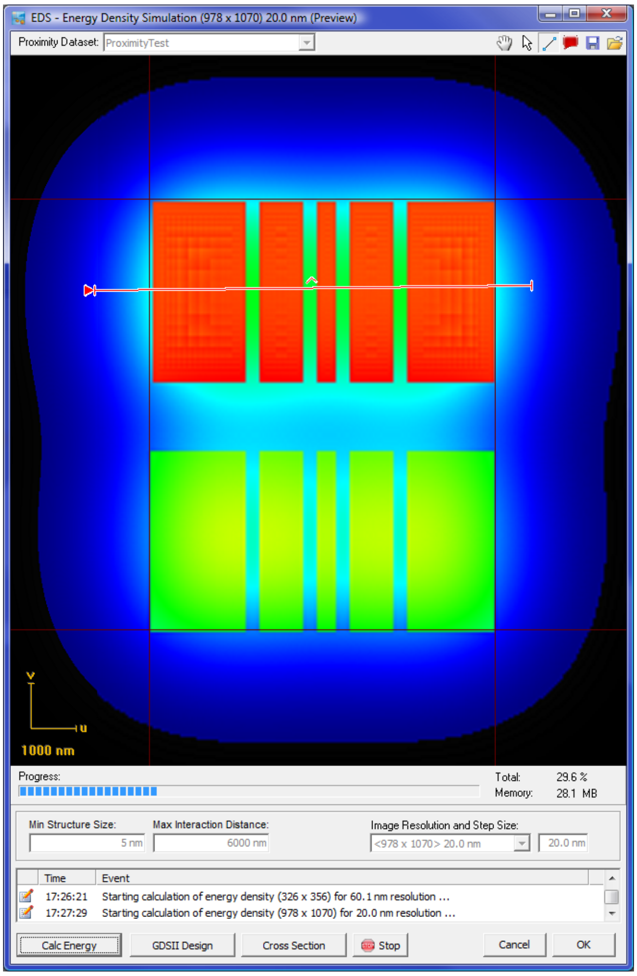
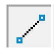


Figure 6-10: Result of the EDS for corrected (above) and uncorrected (below) element groups of the example design; displayed in the **EDS - Energy Density Simulation** dialog

It can clearly be seen that the lower, uncorrected element group appears to have a yellow peak in the middle of the energy distribution after the simulation, while the upper, corrected element group has an even red coloring, indicating homogenous energy distribution.

To make this subjective finding more precise, a cross-section line was drawn into both element groups in succession with the tool . The cross-section line marks an area of the design which can be subjected to a cross-sectional analysis by clicking on **Cross Section**. The cross-section is displayed in the **EDS - Cross Section** dialog (⇒ Figure 6-11 on page 6-26).

(⇒ Figure 6-11 on page 6-26) shows a comparison of the cross-sections of the corrected and uncorrected element groups.

The energy deposition is given as a dose factor on the Y-axis, the position of the elements in the design is given on the X-axis in nm.

The red line indicates the actual energy distribution after the simulated patterning. The yellow line indicates the dose factor of the surface elements for the patterning before the EDS. This is only unequal to 1 for the upper element group elements which have been adjusted by the proximity correction. The blue line indicates the calculated resist profile based on a measured resist sensitivity curve.

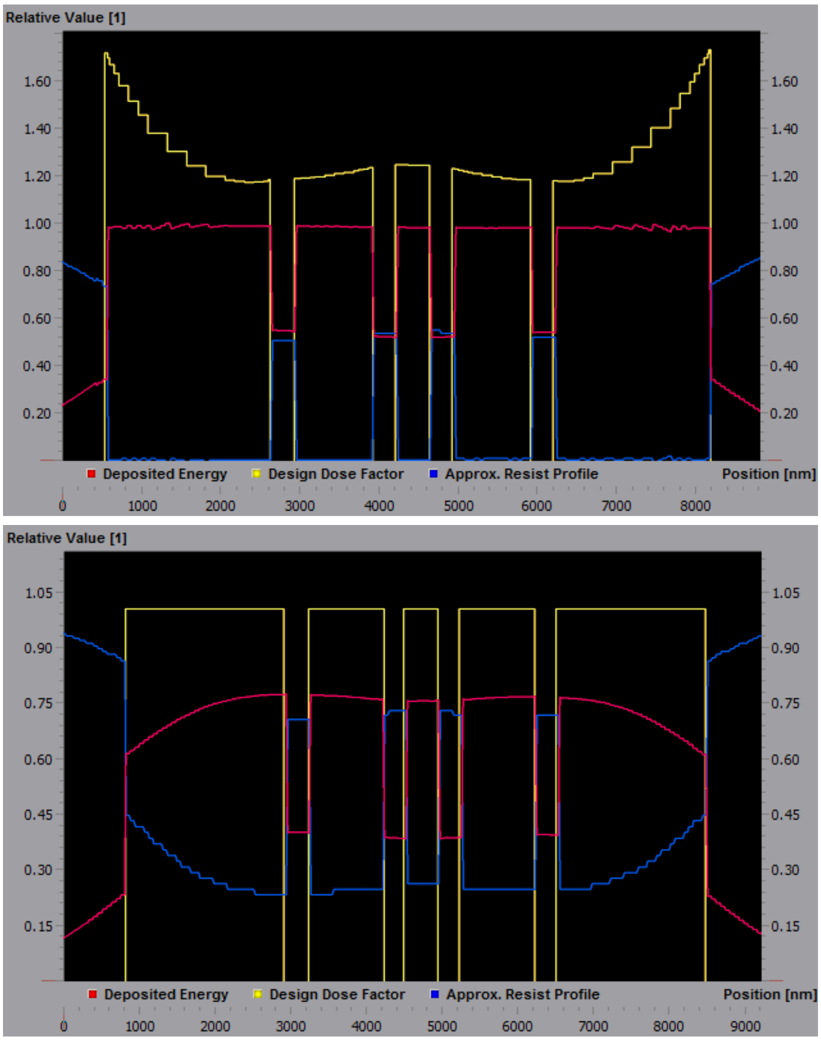


Figure 6-11: Result of the EDS in cross-section: corrected element group above, uncorrected element group below, displayed in the **EDS - Cross Section** subdialog

An homogenous energy distribution corresponding to the target energy distribution of the unexposed drafted design can be seen in the upper corrected element group: The edges of the boxes are steep and rectangular, the dose factor at the upper edges of the boxes is about 1.0. The deviation is less than 5%.

The dose factors of the design were adjusted considerably by the proximity correction in order to attain this almost smooth energy distribution pattern after patterning. The stepped pattern of the red and yellow lines is a result of the incremental adjustment of the energy deposition in accordance with the partitioning and scattering of the elements during the proximity effect correction. The energy dose needed was raised at the edges of the design as a precaution to avoid underexposure.

It can now be seen in the lower, uncorrected element group how bad a pattern may look without the proximity effect correction: In the case of an actual energy distribution with dose factors between approximately 0.75

and 0.6, there is clear underexposure when compared with the targets of the design. The column height of the boxes is too low and the edges are rounded. The energy dose for the structures corresponds only to the values of the draft design. The desired structure could not be attained due to the proximity effect during patterning.

1.4 Simulation of optimum Patterning conditions

The blue curves in the cross-sections depict an approximated resist profile and are the result of folding of the resist sensitivity with the deposited energy.

To simulate optimum patterning settings, you can choose to raise or lower the overall energy dose which is introduced, e.g. by lengthening the patterning time, or to change the resist sensitivity by multiplying by a factor. The latter can only be done easily in the simulation.

NanoPECS™ makes it possible to adjust the resist sensitivity with a slide control in real time. Setting the resist sensitivity enables you to determine the maximum possible column height, or – depending on the structuring process and resist – the maximum possible pit depth in relation to the layer thickness of the resist. To set the resist sensitivity, the right-hand slide control – circled in red in (⇒ Figure 6-12 on page 6-28) – was dynamically adjusted until an optimum setting was achieved.

The following two figures show examples of optimum, as well as overdosed and underdosed, resist sensitivities.

(⇒ Figure 6-12 on page 6-28) shows an optimum setting of the resist sensitivity, in which the column height is approximately 55% of the resist height (**relative resist thickness: 0.550**):

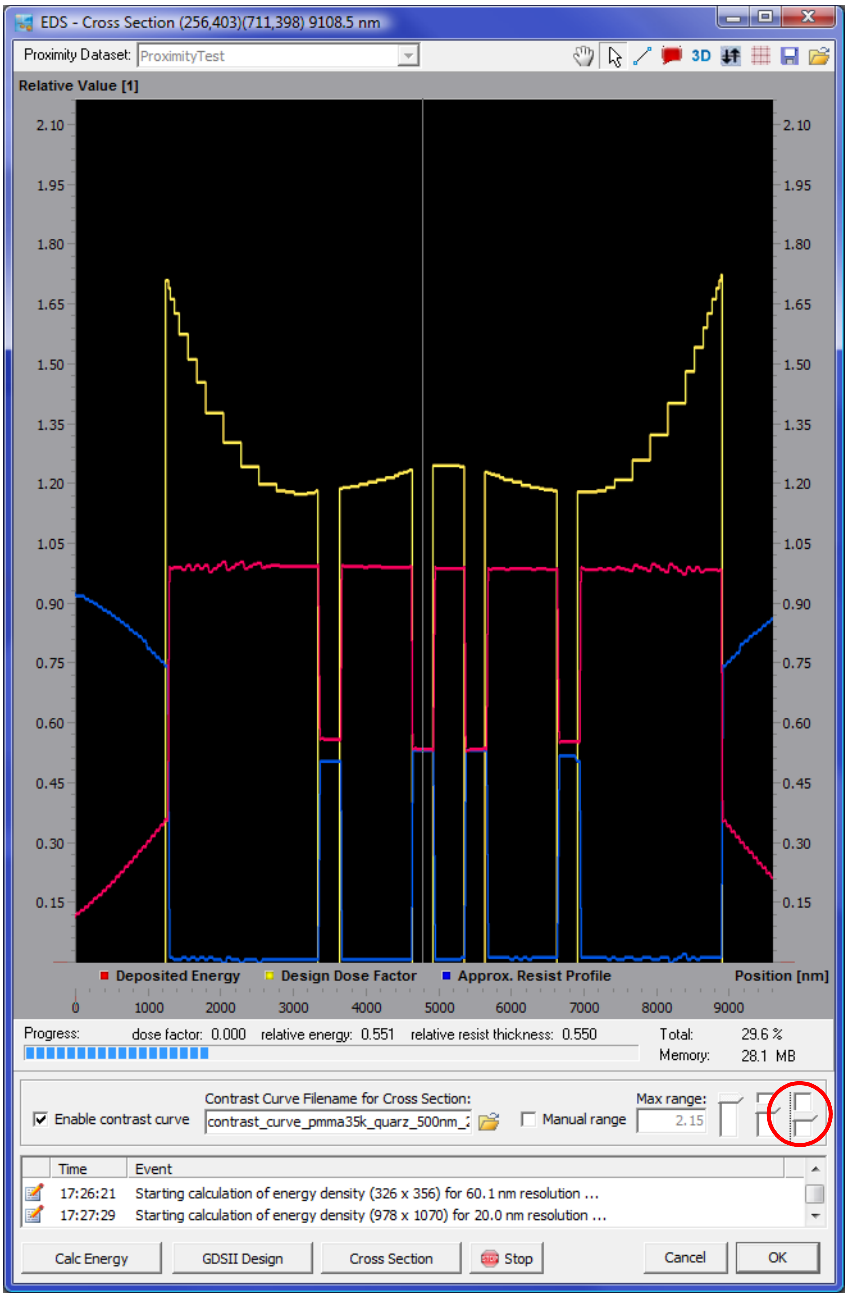


Figure 6-12: Setting the resist sensitivity of the example design: optimum relationship between column height and layer thickness of the resist

(⇒ Figure 6-13 on page 6-29) shows an example of one overdosed or underdosed resist sensitivity setting:

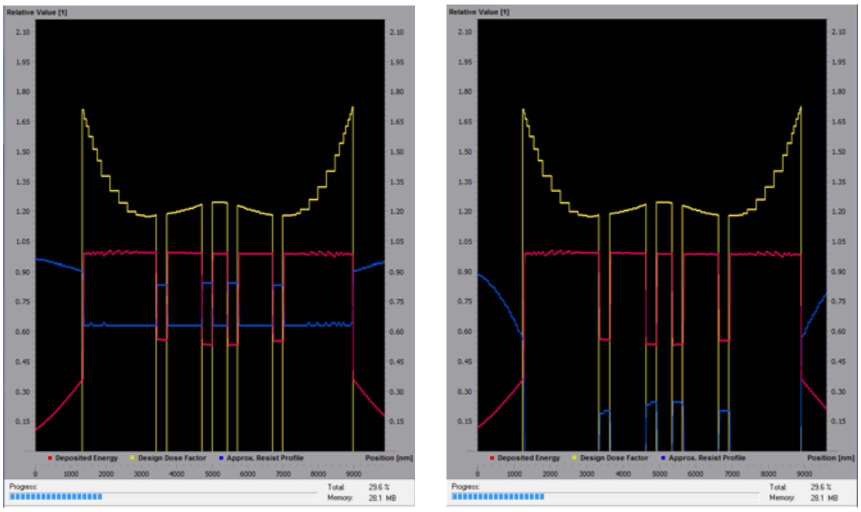


Figure 6-13: Setting resist sensitivity: example of underdosed (left) and overdosed (right) resist sensitivity setting

Based on other energy doses or resist sensitivities, a relative column height of about 55% will not be exceeded if no changes are made to the voltage/material combination, and therefore not to the proximity parameter either.

The result of this example investigation can enable you to balance the maximum attainable result with the requirements specified. If, for example, you need a column height or a pit depth of 80% of the resist height, other parameters may need to be modified, such as:

- Structure density and resolution of the design,
- Type, layer thickness and sensitivity curve of the resist,
- Type of substrate and layer structure,
- Acceleration voltage.

C Software Reference

This chapter describes the structure and function of the NanoPECS™ dialogs, as well as all fields and control elements within them. You will also receive notes and recommendations on entering values for individual parameters.

1 Proximity Parameter Database dialog

You can call up the **Proximity Parameter Database** dialog by selecting **Proximity** ➡ **Proximity Parameter Database** in the menu bar of the NANOSUITE software.

The menu entry **Proximity Parameter Database** is only active when you are within the GDSII module editor of the NANOSUITE software.

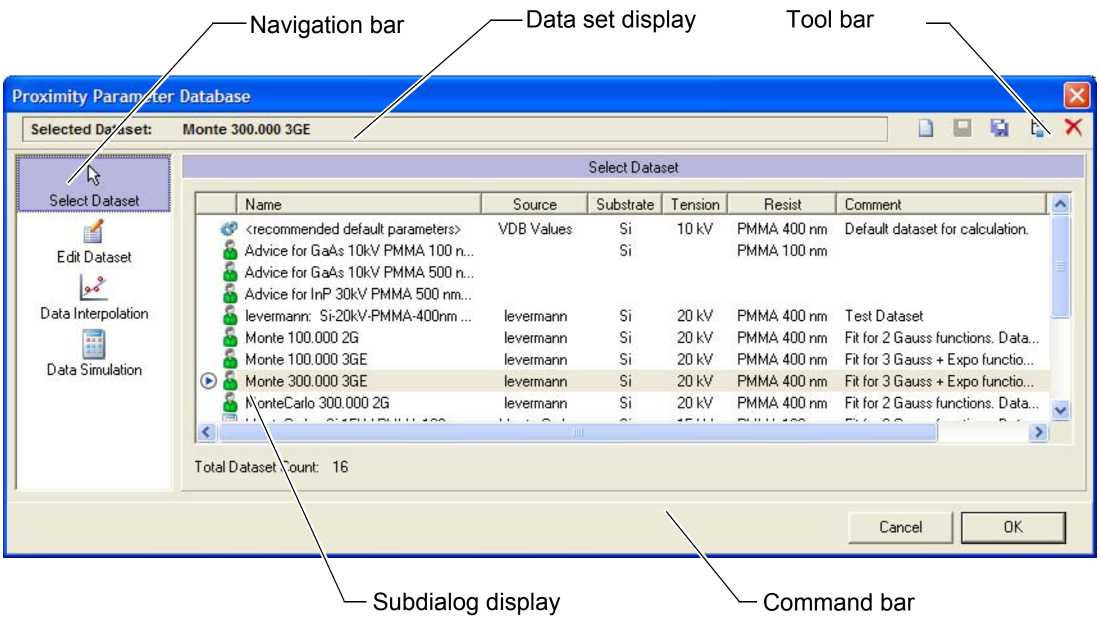


Figure 6-14: **Proximity Parameter Database** dialog

You can use the dialog to

- Create and classify new datasets,
- Select and edit datasets,
- Define proximity parameters for the proximity function,
- Execute MC simulations and simulations by interpolation.

The dialog is divided into five subsets:

Subset	Function
Navigation bar	To navigate between the subdialogs. The **Proximity Parameter Database** dialog is made up of four subdialogs: – **Select Dataset** (⇒ *Select Dataset subdialog* on page 32) – **Edit Dataset** (⇒ *Edit Dataset subdialog* on page 33) – **Data Interpolation** (⇒ *Data Interpolation subdialog* on page 35) – **Data Simulation** (⇒ *Data Simulation subdialog* on page 37)
Dataset display	Displays the currently selected proximity dataset.
Tool bar	To call up and manage the datasets (⇒ *Tool bar* on page 31).
Subdialog display	To display subdialogs.
Command bar	To confirm or reset the entry (⇒ *Command bar* on page 32).

Tool bar

The buttons and displays in the tool bar have the following functions:

Button/display	Function
	Creates a new proximity dataset.
	Saves changes to the currently selected proximity data-set. Note: The button is only active if you have changed set-tings in one of the subdialogs.
	To save the currently selected proximity dataset with a dif-ferent name.
	Calls up the **Edit Proximity Dataset Classification** dialog (⇒ *Edit Proximity Dataset Classification dialog* on page 47).
	Deletes the currently selected proximity dataset.

Command bar

The buttons in the command bar have the following functions:

Button	Function
OK	– Calls up a save query. Confirm the query if you would like to save the current dialog settings in the selected proximity dataset. – Closes the dialog and adopts the selected dataset into the calling form. Note: The selected dataset is only adopted into the form if you have called it up from within the form beforehand.
Cancel	Resets the entries in the individual subdialogs and closes the window.

1.1 Select Dataset subdialog

The datasets saved in the proximity database are listed in the **Select Dataset** subdialog. You can select a dataset from the list which you would like to edit or use as the basis for a simulation. You can sort the column entries in ascending or descending order by clicking on the individual column headers.

The column entries use the proximity dataset classifications which you have defined in the **Edit Proximity Dataset Classification** dialog (⇒ *Edit Proximity Dataset Classification dialog* on page 6-47).

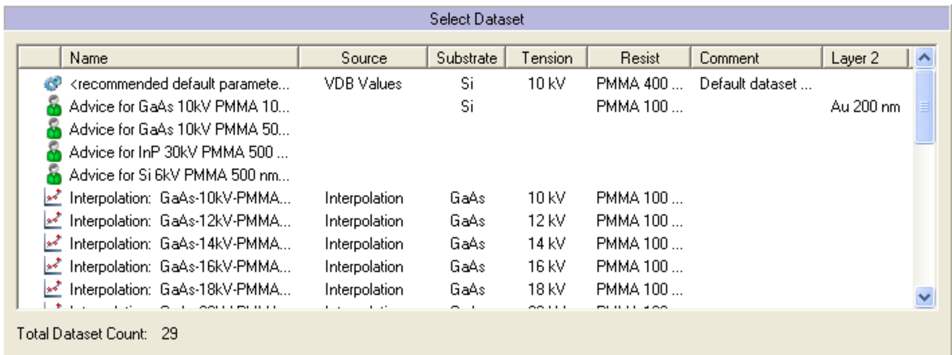


Figure 6-15: **Proximity Parameter Database** dialog – **Select Dataset** subdialog

The column entries have the following meaning:

Column entry	Function
Name	Displays the name of the dataset. A specification is automatically prefixed to the dataset name, depending on how the proximity dataset was generated. You can see the different types of dataset in (⇒ Table 6-2 on page 6-15).
Source	Displays how the dataset was generated.
Substrate	Shows the substrate type.
Tension	Displays the acceleration voltage in kV.
Resist	Specifies the resist type.
Comment	Displays the comment which has been added to the proximity dataset.
Layer 2 - N	Displays more function and resist layers of the dataset.

1.2 Edit Dataset subdialog

You can define the parameters for the proximity function in the **Edit Dataset** subdialog. You can define up to three Gaussian terms and one exponential term. The complete proximity function is displayed in the lower subset of the subdialog. The proximity function forms the basis of a NanoPECS™ proximity correction (⇒ *Determining a Proximity function* on page 6-10).

You can also
- generate the proximity function using an MC simulation (⇒ *Simulation Visualizer* on page 53),
- generate the proximity function using interpolation (⇒ *Data Interpolation subdialog* on page 35),
- or define the proximity function manually using the **Proximity Correction** dialog (⇒ *Proximity Effect Function subdialog* on page 70).

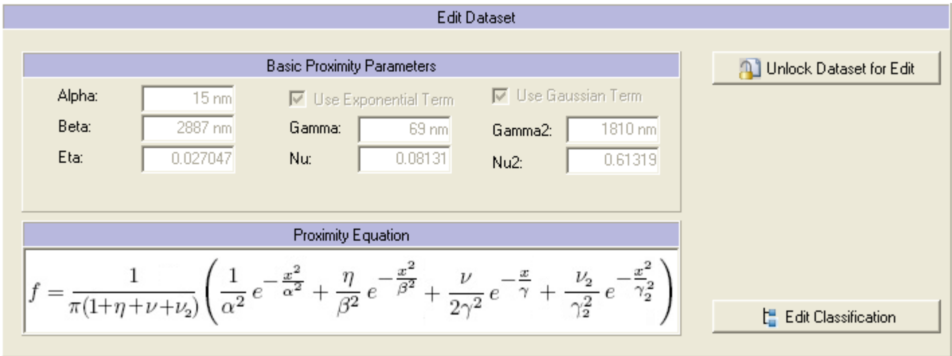


Figure 6-16: **Proximity Parameter Database** dialog – **Edit Dataset** subdialog

You can define the following parameters:

Parameter	Function
Alpha	To enter an α value for the beam broadening due to primary electrons in nm. Recommendation: Default value: 2 nm*
Beta	To enter a β value for the beam broadening due to backscattered electrons in nm. Recommendation: Default value: 1700 nm*
Eta	To enter a η factor for the efficiency of the back-scattered electrons. Recommendation: Default value: 1.3*
Use Exponential Term	To switch the exponential term on and off.
Gamma:	To enter a γ value in the exponential term in nm. Recommendation: Default value: 1000 nm*
Nu:	To enter a ν value in the exponential term. Recommendation: Default value: 0, switched off*
Use Gaussian Term	To hide and reveal the entry fields for the third Gaussian term.

Parameter	Function
Gamma2	To enter a γ_2 value in the third Gaussian term in nm. Recommendation: Default value: 1000 nm*
Nu2	To enter a ν_2 value in the third Gaussian term in nm. Recommendation: Default value: 0, switched off*

* The default values specified here are only test values to check the correction mechanism: Real values depend heavily on the material combination used.

The buttons have the following function:

Parameter	Function
	Calls up the **Edit Proximity Dataset Classification** dialog (⟹ *Edit Proximity Dataset Classification dialog* on page 47).
	Locks the entry fields.
	Unlocks the entry fields.

1.3 Data Interpolation subdialog

You can execute an interpolation of possible proximity parameters for the double gaussian functions in the **Data Interpolation** subdialog. The interpolation procedure is based on interpolation values which are attained from analytical formulae. The Parameters of the formulae are based on experimental results published in literature. You can also make settings for voltage and beam diameter, as well as substrate and resist.

You can create a new proximity dataset using the proximity parameters determined by means of interpolation. The dataset bears the specification **Interpolation:** in its name.

The interpolation procedure is suitable for quick generation of proximity dataset quantities, but only supplies double gaussian function values as a basis for the proximity function.

You can also

- define the proximity function manually using the **Proximity Parameter Database** dialog
  (⇒ *Edit Dataset subdialog* on page 33),
- generate the proximity function using an MC simulation
  (⇒ *Simulation Visualizer* on page 53),
- or define the proximity function manually using the **Proximity Correction** dialog
  (⇒ *Proximity Effect Function subdialog* on page 70).

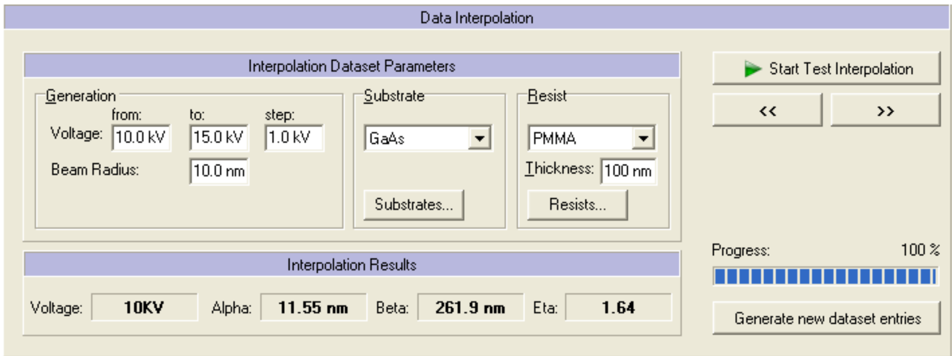


Figure 6-17: **Proximity Parameter Database** dialog – **Data Interpolation** subdialog

The fields and buttons have the following function:

Field/button	Function
Generation	Subset for setting acceleration voltage and electron beam radius.
Voltage	Rows with text fields to define a value range for the acceleration voltage. The acceleration voltage during the simulation is calculated step-by-step within this value range.
from:	To define the lower value of the acceleration voltage in kV.
to:	To define the upper value of the acceleration voltage in kV.
step	To define the steps between the upper and the lower acceleration voltage value in kV at which the simulation should be executed.
Beam Radius	To define the electron beam radius.
Substrate	Drop-down list to select a substrate.

Field/button	Function
Substrates...	Calls up the **Edit Substrates** dialog (⇒ *Edit Substrates dialog* on page 51).
Resist	Drop-down list to select a resist.
Thickness:	To enter the layer thickness of the resist layer in nm.
Resists...	Calls up the **Edit Resists** dialog (⇒ *Edit Resists dialog* on page 52).
Interpolation Results	Subset for displaying the result of the simulation.
Voltage	Shows the acceleration voltage determined during the simulation.
Alpha	Shows the α value determined for the proximity function.
Beta	Shows the β value determined for the proximity function.
Eta	Shows the η factor determined for the proximity function.

Start Test Interpolation

Starts the interpolation for test purposes.

<< >>	To scroll through the simulation results under **Interpolation Results**.
Progress:	Displays the current status of the interpolation.

Generate new dataset entries

Creates one or more proximity datasets in the proximity database.
A dataset is created for each step for which a simulation has been executed. In the example in (⇒ Figure 6-16 on page 6-34), five new datasets are created with the calculated parameters at acceleration voltages of 10.0, 11.0, 12.0, 13.0, 14.0 and 15.0 kV.
The dataset are automatically identified with the specification **Interpolation:** in the dataset name.

1.4 Data Simulation subdialog

You can execute an MC simulation using the **Data Simulation** dialog (⇒ *Executing a Monte Carlo simulation of electron trajectories in the material* on page 6-9). You can use this subdialog to define parameters for

- The acceleration voltage and electron beam shape,
- The number of electron trajectories to be simulated,
- The material and layer structure of the sample,
- The size of the energy deposition grid (EDG) used for the simulation,
- The layer depth in the resist from which the energy distribution is determined as a basis for fitting the proximity parameters.

Based on these parameters, the electron trajectories are calculated during the simulation and the energy deposited by the electrons on their way through the material is saved in an array.

You can fit the determined energy distribution with a spectrum of circular gaussian and exponential functions and adopt the fit parameters as basic proximity parameters into a proximity database (⇒ *Determining a Proximity function* on page 6-10). The dataset name bears the specification **Monte Carlo:**.

Select the MC simulation to obtain reliable parameters for the proximity function.

You can also
- define the proximity function manually using the **Proximity Parameter Database** dialog
  (⇒ *Edit Dataset subdialog* on page 33),
- generate the proximity function using a simulation with interpolation
  (⇒ *Data Interpolation subdialog* on page 35),
- or define the proximity function manually using the **Proximity Correction** dialog
  (⇒ *Proximity Effect Function subdialog* on page 70).

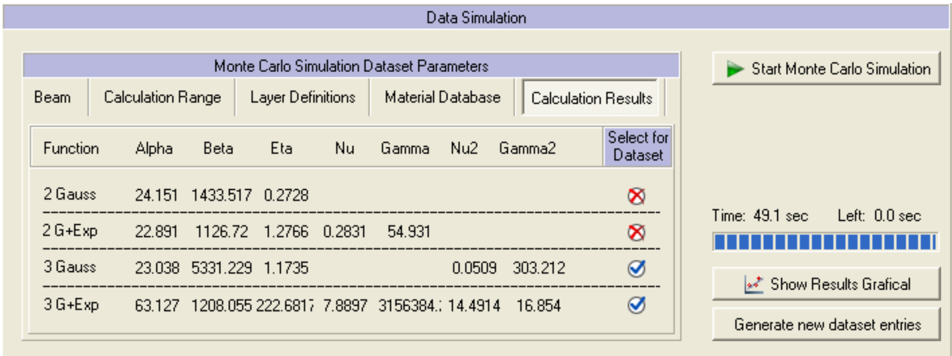


Figure 6-18: **Proximity Parameter Database** dialog – **Data Simulation** subdialog

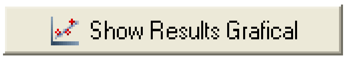
The **Data Simulation** subdialog is divided into five tabs:

Tab	Function
Beam	To define the electron trajectories to be simulated and the parameters for the electron beam (⇒ *Data Simulation subdialog – Beam tab* on page 40).
Calculation Range	To define parameters for the energy deposition grid (EDG) and the fit depth in the resist layer (⇒ *Data Simulation subdialog – Calculation Range tab* on page 42).
Layer Definitions	To define the layer structure of a sample (⇒ *Data Simulation subdialog – Layer Definitions tab* on page 43).
Material Database	To create new material datasets and to edit existing material datasets in the proximity database (⇒ *Data Simulation subdialog – Material Database tab* on page 45).
Calculation Results	To display the results of the MC simulation (⇒ *Data Simulation subdialog – Calculation Results tab* on page 46).

The following buttons and fields are available in all tabs:

Starts an MC simulation with the values displayed in the tabs.

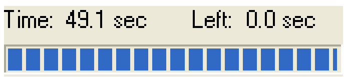
Interrupts an MC simulation which has been started. This button only appears if an MC simulation has been started.



Calls up the Simulation Visualizer. You can use the Simulation Visualizer to display the results of the MC simulation graphically and to balance the target energy distribution of the design with the calculated Gaussian functions (⇒ *Simulation Visualizer* on page 53).

Creates a new proximity database with the calculated Gaussian functions forming a basis for the proximity function.

The proximity datasets are automatically identified with the specification **Monte Carlo:** in the dataset name.



Displays the current status and the time required to execute the MC simulation.

1.4.1 Data Simulation subdialog – Beam tab

In the **Beam** tab, you can define parameters for the energy and shape of the electron beam, as well as the electron trajectories to be taken into account.

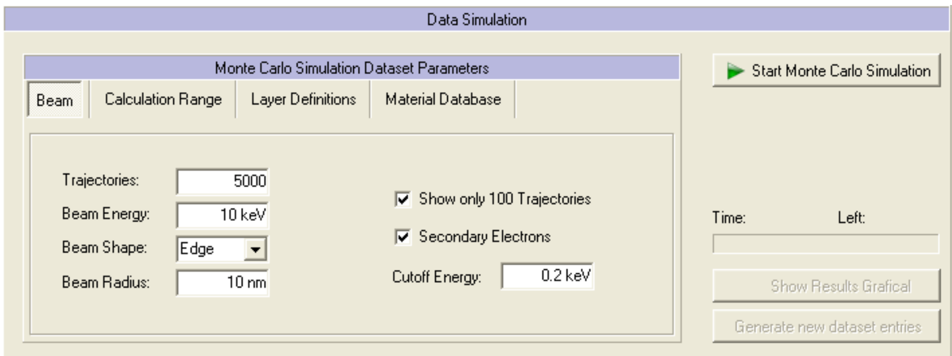


Figure 6-19: **Proximity Parameter Database** subdialog – **Beam** tab

You can define the following parameters in the **Beam** tab:

Field/command	Function
Trajectories	To define the number of electron trajectories to be simulated during the MC simulation in the sample. Recommendation: Enter a value >100,000 to obtain meaningful values for the proximity function. Smaller values – such as 5000 – result in a quicker execution of the MC simulation but are only suitable for testing whether the MC simulation is being executed without errors using the underlying proximity dataset.
Beam Energy	To define the acceleration voltage in keV.
Beam Shape	Drop-down list to select an electron beam profile: – **Edge**: Cylindrical electron beam profile with evenly distributed trajectory density. – **Gaussian**: Gaussian electron beam profile with a variation in the density of the simulation electrons from the center to the edge of the electron beam. – **Point**: A point-shaped electron beam profile 1 nm in diameter.
Beam Radius	To define the radius of the electron beam in nm. This field is only active if you have selected the entry **Edge or Gaussian** in the **Beam Shape** drop-down list.
Show only 100 Trajectories	To limit the number of electron trajectories displayed in the Simulation Visualizer (⟹ *Visualization of electron trajectories* on page 60). If this checkbox is activated, only the first 100 electron trajectories calculated during the simulation are displayed. Recommendation: Activate the checkbox in order to be able to differentiate between individual electron trajectories in the Simulation Visualizer. If too many electron trajectories are displayed there, they simply become one large area and can no longer be individually identified.
Secondary Electrons	To limit the type of electrons to be simulated. If this checkbox is activated, the trajectories of the secondary electrons are also simulated and displayed in the Simulation Visualizer.
Cutoff Energy:	To enter the maximum energy in keV up to which the electron trajectories in a sample are to be tracked.

1.4.2 Data Simulation subdialog – Calculation Range tab

In the **Calculation Range** tab, you can define settings for the energy deposition grid (EDG) and the fit depth in the resist layer.

The EDG is made up of cubes with an internal volume of 1 nm^3. It defines the cylindrical section of a sample in which the energy is saved during the simulation. The energy comes from the modeled electron beam in the center of the cylinder.

The fit depth specifies the depth from which the energies saved in the EDG in the simulation are taken to calculate the proximity parameters.

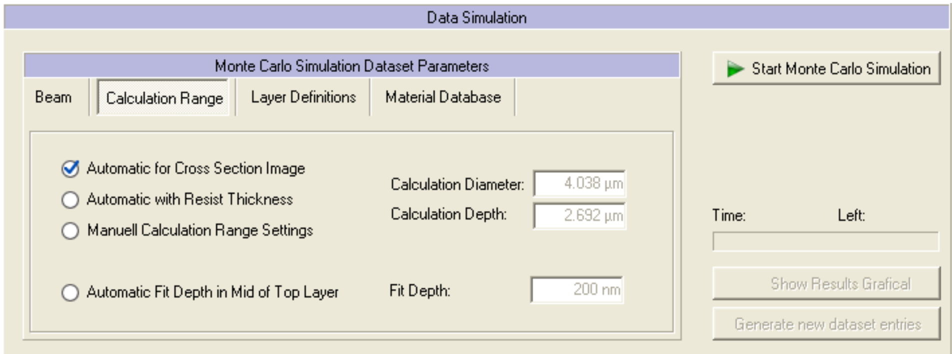


Figure 6-20: **Data Simulation** subdialog – **Calculation Range** tab

The checkboxes and fields have the following function:

Checkbox/field	Function
Automatic for Cross Section Image	For automated determination of the size of the EDG for an optimized view of all electron trajectories. The checkbox is deactivated when the **Manual Calculation Range Settings** checkbox is activated. Recommendation: If acceleration voltages are high, this setting may mean that the EDG requires too much memory. If so, it is better to use a smaller EDG – e.g. one which is reduced to the resist. This does not result in any difference in the calculation of the proximity parameters (⇒ Automatic with Resist Thickness on page 42).
Automatic with Resist Thickness	For automated determination of the size of the EDG for optimized tracking of all electron trajectories in the resist. The checkbox is deactivated when the **Manual Calculation Range Settings** checkbox is activated. Recommendation: This setting requires less main memory than the **Automatic for Cross Section Image** setting. There is no restriction to the calculation of proximity parameters.

Checkbox/field	Function
Manual Calculation Range Setting	To define the size of the EDG manually. The **Calculation Diameter** text fields and **Calculation Depth** are activated and deactivated. You can activate this checkbox parallel to the **Automatic for Cross Section Image and Automatic with Resist Thickness** checkboxes in order to define individual values for diameter and depth of the EDG for these parameters. Recommendation: Only use the manual settings if the memory requirement for both automatic settings is too large. Too small an EDG distorts the proximity parameter calculation as the electron trajectories are not included fully in the calculation.
Calculation Diameter	To enter the diameter of the EDG individually in nm.
Calculation Depth	To enter the depth of the EDG individually in nm.
Automatic Fit Depth in Mid of Top Layer	To activate or deactivate automatic setting of the fit depth. If the checkbox is activated, the fit is always executed in the middle of the upper resist layer.
Fit Depth:	To set the fit depth individually. Depending on the sample, the layer structure can be made up of several resist and function layers (⟹ Table 6-3 on page 6-43). While the **Automatic Fit Depth in Mid of Top Layer** only refers to the upper resist layer, you can define fit depths in lower resist layers here.

1.4.3 Data Simulation subdialog – Layer Definitions tab

The sample layers which are saved in the selected proximity dataset are listed in the **Layer Definitions** tab. You can redefine the sample's layer structure here.

You can compose a sample from three types of layers:

Table 6-3: Sample layer types

Layer	Description
Function layer	Layer made of material which is not developed, such as gold. A function layer protects the resist, among other things.
Resist layer	Layer which is not made up of a resist, such as PMMA. Note: You can arrange several resist layers, one above the other.

Table 6-3: Sample layer types

Layer	Description
Substrate	Carrier layer for the other layers. Note: A substrate is the only basic material you can assign to your hypothetical sample.

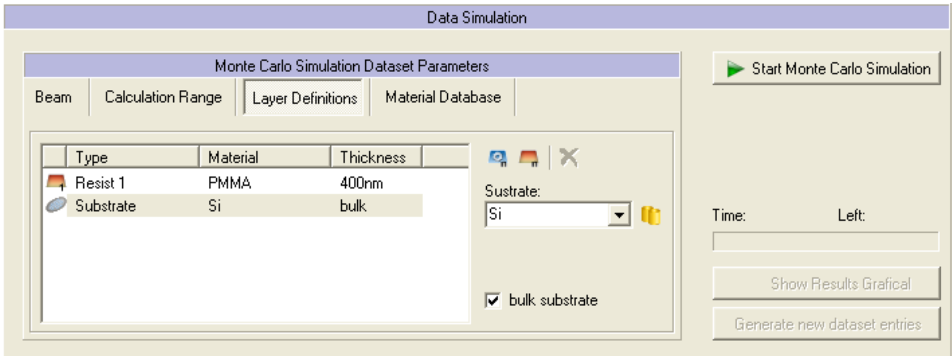


Figure 6-21: **Data Simulation** subdialog – **Layer Definitions** tab

The fields and buttons have the following function:

Field/button	Function
Type	Specifies the layer type.
Material	Specifies the layer material.
Thickness	Specifies the layer thickness in nm.
	Adds a function layer to the list.
	Adds a resist layer to the list.
	Deletes a selected entry from the list.
Substrate	Drop-down list to select a substrate.
	Switches to the **Material Database** tab. Here you can look at and edit individual parameters of the substrates displayed in the **Substrates** drop-down list and create a new substrate or new material (⟹ *Data Simulation subdialog – Material Database tab* on page 45).

Field/button	Function
bulk substrate	To toggle the value **bulk** in the **Thickness** column. The value **bulk** defines an infinite layer thickness for the substrate. If the checkbox is deactivated, the **Thickness** column for the substrate is active and you can enter an individual value for the layer thickness of the substrate.

1.4.4 Data Simulation subdialog – Material Database tab

The material names and properties saved in the proximity database are listed in the **Material Database** tab. You can define new materials or modify the parameters of the materials displayed here.

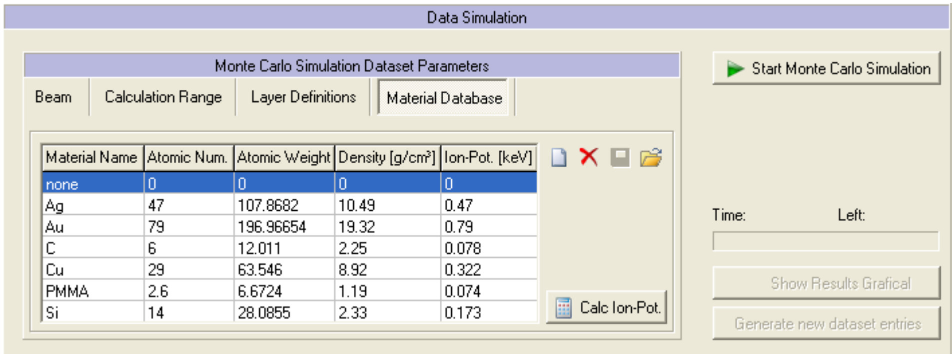


Figure 6-22: **Data Simulation** subdialog – **Material Database** tab

The columns and buttons have the following function:

Column/button	Function
Material Name	Specifies the name of the material.
Atomic Num.	Specifies the average atomic number of the material.
Atomic Weight	Specifies the average atomic weight in g/mol.
Density [g/cm3]	Specifies the density of the material in g/cm^3.
Ion-Pot. [keV]	Specifies the material's ionization potential in keV.
	Creates a new row in the table for a material. If you have created a new row, the table cells in the individual columns are active and you can enter a material name and values for the individual parameters.
	Deletes a highlighted row from the table.

Column/button	Function
	Saves the currently selected dataset after a change. The button is grayed-out if you have not made changes to the table.
	– Loads a dataset of materials. – Allows an undo when you wish to overwrite changes made to the table.
	Recalculates the **Ion-Pot. [keV]** column after you have correctly filled other columns with the material properties.

1.4.5 Data Simulation subdialog – Calculation Results tab

The **Calculation Results** tab is only shown in the **Data Simulation** subdialog when an MC simulation is finished.

The results of the MC simulation are listed in the **Calculation Results** tab. You can also graphically display and analyze the results in the Simulation Visualizer (⇒ *Simulation Visualizer* on page 6-53).

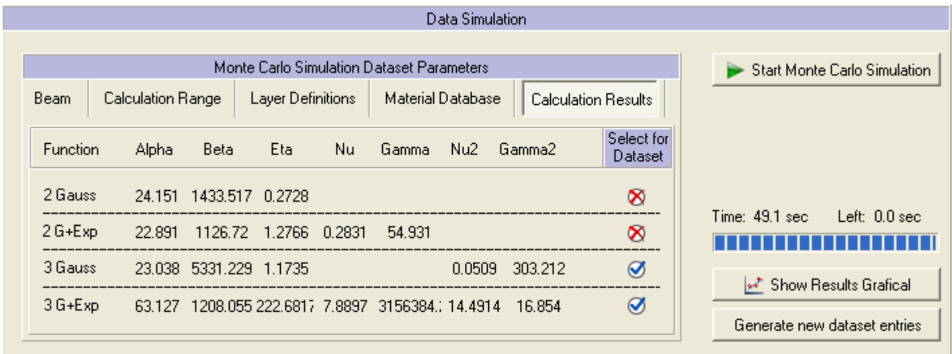


Figure 6-23: **Data Simulation** subdialog – **Calculation Results** tab

The following results are displayed:

Column	Value
Function	Specifies the designation of the calculated Gaussian functions.
Alpha	Specifies the calculated α value for the beam broadening due to primary electrons in nm.
Beta	Specifies the calculated β value for the beam broadening due to back-scattered electrons in nm.

Column	Value
Eta	Specifies the calculated η factor for the efficiency of the back-scattered electrons.
Nu	Specifies the calculated ν value of the exponential term.
Gamma	Specifies the calculated γ value of the exponential term in nm.
Nu2	Specifies the calculated ν_2 value of the third Gaussian term.
Gamma2	Specifies the calculated γ_2 value of the third Gaussian term in nm.
Select for Dataset	To select which Gaussian functions are to be adopted into the proximity dataset as the basis for the proximity function. You can switch between ☑ = adopt and ☒ = do not adopt by clicking on the checkbox. Note: Adopt the Gaussian functions which most closely reproduce the target energy distribution of the design. You can check which ones these are in the Simulation Visualizer (⇒ *Visualization of fitting functions* on page 58).

1.5 Edit Proximity Dataset Classification dialog

You can call up the **Edit Proximity Dataset Classification** dialog by selecting the dialog **Proximity Parameter Database** ➡ Subdialog **Edit Dataset** ➡ (⇒ *Select Dataset subdialog* on page 6-32).

You use the **Edit Proximity Dataset Classification** dialog to define basic parameters such as substrate, acceleration voltage and layer structure for the proximity dataset and to assign it the appropriate classifications.

The settings in the **Edit Proximity Database Classification** dialog are displayed in the **Proximity Parameter Database** dialog – **Select Dataset** subdialog.

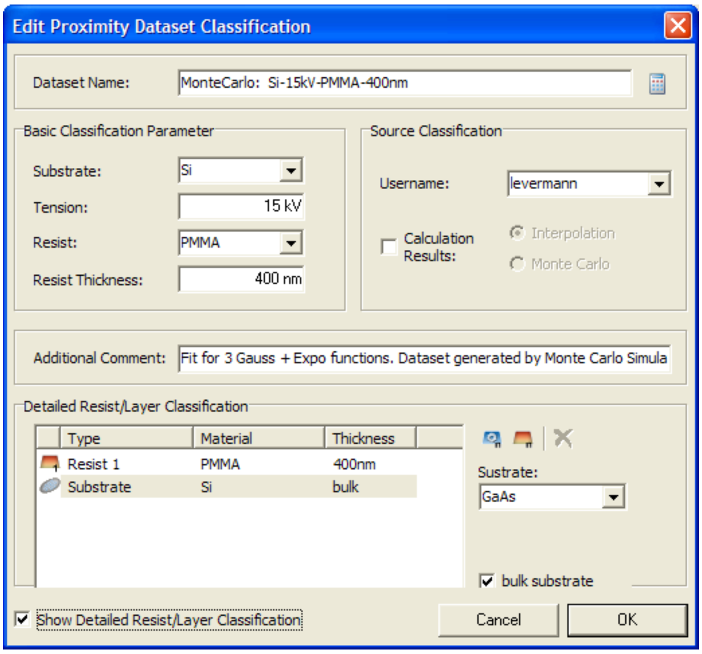


Figure 6-24: **Edit Proximity Dataset Classification** dialog

The dialog is divided into the following subsets:

Subset	Function
Dataset Name	To display and generate the proximity dataset name (⇒ *Dataset Name subset* on page 48).
Basic Classification Parameter	To define the basic parameters of the dataset classification (⇒ *Basic Classification Parameter subset* on page 49).
Source Classification	To define which specification should be prefixed to the dataset name (⇒ *Source Classification subset* on page 49).
Additional Comment	To enter a commentary for the selected proximity dataset.
Detailed Resist/ Layer Classification	Table to select and define the layer composition of a hypothetical sample (⇒ *Detailed Resist/Layer Classification subset* on page 50).
Commands	To confirm or reset the entries (⇒ *Command bar* on page 32).

1.5.1 Dataset Name subset

The fields and buttons have the following function:

Field/button	Function
Dataset Name:	Displays the selected proximity dataset name.
	Automatically generates a standardized proximity dataset name from the values and classifications defined in the dialog.

1.5.2 Basic Classification Parameter subset

You can define the following parameters:

Field/command	Function
Substrate	Drop-down list to select a substrate.
Tension	To define the acceleration voltage in kV.
Resist	Drop-down list to select a resist.
Resist Thickness	To enter the density of the resist layer.

1.5.3 Source Classification subset

You can make the following settings:

Field/checkbox	Function
Username	Drop-down list to select a user name which forms the specification at the beginning of the dataset name. The user name selected here replaces the standard specification which was generated as part of a simulation method or through pre-defined proximity datasets – e.g. **Monte Carlo:**, **Interpolation:** or **Advice for** (⇒ *Classification and nomenclature of Proximity datasets* on page 15). Recommendation: Define a user name as a specification if you would like to make one or more proximity datasets available to a work group.
Calculation Results	To activate and deactivate the option **Interpolation** and **Monte Carlo** fields. Note: Deactivate the checkbox if you would like to select a user name as a specification for the dataset name under **Username**.
Interpolation	Defines a dataset as the result of an interpolation for the purposes of dataset classification.

Field/checkbox	Function
Monte Carlo	Defines a dataset as the result of an MC simulation for the purposes of dataset classification.

1.5.4 Detailed Resist/Layer Classification subset

The columns, fields and buttons have the following function:

Column/field/button	Function
Type	Specifies the layer type. If there are several layers, the layer designation is numbered in ascending order – e.g. **Resist 1**, **Resist 2**.
Material	Specifies the layer material or resist material.
Thickness	Specifies the layer height or resist height.
	Adds a function layer to the list. After adding, you define the material and the density of the function layer (⇒ *Edit Resists dialog* on page 52).
	Adds a resist layer to the list. After adding, you define the material and the density of the resist layer (⇒ *Edit Resists dialog* on page 52).
	Deletes a selected entry from the list.
Substrate	Drop-down list to select a substrate. The selected substrate is displayed in the list.
bulk substrate	To toggle the value **bulk** in the **Thickness** column. The value **bulk** defines an infinite layer thickness for the substrate. If the checkbox is deactivated, the Thickness column for the substrate is active and you can enter an individual value for the height of the substrate.
Show Detailed Resist/Layer Classification	To show and hide the **Detailed Resist/Layer Classification** subset.

1.5.5 Commands subset

The buttons have the following function:

Field/command	Function
Cancel	Resets all entries and closes the dialog.
OK	Adopts all entries and closes the dialog.

1.6 Edit Substrates dialog

You can call up the **Edit Substrates** dialog by selecting the **Proximity Parameter Database** dialog ➡ **Data Interpolation** subdialog ➡ **Substrates...** (⇒ Figure 6-17 on page 6-36).

You can create new substrates with specific properties in the **Edit Substrates** dialog.

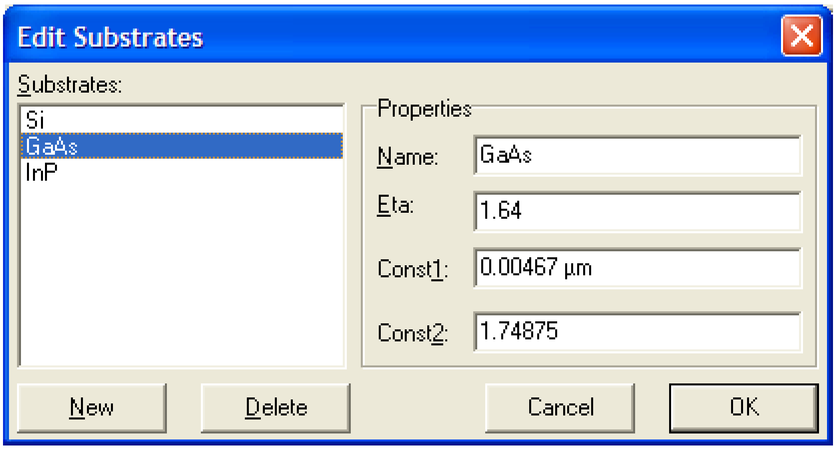


Figure 6-25: **Edit Substrates** dialog

The following fields and commands are available:

Field/command	Function
Substrates	Input field to display the available substrates.
Properties	Subset to define the properties of a new or selected substrate.
Name	To enter a name for the substrate.
Eta	To enter an η value for the proximity function.
Const1: **Const2:**	To determine the constants C_1 and C_2 in the following function: $\beta = C_1 E^{C_2}$ This function is used for determing an accurate β value for the proximity function by fitting it to the measurement.

Field/command	Function
New	Creates a new substrate in the **Substrates** input field.
Delete	Deletes a highlighted substrate from the **Substrates** input field.
Cancel	Resets all entries and closes the dialog.
OK	Adopts and saves the entries and closes the dialog.

1.7 Edit Resists dialog

You can call up the **Edit Resists** dialog by selecting the **Proximity Parameter Database** ➡ **Data Interpolation** subdialog ➡ **Resists...** (⇒ Figure 6-17 on page 6-36).

You can create new resists with specific properties in the **Edit Resists** dialog.

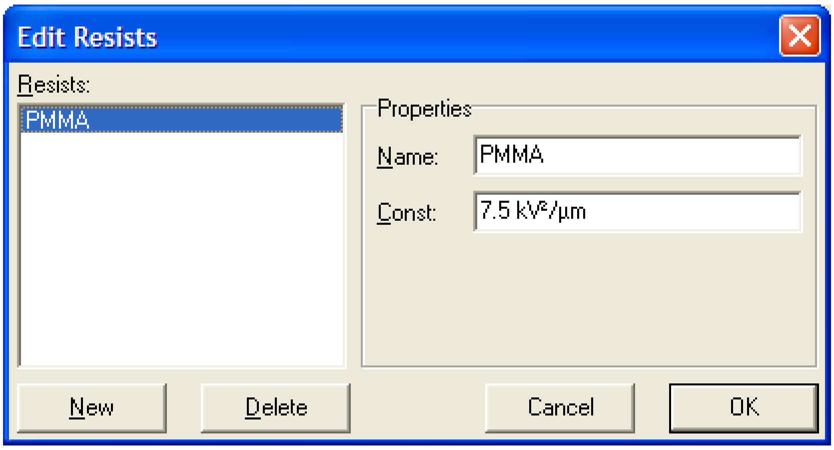


Figure 6-26: **Edit Resists** dialog

The following fields and commands are available:

Field/command	Function
Substrates	Input field to display the available resists.
Properties	Subset to define the properties of a new or selected resist.
Name	To enter a name for the resist.

Field/command	Function
Const:	To determine the constant C in the following function: $$\alpha = \sqrt{\alpha_0^{\ 2} + C\frac{H_0^{\ 3}}{E^2}}$$ This function is used for determing an accurate α value for the proximity function by fitting it to the measurement.
New	Creates a new resist in the **Resists** input field.
Delete	Deletes a highlighted resist from the **Resists** input field.
Cancel	Resets all entries and closes the dialog.
OK	Adopts and saves the entries and closes the dialog.

1.8 Simulation Visualizer

You can call up the Simulation Visualizer by selecting the **Proximity Parameter Database** dialog ➡ **Data Simulation Database** subdialog ➡ **Calculation Results** tab ➡ **Show results graphically** (⇒ *Data Simulation subdialog – Calculation Results tab* on page 6-46).

The Simulation Visualizer offers a graphic interface which you can use to analyze the results of a MC simulation. You can

- View the parameters on which the MC simulation is based,
- Show the simulated energy deposition grid (EDG),
- Show the simulated electron trajectories in the sample,
- Show the Gaussian functions calculated during the simulation and balance them with the simulated energy distribution in the resist.

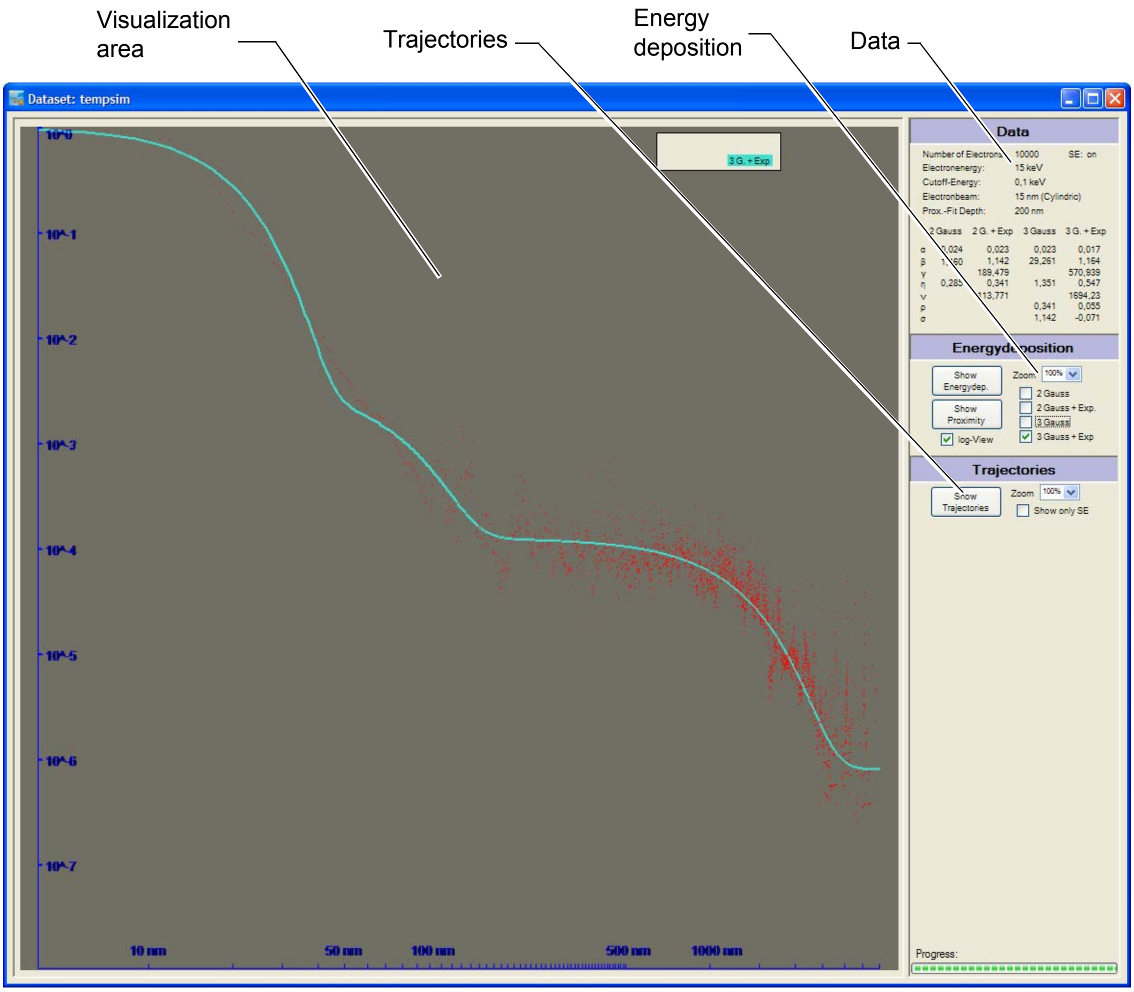


Figure 6-27: Interface of the Simulation Visualizer

The Simulation Visualizer is divided into four subsets:

Subset	Function
Visualization area	For graphic display and analysis of – EDG (⟹ *Visualization of Energy Deposition Grid (EDG)* on page 56), – electron trajectories (⟹ *Visualization of electron trajectories* on page 60), – Gaussian functions (⟹ *Visualization of fitting functions* on page 58).
Trajectories	To display the simulated electron trajectories (⟹ *Trajectories subset* on page 56).
Energy deposition	To switch between displaying the simulated EDG and the calculated Gaussian functions (⟹ *Energy deposition subset* on page 55).

Subset	Function
Data	View the parameters on which the MC simulation is based (⇒ *Data subset* on page 55).

1.8.1 Data subset

The parameters displayed have the following meaning:

Parameter	Function
Number of Electrons	Specifies the number of simulated electron trajectories.
SE:	Shows whether the trajectories of the secondary electrons were taken into account during the simulation or not.
Electron energy	Displays the simulated acceleration voltage in keV.
Cutoff-Energy	Specifies the residual energy amount of the electrons in keV up to which the trajectory in a sample was tracked.
Electron beam	Specifies the radius of the electron beam in nm and, in brackets, the profile of the electron beam.
Prox.-Fit Depth	Specifies the fit depth in nm (⇒ *Data Simulation subdialog – Calculation Range tab* on page 42).
2Gauss	Specifies the proximity parameters for the double Gaussian function.
2G+Exp	Specifies the proximity parameters for the double Gaussian function with exponential term.
3Gauss	Specifies the proximity parameters for the triple Gaussian function.
3G+Exp	Specifies the proximity parameters for the triple Gaussian function with exponential term.

1.8.2 Energy deposition subset

The control elements have the following function:

Control element	Function
Show Energydep.	Shows the simulated EDG in the visualization window (⇒ *Visualization of Energy Deposition Grid (EDG)* on page 56).
Zoom	To enter or select a zoom factor for the display in the visualization area.
Show Proximity	Shows the energy distribution of the resist in the fit depth as a scatter diagram in the visualization (⇒ *Visualization of fitting functions* on page 58).
log-View	Switches between logarithmic and normal display of the energy distribution.
2Gauss	Checkboxes to show and hide individual Gaussian functions in the visualization area. If you have used the **Show Proximity** button to show the energy distribution in the visualization area, you can use these checkboxes to show the individual Gaussian functions selectively and to balance them with the simulated energy distribution (⇒ *Visualization of fitting functions* on page 58).
2Gauss+Exp	
3Gauss	
3Gauss+Exp	

1.8.3 Trajectories subset

The control elements have the following function:

Control element	Function
Show Trajectories	Shows the simulated electron trajectories in the visualization area (⇒ *Visualization of electron trajectories* on page 60).
Zoom	To enter or select a zoom factor for the display in the visualization window.
Show only SE	To show and hide the trajectories of the secondary electrons. If this checkbox is activated, the trajectories of the secondary electrons are shown.

1.8.4 Visualization of Energy Deposition Grid (EDG)

You can call up the visualization of the EDG by clicking on **Show Energydep.** in the **Energydeposition** subset within the Simulation Visualizer (⇒ *Simulation Visualizer* on page 6-53).

In the visualization of the EDG, you can reproduce the behavior of the electron beam as it penetrates a sample, as well as the resultant energy distribution.

The following information is shown in the visualization area:

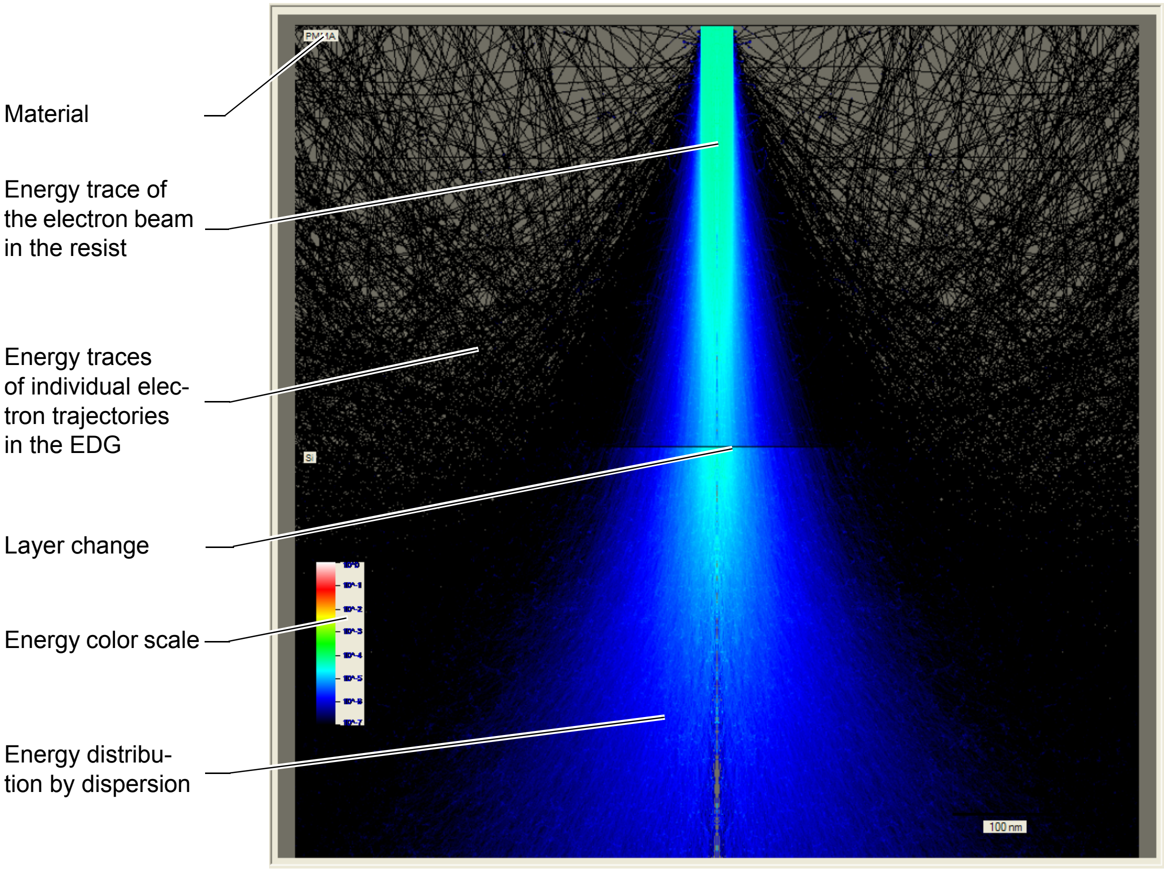


Figure 6-28: Simulation Visualizer – visualization of the EDG

Information	Function
Material	Specifies the designation of the material in the individual sample layers. In the example (⟹ Figure 6-28 on page 6-57), the sample is composed of the resist **PMMA** and the substrate silicon (**Si**).
Energy trace of the electron beam in the resist	Displays the path of the electron beam when it penetrates the sample. The high energy values deposited result from the electron trajectories in the simulation shortly after they penetrate the probe. Initially they are high in energy and only have a low level of interference.

Information	Function
Electron trajectories	The single black lines are in those areas of the sample which are hit by low-energy backscattered electrons only. No electrons fly into the gray background.
Layer change	Identifies the boundary between the sequential layers of the sample. On the boundary between the resist and the substrate, in particular, you can reproduce the rapidly changing material properties.
Energy color scale	Key for the energy/color assignment.
Energy distribution by scattering	Displays the energy distribution arising from the backscatter of the electron beam in the sample. The color shades identify areas with varying energy deposition. You can read the corresponding keV value from the energy color scale. The characteristic scatter bulb can be seen quite clearly.

1.8.5 Visualization of fitting functions

You can call up the visualization of the fitting functions by clicking on **Show Proximity** in the **Energydeposition** subset within the Simulation Visualizer (⇒ *Simulation Visualizer* on page 6-53).

The following information is shown in the visualization area:

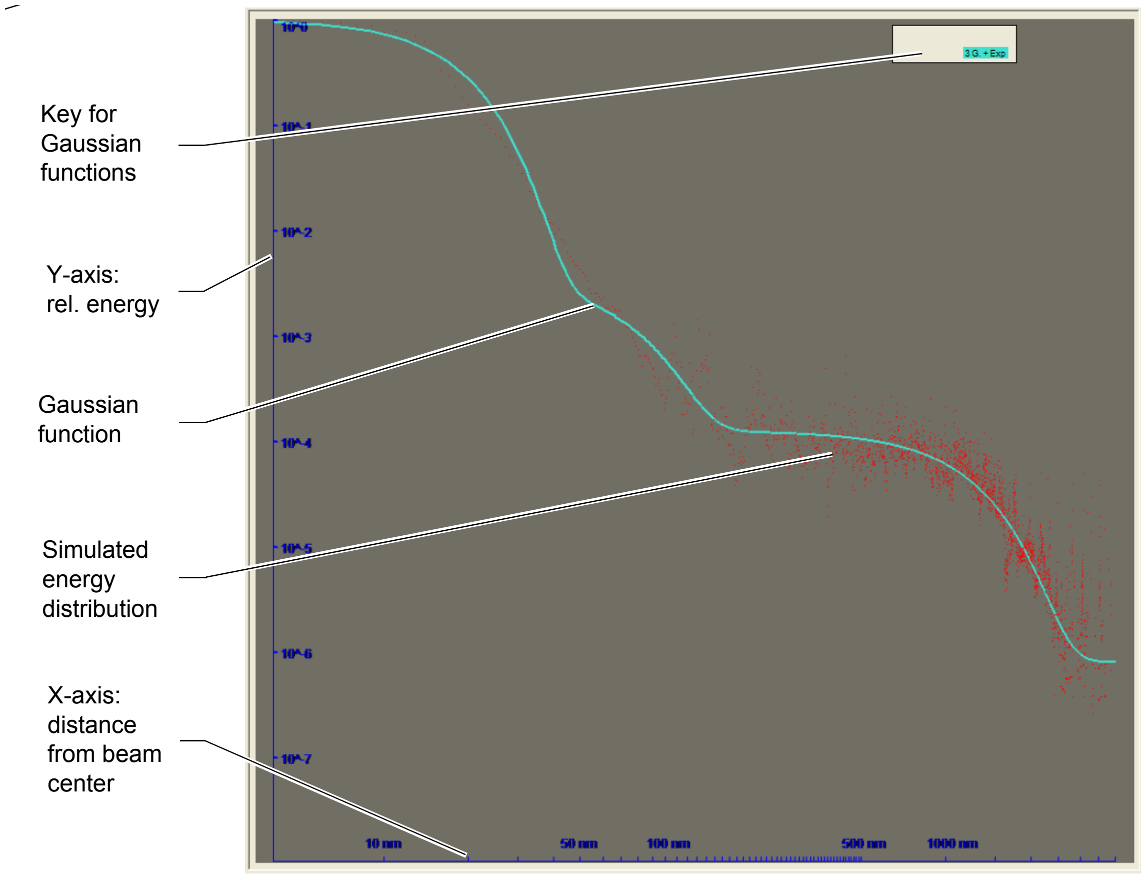


Figure 6-29: Simulation Visualizer – visualization of fitting functions

Information	Function
Key for Gaussian functions	Identifies the color of different shown Gaussian functions.
Y-axis: rel. energy	Specifies the relative average energy deposition in grid positions at distance X from the center of the electron beam. The total of the energy depositions is standardized at 1.
Gaussian function	Displays the selected Gaussian function as a curve.
Simulated energy distribution	Displays the simulated energy deposition into the EDG in the fit depth as a scatter diagram in projection on the original state.
X-axis: distance from beam center	Specifies the distance from the center of the EDG in nm.

You can balance the calculated Gaussian functions with the simulated energy distribution in EDG in the fit depth using the visualization of the Gaussian functions. To do this, show the Gaussian functions in sequence in the visualization window by activating the appropriate checkboxes in the **Energydeposition** subset and check which of the Gaussian functions best fits the scatter diagram of energy distribution.

You can adopt the proximity parameters from these Gaussian functions into the database as a proximity dataset, and use them to correct the design or to simulate the energy density.

You can create a proximity dataset with parameters from the MC simulation by selecting the **Proximity Parameter Database**, dialog ➡ **Data Simulation** subdialog ➡ **Calculation Results** tab (⇒ *Data Simulation subdialog – Calculation Results tab* on page 6-46).

1.8.6 Visualization of electron trajectories

You can call up the visualization of the electron trajectories by clicking on **Show Trajectories** in the **Trajectories** subset within the Simulation Visualizer (⇒ *Simulation Visualizer* on page 6-53).

You can use the visualization of electron trajectories to analyze the behavior of the simulated primary and backscattered electron trajectories in the defined sample.

The following information is shown in the visualization area:

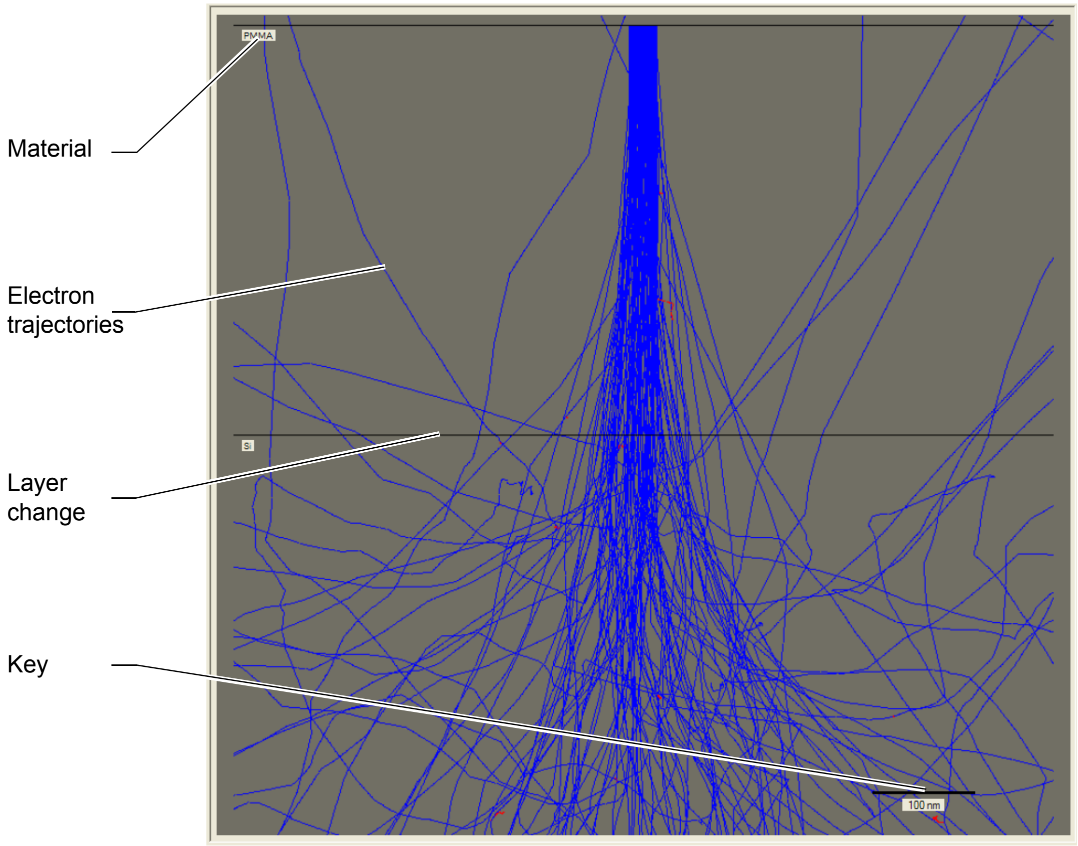


Figure 6-30: Simulation Visualizer – visualization of the electron trajectories

Information	Function
Material	Specifies the designation of the material in the individual sample layers. In the example (⟹ Figure 6-30 on page 6-60), the sample is composed of the resist **PMMA** and the substrate silicon (**Si**).
Electron trajectories	The blue lines indicate the path of the electron trajectories in the sample.
Layer change	Identifies the boundary between the sequential layers of the sample.
Key	Specifies the size of a reference area in nm.

2 Proximity Correction dialog

You can call up the **Proximity Correction** dialog by selecting **Proximity ➡ Proximity Correction...** in the menu bar of the NANOSUITE software.

The menu entry **Proximity Correction...** is only active when you are within the GDSII module editor of the NANOSUITE software.

You can use the dialog to

- Execute proximity corrections with selected proximity datasets,
- Create new proximity datasets,
- Select and edit proximity datasets,
- Determine parameters for partitioning and dispersing the elements of the design,
- Define proximity parameters of the proximity function.

The dialog is divided into the following subsets:

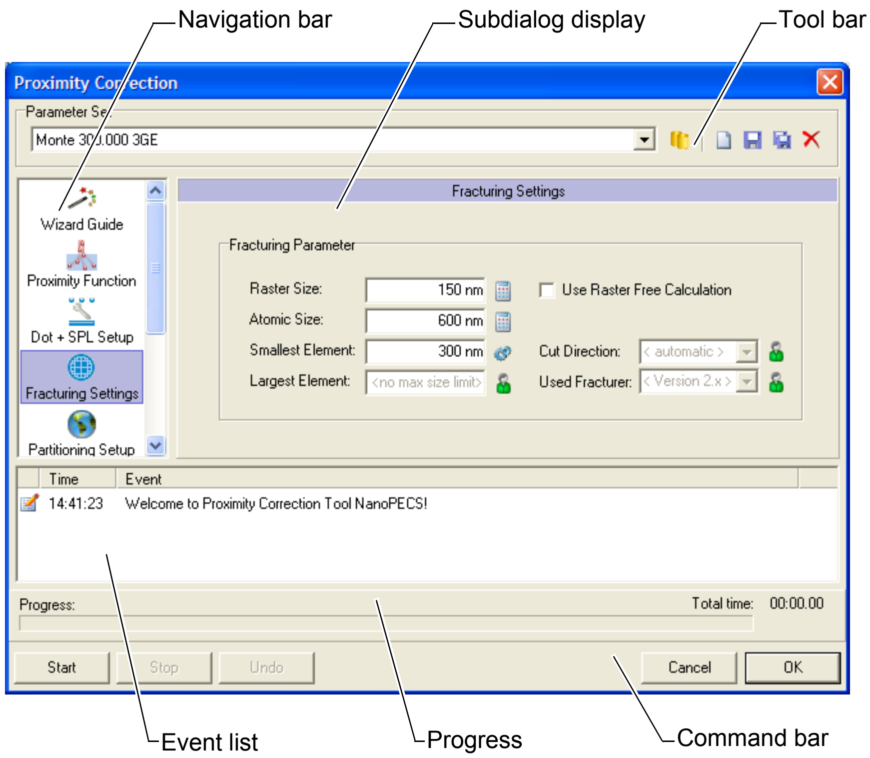


Figure 6-31: **Proximity Correction** dialog

Subset	Function
Navigation bar	To navigate between the subdialogs. The **Proximity Parameter Database** dialog is made up of seven subdialogs: – **Wizard Guide** (⇒ *Wizard Guide subdialog* on page 65) – **Proximity Function** (⇒ *Proximity Effect Function subdialog* on page 70) – **Dot + SPL Setup** (⇒ *Dot + SPL Setup subdialog* on page 72) – **Fracturing Settings** (⇒ *Fracturing Settings subdialog* on page 74) – **Partitioning Setup** (⇒ *Partitioning Setup subdialog* on page 76) – **Dose Settings** (⇒ *Dose Settings subdialog* on page 78) – **Debug Settings** (⇒ *Debug Settings subdialog* on page 79)
Subdialog display	To display subdialogs (⇒ *Subdialog display* on page 63).
Tool bar	To display, call up and manage a proximity dataset (⇒ *Tool bar* on page 64).
Event list	Logs all current and completed procedures (⇒ *Event list* on page 64).
Progress	Displays the status of the currently executed procedure.
Command bar	– To start and cancel a proximity effect correction, – To confirm or reset the entries in the subdialogs (⇒ *Command bar* on page 65).

2.1 Subdialog display

The subdialog display shows the subdialog functions and parameters which can be set.

For some parameters, you can choose between individual and automatic settings. The automatic settings generate preset values based on empirical values or information from specialist literature. They are intended as a recommendation and ensure safe and error-free execution of the proximity effect correction in cases of doubt.

You may encounter the following buttons in the subdialogs:

Button	Function
	Shows that an individual setting can be made. The neighboring text field is activated and you can enter a value. Click on the button to switch to the automatic setting.
	Shows that an automatic setting is activated. The neighboring text field is deactivated and the value of the automatic setting is shown in gray. Click on the button to switch to the individual settings.
	Calls up context-dependent calculation functions to calculate the neighboring value based on suitable heuristics. The meaning of the button is described in each context.

2.1.1 Tool bar

The buttons in the Tool bar have the following functions:

Button	Function
	Calls up the **Proximity Parameter Database** to select a dataset for correction.
	Creates a new proximity dataset.
	Saves changes to the currently selected proximity dataset. The button is only active if you have made settings in one of the subdialogs.
	To save the currently selected proximity dataset with a different name.
	Deletes the currently selected proximity dataset from the proximity database.

2.1.2 Event list

The entries and column headers in the event list have the following meanings:

Entry/column header	Function
	Information about a current procedure.

Entry/column header	Function
✔	Records the successful completion of a procedure.
Event	Descriptive text for the currently running or completed procedure.
Time	Displays the time when a procedure was started or finished.

2.1.3 Command bar

The buttons in the command bar are context-sensitive, i.e. they are only active when the corresponding command can be executed. The buttons have the following function:

Button	Function
Start	Starts a proximity effect correction with the current settings from within any subdialog. Note: You do not need to save the settings beforehand.
Stop	Terminates a proximity effect correction.
Undo	Resets all changes made to the design by the correction.
Cancel	Discards all changes to the design and closes the dialog.
OK	Adopts the calculated corrections into the design and closes the dialog.

2.2 Wizard Guide subdialog

The Wizard Guide allows you to carry out a proximity effect correction step-by-step. You can use the Wizard Guide to

- Select datasets with predefined combinations of voltage and material (**Advice for**... datasets),
- Set the degree of fracturing and thus the precision of the correction,
- Switch to the other subdialogs at any time, modify the parameters listed there and thus change the selected dataset,
- Save the changes under a new file name.

Use the Wizard Guide when

- You would like to familiarize yourself with the functions of NanoPECS™ and the execution of proximity effect corrections.
- You would like to test proximity effect corrections with selected proximity datasets. The Wizard Guide executes a plausibility test of the datasets before a correction is executed. If a correction runs into an error – due to memory requirements being too high, for example – messages to this effect are displayed in the comment field. In such cases, follow the instructions in the comment field.

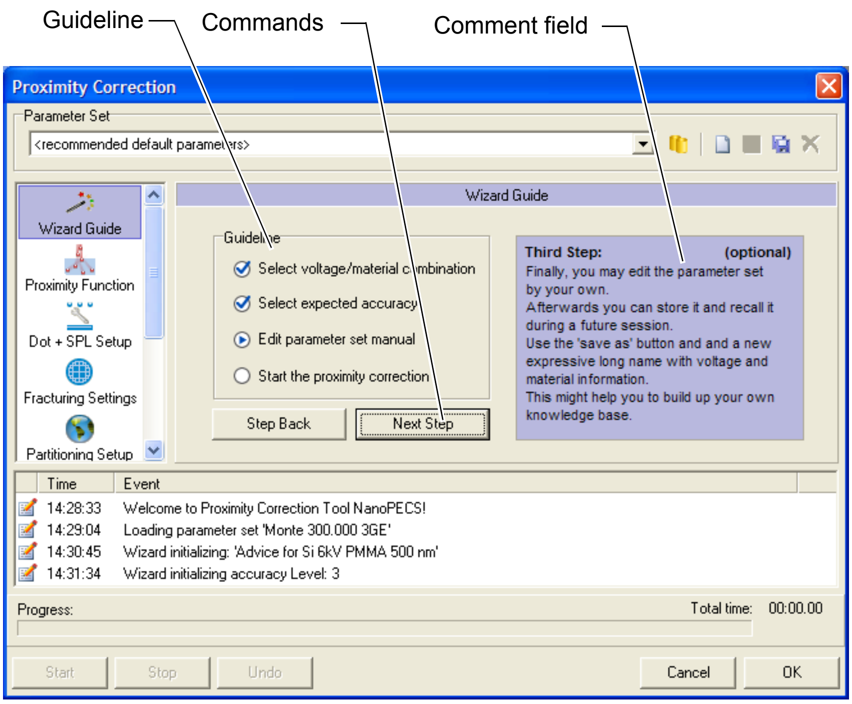


Figure 6-32: **Proximity Correction** dialog – **Wizard Guide** subdialog

The Wizard Guide subdialog is divided into three subsets:

Subset	Function
Guideline	The currently running and completed steps which are executed in preparation for a proximity effect correction are displayed in the Guideline subset. Depending on the step in question, further action may be required. The comment field will ask you to take whatever action is necessary. You can use buttons in the command bar to navigate between the individual steps. The status of the steps is indicated by the following buttons:  Displays the step currently being executed.  Shows that a step has been completed.

Subset	Function
Commands	To navigate within the Wizard Guide. The buttons are context-sensitive; their function depends on which step is currently active (⇒ *Commands* on page 67).
Comment field	Explanations and information about the individual steps, as well as error messages and other notes, appear in the comment field. Depending on the step in question, additional necessary functions are shown above or below the comment field – e.g. a drop-down list to select an **Advice for** dataset or other proximity datasets from the proximity database. Recommendation: Follow the instructions in the comment field.

2.2.1 Commands

The buttons can have the following states:

Button	Function
Next Step	Starts the next step.
Step Back	Returns to the start of the previous step.
Reset Wizard	Shows that a step has been completed.
Start Wizard	Starts the Wizard Guide.
Start Correction	Starts a proximity effect correction.
Finish Wizard	Exits the Wizard Guide after completing a proximity effect correction.

2.2.2 Steps of the Wizard Guide

The following steps are executed within the Wizard Guide:

STEP 1: Select voltage/material combination

In this step, you select an **Advice for** dataset on which the proximity effect correction will later be based. An **Advice for** dataset contains a defined combination of acceleration voltage and material composition as a basis parameter for a new proximity dataset. The definition of these parameters is based on empirical values and theoretical values from specialist literature. During the guided tour, you can modify these parameters using the functions of the individual subdialogs (⇒ *Edit parameter set manual* on page 6-69).

If the step is active, a drop-down list appears above the comment field from which you can select an **Advice for** dataset:

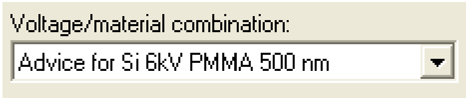


Figure 6-33: **Voltage/material combination** drop-down list

The drop-down list also contains all other proximity datasets in the proximity database. You can also select any other dataset type as a basis for a proximity effect correction.

STEP 2: Select expected accuracy

In this step, you can define the precision with which the proximity effect correction is to be executed.

If the step is active, the **Grade of Accuracy** slider appears above the comment field:

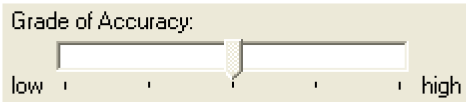


Figure 6-34: **Grade of Accuracy** slider

The precision of a proximity effect correction depends on the degree of fracturing of the design elements (⇒ *Fracturing elements of the design* on page 6-12). If the fracturing is finer, the energy deposition can be assigned to the elements to be corrected in a more targeted way. Too fine a degree of fracturing, however, results in a very high volume of data and requires a correspondingly high computing capability during inversion of the PEM. Depending on the computer capacity of your NANOSUITE system, executing a proximity effect correction may necessitate memory requirements that are so great that the proximity effect correction is terminated. In such a case, the correction should be repeated with coarser fracturing.

The precision level which you set here has a direct effect on the settings in the **Fracturing Settings** subdialog.

STEP 3: Edit parameter set manual

In this step, the Wizard Guide is interrupted to a certain extent and you can individually define more parameters for the selected proximity dataset. To do this, switch to the individual subdialogs and make the desired settings.

Once you have made the desired settings, you can use the button to create a new dataset name with the defined settings. Choose an informative dataset name which contains details of the acceleration voltage and the material composition, at least.

When you click on **Next Step**, the Wizard Guide is resumed and the last step is called up.

The **Edit parameter set manual** step is optional. When you click on **Next Step**, you can skip the step and go straight to the last step. In this case, the parameters of the proximity dataset which you have selected in the first step are taken into account during the proximity effect correction.

STEP 4: Start the proximity correction

In the last step, you can start the proximity effect correction or reset all settings you have made.

You can check the selected dataset and the precision you have set for the correction before you start the proximity effect correction. To this end,
the **Voltage/material combination** drop-down list and the **Grade of Accuracy** slider above the comment field are displayed:

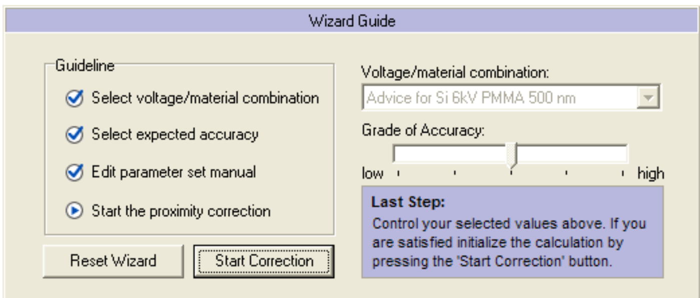


Figure 6-35: **Proximity Correction** dialog – last step of the Wizard Guide

You can start the proximity effect correction by clicking on **Start Correction**.

Once the proximity effect correction is completed, the button **Finish Wizard** appears in the command bar and the comment field requests that you exit the Wizard Guide.

2.3 Proximity Effect Function subdialog

You can define the proximity parameters for the proximity effect function in the **Proximity Effect Function** subdialog. You can define up to three Gaussian terms and one exponential term. The complete proximity function is displayed in the lower subset of the subdialog. The proximity function forms the basis for the execution of a proximity correction.

You can also

– define the proximity effect function manually using the **Proximity Parameter Database** dialog (⇒ *Edit Dataset subdialog* on page 6-33),

– generate the proximity effect function using an MC simulation (⇒ *Data Simulation subdialog* on page 6-37),

– generate the proximity effect function using a simulation with interpolation (⇒ *Data Interpolation subdialog* on page 6-35).

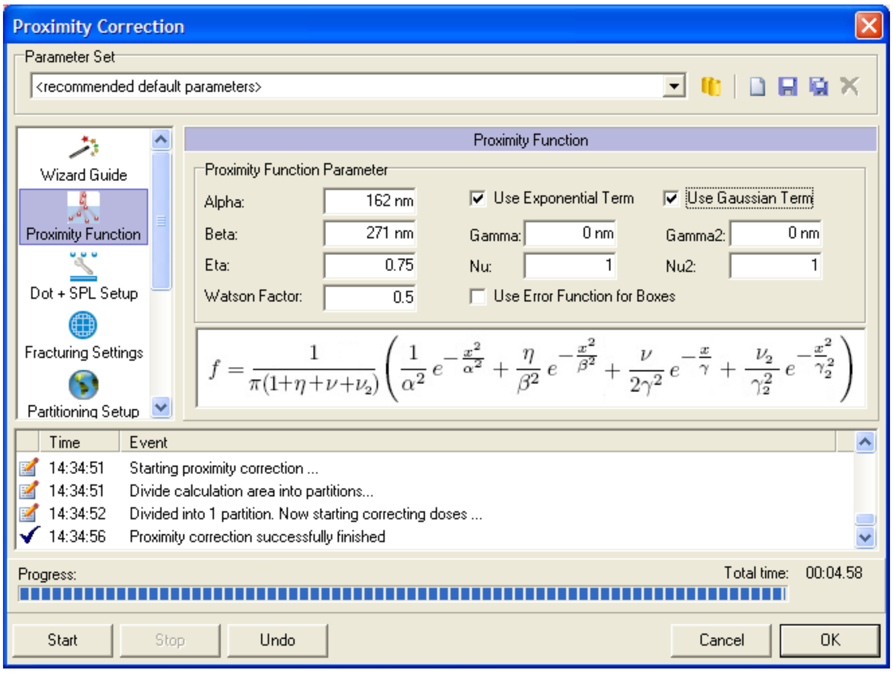


Figure 6-36: **Proximity Effect Correction** dialog – **Proximity Function** subdialog

You can define the following parameters:

Parameter	Function
Alpha	To enter an α value for the beam broadening due to primary electrons in nm. Recommendation: Default value: 2 nm*

Parameter	Function
Beta	To enter a β value for the beam broadening due to backscattered electrons in nm. Recommendation: Default value: 1700 nm*
Eta	To enter a η factor for the efficiency of the back-scattered electrons. Recommendation: Default value: 1.75*
Watson Factor:	To enter a Watson factor for the background radiation. The Watson factor is developed to overcompensate the proximity effect at the edges of a design and is experimentally proved. Recommendation: Use a value of 0.5 for best effect or use 1.0 to avoid overcompensation. Other values can be used if you have proved them by own experiments. Note: The text field is only active if **ShowAdvanced=ON** is defined in the raith_app.db, topic [NANOPECS].
Use Exponential Term	To hide and reveal the entry fields for the exponential term.
Gamma:	To enter a γ value in the exponential term in nm. Recommendation: Default value: 1000 nm*
Nu:	To enter a ν value in the exponential term. Recommendation: Default value: 0, switched off*
Use Gaussian Term	To hide and reveal the entry fields for the third Gaussian term.
Gamma2	To enter a γ_2 value in the third Gaussian term in nm. Recommendation: Default value: 1000 nm*
Nu2	To enter a ν_2 value in the third Gaussian term. Recommendation: Default value: 0, switched off*

Parameter	Function
Use Error Functions for Boxes	To activate and deactivate the correction function for rectangular elements. If this checkbox is activated, a special equation for shape interction of rectangles is used, taking the aspect ratio of the rectangle into account (⟹ *Self-interaction formulae* on page 17). Recommendation: Use this option only for designs which are based on rectangles only. Otherwise the normalization between rectangles and other shapes is difficult. Note: You can see which correction functions are applied in the self-interaction formulae in the appendix to this software description. Note: The text field is only active if **ShowAdvanced=ON** is defined in the raith_app.db, topic [NANOPECS].

* The default values specified here are only test values to check the correction mechanism: Real values depend heavily on the material combination used.

2.4 Dot + SPL Setup subdialog

You can define the width and diameter of lines and points in the **Dot + SPL Setup** subdialog. You need these parameters in order to be able to determine the interaction of lines and points with flat elements.

Specially derived formulae are used for the self-interaction between lines and points (referred to as self-interaction formulae) which you can use to derive the values for width and diameter.

You will find a collection of self-interaction formulae for circles, lines and rectangles in the appendix to this software description.

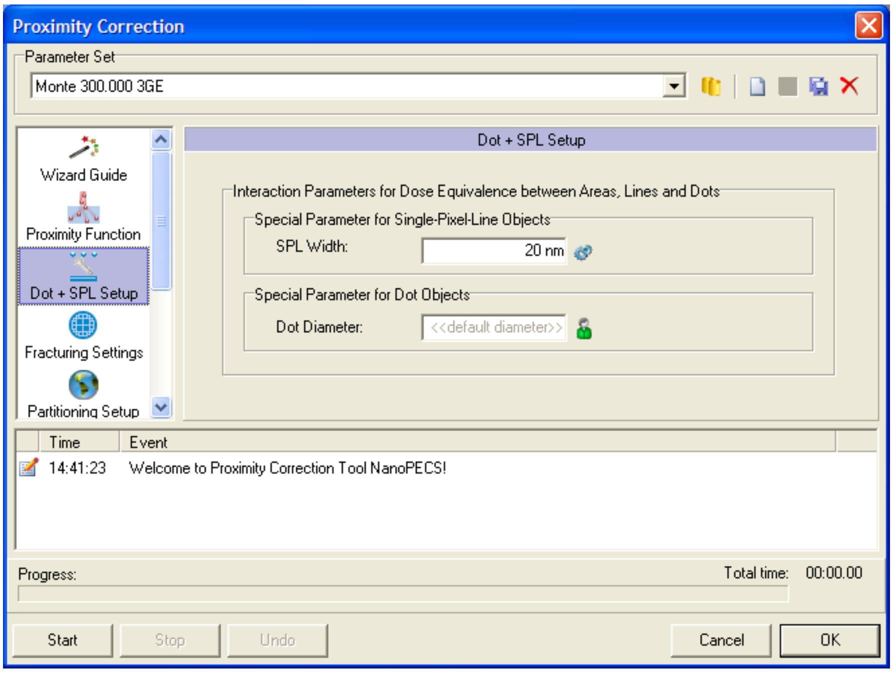


Figure 6-37: **Proximity Correction** dialog – **Dot + SPL Setup** subdialog

You can define the following parameters:

Parameter	**Function**
SPL Width	To enter a value for one-pixel-wide lines in nm.
Dot Diameter	To enter a value for the diameter of a point in nm.

2.5 Fracturing Settings subdialog

You can define the type and degree of fracturing of the design elements in the **Fracturing Settings** subdialog.

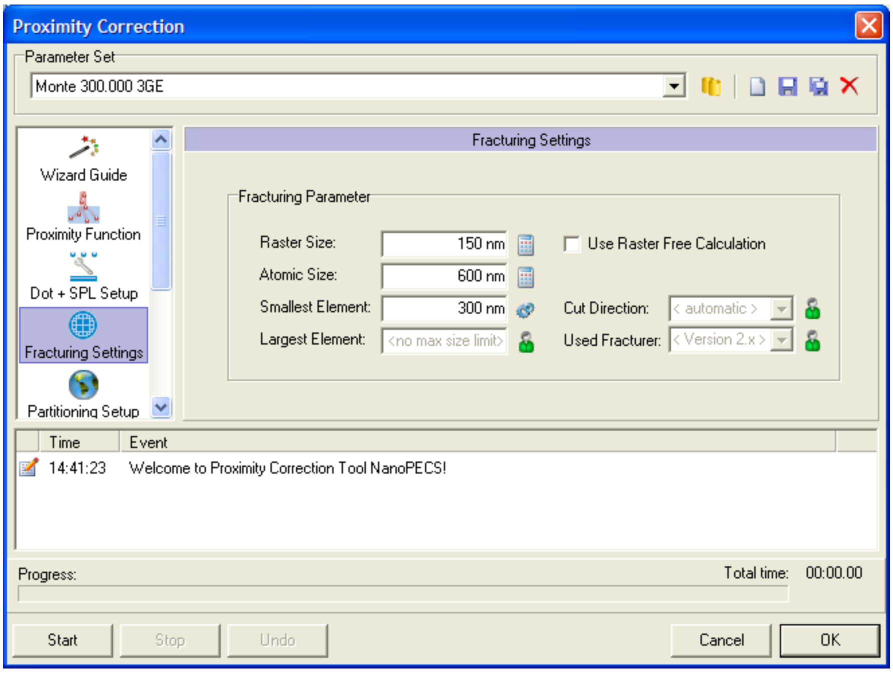


Figure 6-38: **Proximity Effect Correction** dialog – **Fracturing Setting** subdialog

You can define the following parameters:

Parameter	Function
Raster Size	– Defines that the fracturing should be oriented by a raster. – To enter the raster size.
Atomic Size	To enter the atomic size. Atomic size refers to the size of the design elements which are not to be dispersed. Recommendation: Select an atomic size about four times the size of the raster.
	To execute heuristics for the calculation – of the necessary raster size from parameter β (0.1 x β), – of the atomic size based on the raster size (4 x raster size).

Parameter	Function
User Raster Free Calculation	To activate and deactivate automatic raster-free fracturing of the elements. If the checkbox is activated, the **Raster Size** field is automatically deactivated. Recommendation: Have the size of the raster calculated automatically, particularly in the case of very unevenly shaped polygons.
Smallest Element	To define the size of the smallest fracturing shape. The smaller the value, the more precisely the proximity effect correction will be executed. Recommendation: Do not select a too low value, as this may cause memory overload and a termination of the proximity effect correction. Test the execution of a correction with different values – e.g. with the simulation using interpolation (⇒ *Data Interpolation subdialog* on page 35). This will enable you to avoid executing time-wasting proximity effect corrections which turn out to be unsuccessful. Select an automatic setting if you are unsure.
Largest Element	To define the size of the largest fracturing shape. Recommendation: Select an automatic setting.
Cut Direction	To define the cut direction for cutting the design. Recommendation: Select an automatic setting. Note: The text field is only active if **ShowAdvanced=ON** is defined in the raith_app.db, topic [NANOPECS].
Used Fracturer	To select the fracturer. Recommendation: Select the automatic setting so that you always use your system's most up-to-date fracturer. Note: The text field is only active if **ShowAdvanced=ON** is defined in the raith_app.db, topic [NANOPECS].

2.6 Partitioning Setup subdialog

You can define parameters for the partitioning of the design in the **Partitioning Setup** subdialog.

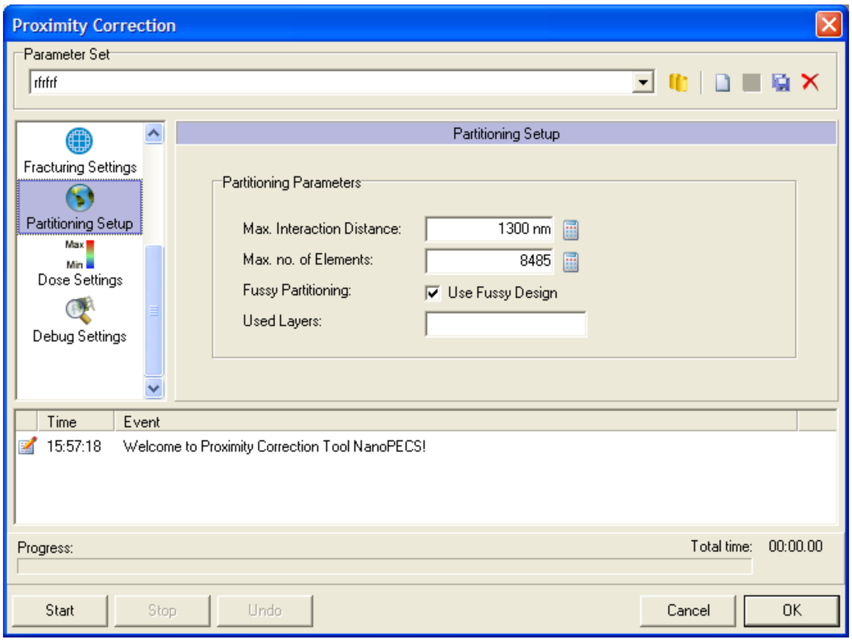


Figure 6-39: **Proximity Correction** dialog – **Partitioning Setup** subdialog

You can define the following parameters:

Parameter	Function
Max Interaction Distance	– Defines the maximum distance between two non-interactive clusters when creating the design partitions. – Defines the maximum size of the overlap area when creating the memory partitions.
	Heuristics for the **Max Interaction Distance**. The value of the automatic setting is derived from the β value of the proximity function (⇒ *Proximity Effect Function subdialog* on page 70). The β value is displayed as bubble-help when you pass over the button.
Max. no. of Elements	Maximum number of elements per memory partition. This value depends on how much working memory is available and can be calculated with the neighboring calculator. Recommendation: Calculate the input with the calculator or choose a lower value to avoid terminations during the correction. If the value is too low, there are a large number of partitions and the time needed for calculation is increased significantly.

Parameter	Function
	Heuristics for the **Max. no. of Elements**. The value of the automatic setting is derived from the available working memory of the computer currently being used. The value should be redefined before a simulation to ensure optimum memory use.
Fuzzy Partitioning	To activate the intelligent fuzzy algorithm to find non-interactive clusters. The fuzzy algorithm is used for partioning by design. This algorithm is our recommendation. Note: The text field is only active if **ShowAdvanced=ON** is defined in the raith_app.db, topic [NANOPECS].
Used Layer	To enter the number of one or more layers of a design which you would like to partition. Enter the numbers of individual layers separated by a comma, and the numbers of layer ranges separated by a hyphen – e.g. 1, 5, 7–12. Note: The layer numbers of a design can be found in the GDSII module under **View ➡ Layer selection**. Note: The text field is only active if **ShowAdvanced=ON** is defined in the raith_app.db, topic [NANOPECS].

2.7 Dose Settings subdialog

You can define parameters for the dose of the corrected design in the **Dose Settings** subdialog:

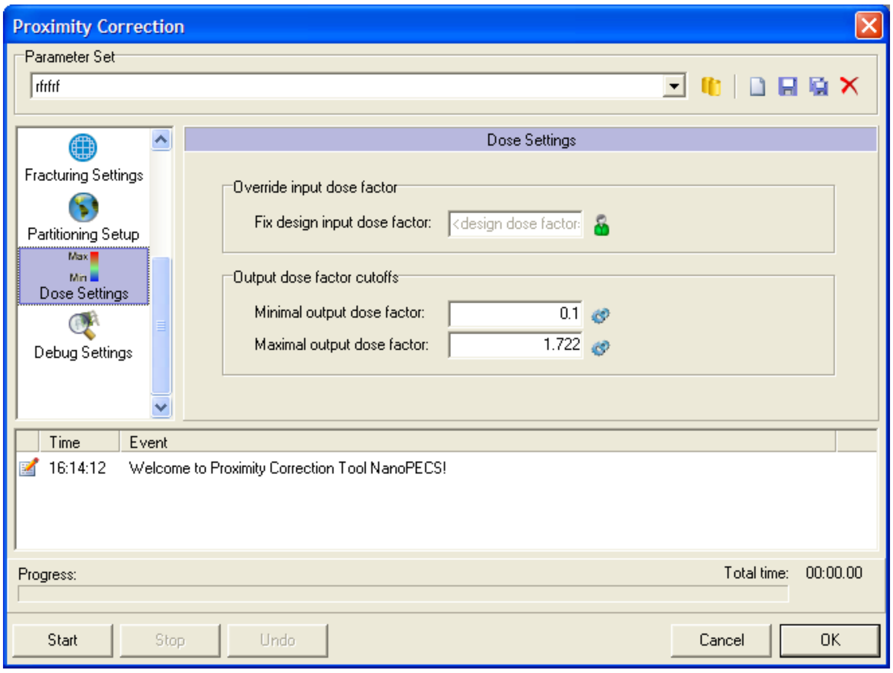


Figure 6-40: **Proximity Effect Correction** dialog – **Dose Settings** subdialog

You can define the following parameters:

Parameter	Function
Fix design input dose factor	To enter a dose factor. The value entered here overwrites the dose factor for every element of a design as defined in the GDSII editor. If you enter a value here you can recorrect a design which you have already corrected. Note: Enter a value here if, for example, you would like to vary the design dose factor as a test.
Minimal output dose factor	To enter the smallest dose factor to be used. Note: The **Minimal output dose factor** may prevent the possible formation of artefacts. Recommendation: Always enter a value < 1 here or select the automatic setting (= 0.0).

Parameter	Function
Maximal output dose factor	To enter the largest dose factor to be used. The default value is developed for area elements. For SPL and Dot elements use self defined limits. Note: The **Maximal output dose factor** may prevent the possible formation of artefacts. Recommendation: Calculate a value using the formula $\eta+\nu+\nu_2$ or select the automatic setting in order to have the value calculated automatically.

2.8 Debug Settings subdialog

You can make settings in the **Debug Settings** subdialog to monitor the execution of a proximity correction.

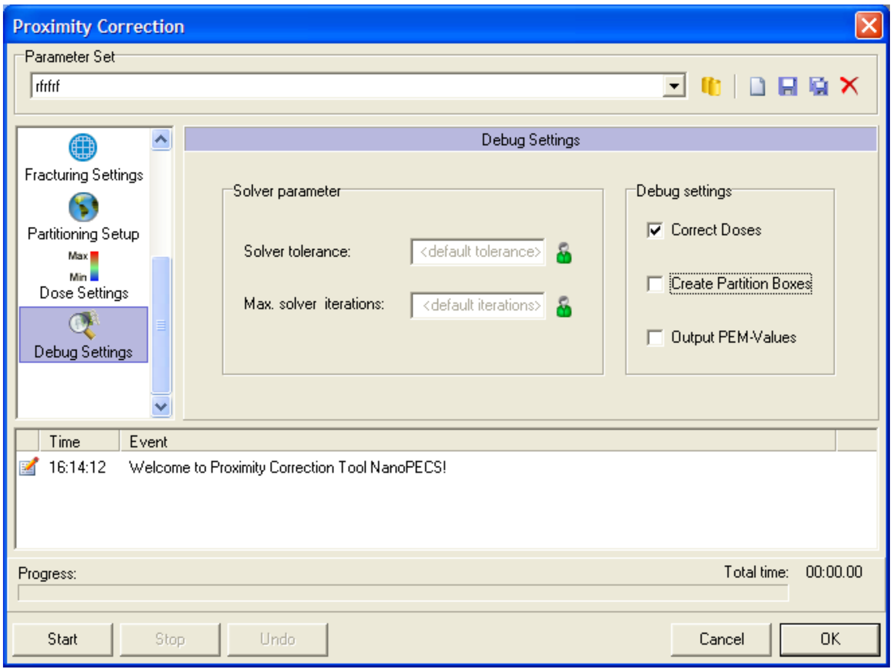


Figure 6-41: **Proximity Effect Correction** dialog – **Debug Settings** subdialog

You can define the following parameters:

Parameter	Function
Solver tolerance	To define the maximum permitted error when solving the linear equation system. Recommendation: If no convergence can be achieved when solving the equation system with the automatic settings, you should try to permit a greater tolerance.

Parameter	Function
Max. solver iterations	To define the number of times the PEM should be inverted until the tolerance value has been attained. Recommendation: Select an automatic setting to avoid long computing times.
Correct Doses	To activate or deactivate the calculation of the energy dose. Note: You can uncheck the calculation of the energy dose to monitor the fracturing of the design elements.
Create Partition Boxes	To display the created partitions in the corrected design. This allows you to monitor in the design whether the partitions have been created correctly as part of the proximity effect correction and whether the overlap areas are sufficient.
Output PEM-Values	To output the applied PEM. Note: Activate this checkbox only for debugging your results as this will cause the output to increase uncontrollably.

3 EDS - <Subdialog> dialog

The **EDS - <Subdialog>** dialog is made up of the following subdialogs:

- **EDS - GDSII Design**
  (⇒ *EDS - GDSII Design subdialog* on page 81)
- **EDS - Energy Density Simulation**
  (⇒ *EDS - Energy Density Simulation subdialog* on page 86)
- **EDS - Cross Section**
  (⇒ *EDS - Cross Section subdialog* on page 88)

Using the subdialogs,

- Execute the energy density simulation (EDS) of a corrected design,
- Check and analyze the energy distribution and the shape of the structuring result after the simulated patterning,
- Set the resist sensitivity.

3.1 EDS - GDSII Design subdialog

You can call up the **EDS - GDSII Design** subdialog by selecting **Proximity ➡ Energy density simulation ...** in the menu bar of the NANOSUITE software.

In the subdialog **EDS - GDSII Design**, you can

- View the energy distribution of a corrected design,
- Make settings for structure size, interaction distance and display properties of the design,
- Draw a cross-section line for the cross-sectional analysis at a particular place in the design.

The subdialog is divided into the following subsets:

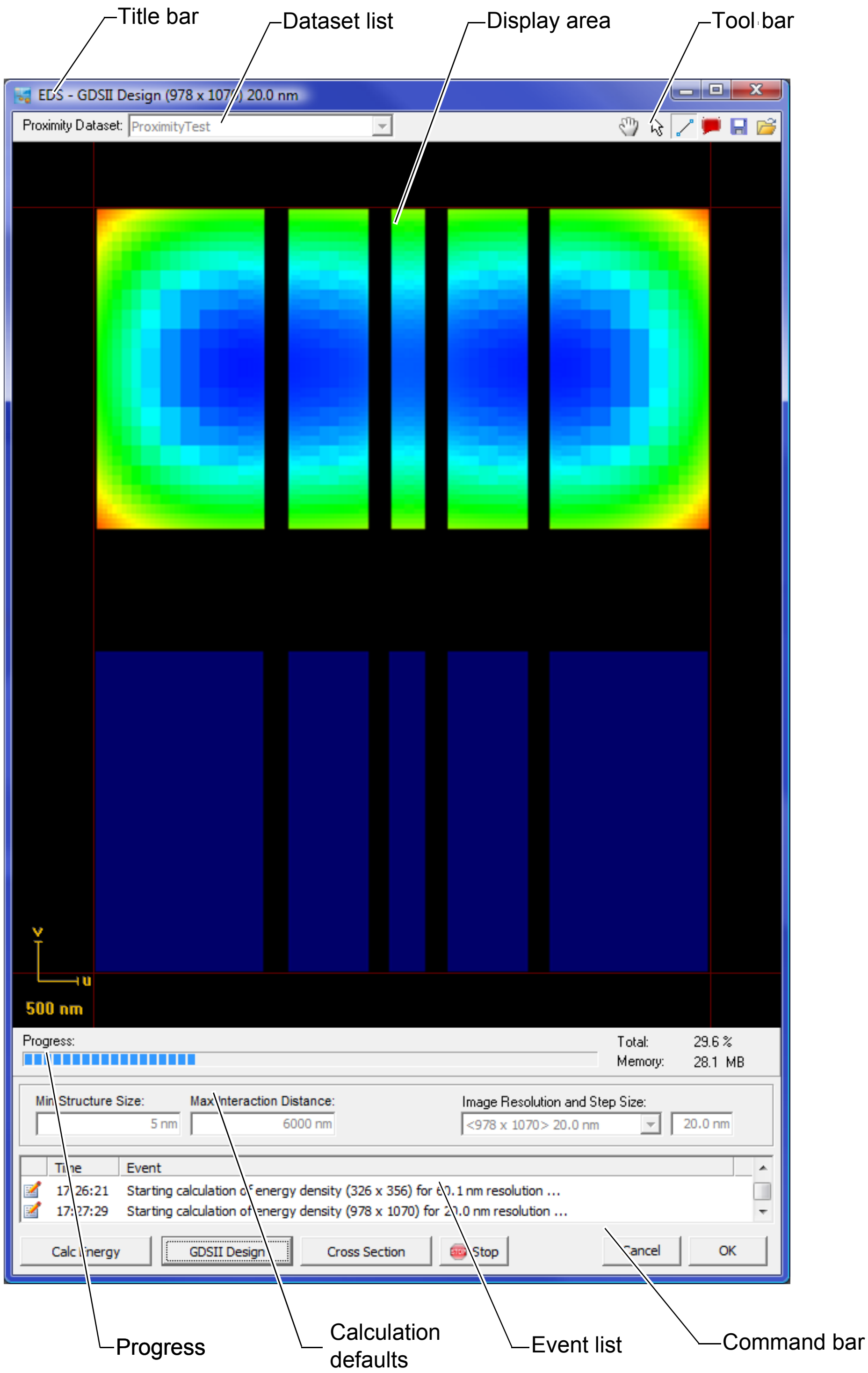


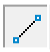
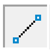
Figure 6-42: **EDS - GDSII Design** subdialog

Subset	Function
Title bar	The title bar shows: – on the left hand side the name of the currently opened subdialog as well as the picture resolution and step size of the display in the coordinate system. – on the right hand side the Windows functions to reduce, expand and close the dialog.
Dataset list	Drop-down list to – display the selected proximity dataset, – select a proximity dataset.
Display area	To display the energy distribution of individual design elements graphically.
Tool bar	With tools and symbols to – work in the visualization area, – select and save datasets (⇒ *Tool bar* on page 83).
Progress	To display the current status of the energy density simulation (⇒ *Progress subset* on page 84).
Calculation defaults	To display and set the resolution of the simulation (⇒ *Calculation defaults subset* on page 84).
Event list	List to display the currently executed and last procedures. You can scroll through the list using the scroll bar on the right-hand edge of the event list.
Command bar	To navigate between individual EDS subdialogs and to execute the EDS (⇒ *Command bar* on page 85).

3.1.1 Tool bar

The buttons in the Tool bar have the following functions:

Button	Function
	To move the display within the visualization area.
	Provides a measuring tool which you can use to read individual values from the graphic.

Button	Function
	To draw a cross-section line in any direction on the design. The cross-section line marks a cross section in the design at the place drawn in. By clicking on Cross Section in the **EDS - Cross Section** dialog, you can display the energy distribution and the resist development in two or three dimensions for this cross-section (⇒ *EDS - Cross Section subdialog* on page 88). To draw a cross-section line proceed as follows: ▶ Select . ▶ Click with the left mouse button at a place in the visualization area, where the cross-section line should start. ▶ Hold down the left mouse button and drag the cursor in the desired direction and for the desired distance. ▶ Release the mouse button at a place where the cross-section line should end.
	To display information about the cross-section in the form of a key.
	Saves the simulation results.
	Loads a saved energy density simulation.

3.1.2 Progress subset

The fields in the Progress subset have the following function:

Field	Function
Progress	Displays the progress of the energy density simulation graphically.
Total	Displays the progress of the energy density simulation in percent.
Memory	Displays the memory requirement of the energy density simulation at the current point in time.

3.1.3 Calculation defaults subset

The fields in the Calculation defaults subset have the following function:

Field	Function
Min Structure Size	To define the smallest possible structure size of an element. If **SPL** and **Dots** are imported from the design, structure sizes must be assigned to them for the simulation.
Max Interaction Distance	To define the largest possible interaction distance in nm. Note: If you enter a low value, the speed of the simulation increases. On the other hand, any interactions which arise are neglected and the simulation becomes less precise.
Image Resolution and Step Size	To set the picture resolution in pixels and the step size in nm. You can choose from the possible combinations from the drop-down list. Note: In each energy density simulation the resolution is refined incrementally: As the visualization is accessible during a running calculation, you can analyze results within a short time. To speed things up, further refinement takes place in the zoomed section.

3.1.4 Command bar

The buttons in the command bar have the following functions:

Button	Function
Calc Energy	Starts and EDS and calls the **EDS - Energy Density Simulation** subdialog (⇒ *EDS - Energy Density Simulation subdialog* on page 86).
GDSII Design	Switches to the **EDS - GDSII Design** subdialog (⇒ *EDS - GDSII Design subdialog* on page 81).
Cross Section	Switches to the **EDS - Cross Section** subdialog (⇒ *EDS - Cross Section subdialog* on page 88).
Stop	Terminates EDS.
Cancel	Discards changes to settings and closes the window.
OK	Confirms and adopts settings.

3.2 EDS - Energy Density Simulation subdialog

You can call up the **EDS - Energy Density Simulation** subdialog by selecting Calc Energy in the **EDS- GDSII Design** subdialog (⇒ Figure 6-42 on page 6-82) .

The result of an EDS is displayed in the **EDS - Energy Density Simulation** subdialog. Here you can

- check the calculation and the result of the energy density simulation,
- draw a cross-section line for the cross-sectional analysis in the **EDS - Cross Section** subdialog.

The **EDS - Energy Density Simulation** subdialog is divided into the following subsets:

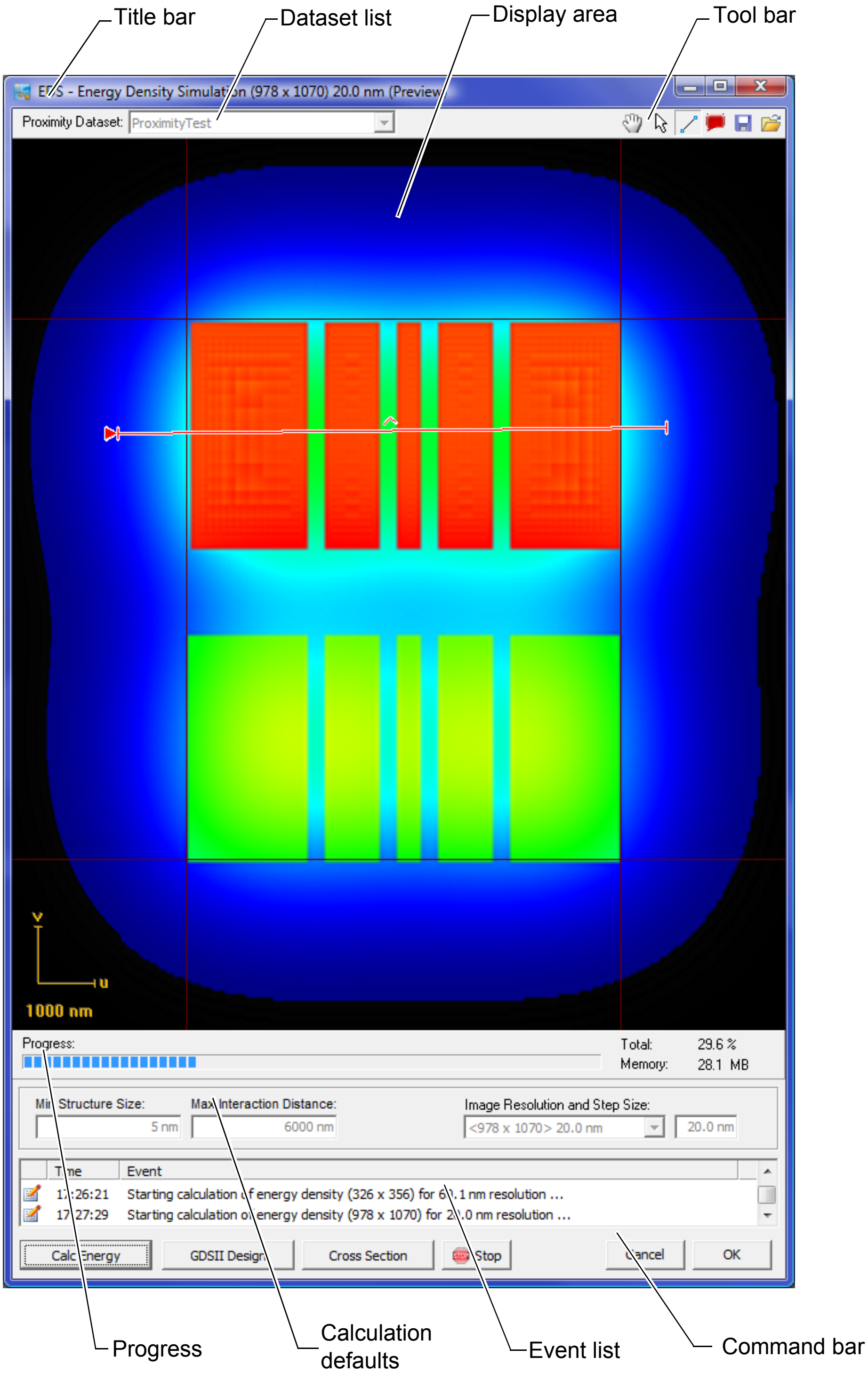


Figure 6-43: **EDS - Energy Density Simulation** dialog

Subset	Function
Title bar	The title bar shows: – on the left hand side the name of the currently opened subdialog as well as the picture resolution and step size of the display in the display area. – on the right hand side the Windows functions to reduce, expand and close the dialog.
Dataset list	Drop-down list to – display the selected proximity dataset, – select a proximity dataset.
Display area	Displays the result of the EDS.
Tool bar	With tools and symbols to – work in the visualization area, – select and save datasets (⇒ *Tool bar* on page 83).
Progress	To display the current status of the energy density simulation (⇒ *Progress subset* on page 84).
Event list	List to display the currently executed and last procedures. You can scroll through the list using the scroll bar on the right-hand edge of the event list.
Calculation defaults	To display and set the resolution of the simulation (⇒ *Calculation defaults subset* on page 84).
Command bar	To navigate between individual EDS subdialogs and to execute the EDS (⇒ *Command bar* on page 85).

3.3 EDS - Cross Section subdialog

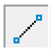
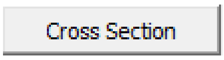
You can call up the **EDS - Cross Section** subdialog by selecting

[Cross Section] in the

- **EDS- GDSII Design** subdialog (⇒ Figure 6-42 on page 6-82) or
- **EDS - Energy Density Simulation** subdialog (⇒ Figure 6-43 on page 6-87).

In the subdialog **EDS - Cross Section** you can obtain data about the energy distribution and the resist development for a cross-section you have defined in the design.

To obtain data for cross section data proceed as follows:

- Change to **EDS- GDSII Design** or **EDS - Energy Density Simulation** subdialog.
- Select the cross section tool .
- Click with the left mouse button at a place in the visualization area, where the cross-section line should start.
- Hold down the left mouse button and drag the cursor in the desired direction and for the desired distance.
- Release the mouse button at a place where the cross-section line should end.
- Click on 

.

  The subdialog **EDS - Cross Section** is opened, and you can

  - View selected areas of a design in cross-section as a line diagram or in 3D view, allowing you to analyze details of the EDS,
  - Import a contrast curve for the resist used and include it in calculating the resist profile to be expected,
  - Change the cross-section in real time using slide controls,
  - Move the cross-section through the design in real time using the mouse and the cursor keys,
  - Switch to the **EDS - GDSII Design** and **EDS - Energy Density Simulation** subdialogs and draw more cross-section lines for the cross-sectional analysis there.

  The **EDS - GDSII Design** subdialog is divided into the following subsets:

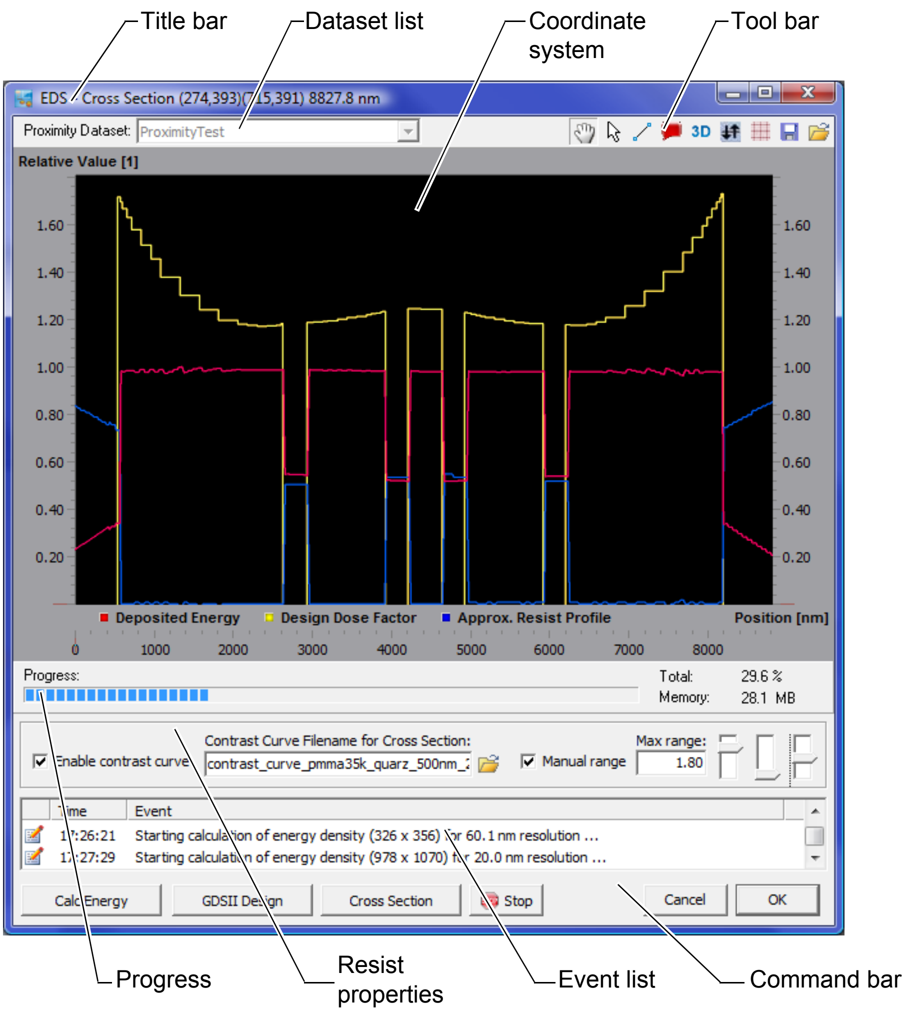


Figure 6-44: **EDS - Cross Section** subdialog

Subset	Function
Title bar	The title bar shows: – on the left hand side the name of the currently opened subdialog as well as the coordinates of the cross-section. – on the right hand side the Windows functions to reduce, expand and close the dialog.
Dataset list	Drop-down list to select a proximity dataset.
Coordinate system	Displays the result of an EDS in cross-section as a line diagram or a 3D view (⇒ *Coordinate system area* on page 91).

Subset	Function
Tool bar	With tools and symbols to – work in the visualization area, – select and save datasets (⇒ *Tool bar* on page 92).
Progress	To display the current status of the energy density simulation (⇒ *Progress subset* on page 93).
Event list	List to display the currently executed and last procedures. You can scroll through the list using the scroll bar on the right-hand edge of the event list.
Resist properties	To define parameters for the resist sensitivity and the type of display in the coordinate system (⇒ *Resist properties subset* on page 93).
Command bar	To navigate between individual EDS subdialogs and to execute the EDS (⇒ *Command bar* on page 94).

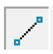
3.3.1 Coordinate system area

The displays in the Coordinate system area have the following meaning:

Display	Function
X-axis	Shows the position of the displayed design elements in nm.
Y-axis	Shows the relative deposited energy as a factor. Also shows the design dose factors and the relative approximated resist profile for comparison.
Deposited Energy	Shows the actual energy distribution of a corrected design after a simulated patterning.
Design Dose Factor	Shows the dose factors of the design.
Approx. Resist Profile	Shows the expected layer thickness of the resist which has developed.

3.3.2 Tool bar

The buttons in the tool bar have the following functions:

Button	Function
	To move the display within the coordinate system.
	To call the cross hairs. You can use the cross hairs to navigate to a desired area of the display and view the appropriate values for **Deposited Energy**, **Design Dose Factor** and **Approx. Resist Profile**.
	To draw a cross-section line in any direction on the design. Note: Click on the button to open the **EDS - GDSII Design** subdialog. Draw a cross-section line and switch back by clicking on Cross Section to analyze the cross-section.
	To display information about the coordinates of the cross-section in the form of a legend.
3D	Switches to the 3D cross-section view (⟹ *3D view* on page 95).
	Inverts the resist sensitivity parameter. Depending on the type of resist used, you can switch between the display of the positive or negative resist.
	Shows and hides a grid in the background of the coordinate system.
	Saves the data of the cross-section in a text file.
	Loads the data of the cross-section.

3.3.3 Progress subset

The fields in the Progress subset have the following function:

Field	Function
Progress	Displays the progress of the EDS graphically.
Total	Displays the progress of the EDS in percent.
Memory	Displays the memory requirement of the EDS at the current point in time.

3.3.4 Resist properties subset

The following control elements are available to you in the Resist properties subset:

Control element	Function
Enable contrast curve	To select whether the EDS should have an underlying contrast curve list. Note: If you do not select a contrast curve list, it will be assumed that there is a direct relationship between the energy deposition and the resist profile.
Contrast Curve Filename for Cross Section:	To enter or select a contrast curve list. The contrast curve list is an ASCII table in which the energy deposition is entered in relation to the degree of resist solvability. The list refers to theoretical or empirical values and specifies the energy deposition at which a certain resist is solved and to what degree. Note: Specify a contrast curve list if you would like to work with known values. If you do not specify a contrast curve list, it will be assumed that there is a direct relationship between the energy deposition and the resist profile. You will find an example of a contrast curve list in the appendix (⟹ *Example of contrast curve list* on page 21).
Manual Range	To select whether you would like to define the upper limit of the Y-axis value scale individually.
Max range:	To enter a maximum coordinate value for the Y-axis. Note: If you do not enter a value in the **Max range:** field, a value oriented to the proportions of the selected EDS section will be preselected. Enter a maximum coordinate value for the Y-axis if you would like to prevent undesired outliers being taken into account in the design.

Control element	Function
Left-hand slide control	To set the value scale of the Y-axis without increments.
Center slide control	To set the brightness of the background in the 3D coordinate system without increments.
Right-hand slide control	To set the resist sensitivity without increments.

3.3.5 Command bar

The buttons in the command bar have the following functions:

Button	Function
Calc Energy	Starts an EDS and calls the **EDS - Energy Density Simulation** subdialog.
GDSII design	Switches to the **EDS - GDSII Design** subdialog.
Cross Section	Switches to the **EDS - Cross Section** subdialog.
Stop	Terminates the EDS. Note: The Stop Button is only visible if the simulation has been started.
Cancel	Discards changes to settings and closes the window.
OK	Confirms and adopts settings.

3.3.6 3D view

The 3D view shows the cross-section of a design in three dimensions:

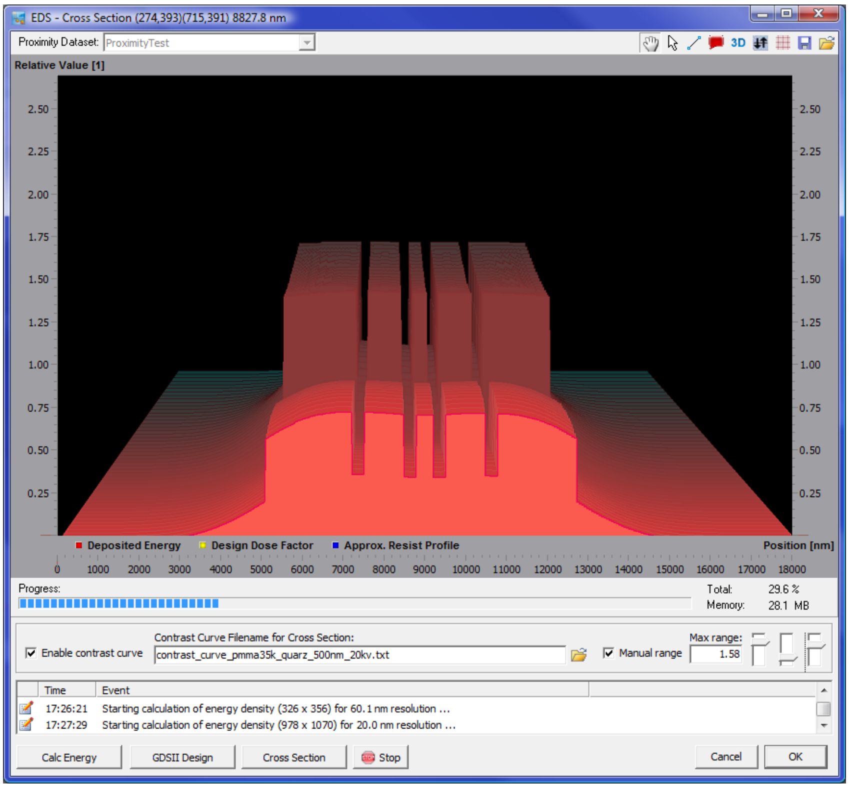


Figure 6-45: **EDS-Cross Section** dialog – 3D view

You can browse through the simulation results by navigating with the mouse. The mouse offers the following functions for navigation:

Mouse action	Function
Left-hand mouse button and drag	Moves the 3D view.
Right-hand mouse button and drag	Rotates the 3D view.
Turn mouse wheel	Zooms in or out on the 3D view.

The commands and functions which can be executed in the 3D view are identical to those in the line diagram view (⇒ *EDS - Cross Section subdialog* on page 6-88).

Chapter 7
Column Control (option)

This chapter gives an overview of column parameters within the **Column Control** function.

The following topics and tasks are included:

- Background information about Group access and differentiation of Groups by color coding, (⇒ *Group access and color coding* on page 7-3),
- Detailed description of the tuning control tools, such as the one-dimensional and two-dimensional slider bars, (⇒ *One-dimensional slider bar* on page 7-5) and (⇒ *Description of control functions* on page 7-15),
- Changing and editing groups and datasets, (⇒ *Creating a new Group or Dataset* on page 7-6),
- How to drag and drop a Column Control parameter set into a Position-list, (⇒ *Drag and drop of Datasets* on page 7-8),
- Step by step explanation of the automated procedure for changing to Column Standby or Column Stop, (⇒ *Switching to Column Standby or Column Stop* on page 7-8),
- How to optimize the aperture alignment, (⇒ *Optimizing the Aperture Alignment* on page 7-17),
- Explanation of how to associate a Writefield to Column Control parameters, in order to carry out an automated patterning procedure, (⇒ *Associating Writefields to Column Control parameters for automated patterning* on page 7-18),
- Detailed description of the Column Control dialog, (⇒ *Column Control dialog* on page 7-20),
- Detailed description of the Fine Tune dialog, including Mag /Foc, Detector, Aperture and Stigmator, (⇒ *Fine Tune dialog* on page 7-34),
- Detailed description of the Focus Wobbling dialog (⇒ *Focus Wobbling dialog* on page 7-46).

You may call up **Column Control** by clicking the (Column Control) icon in the control bar.

The dialogs are displayed to the right, next to the control bar. The text **Column Control** will appear on the control bar:

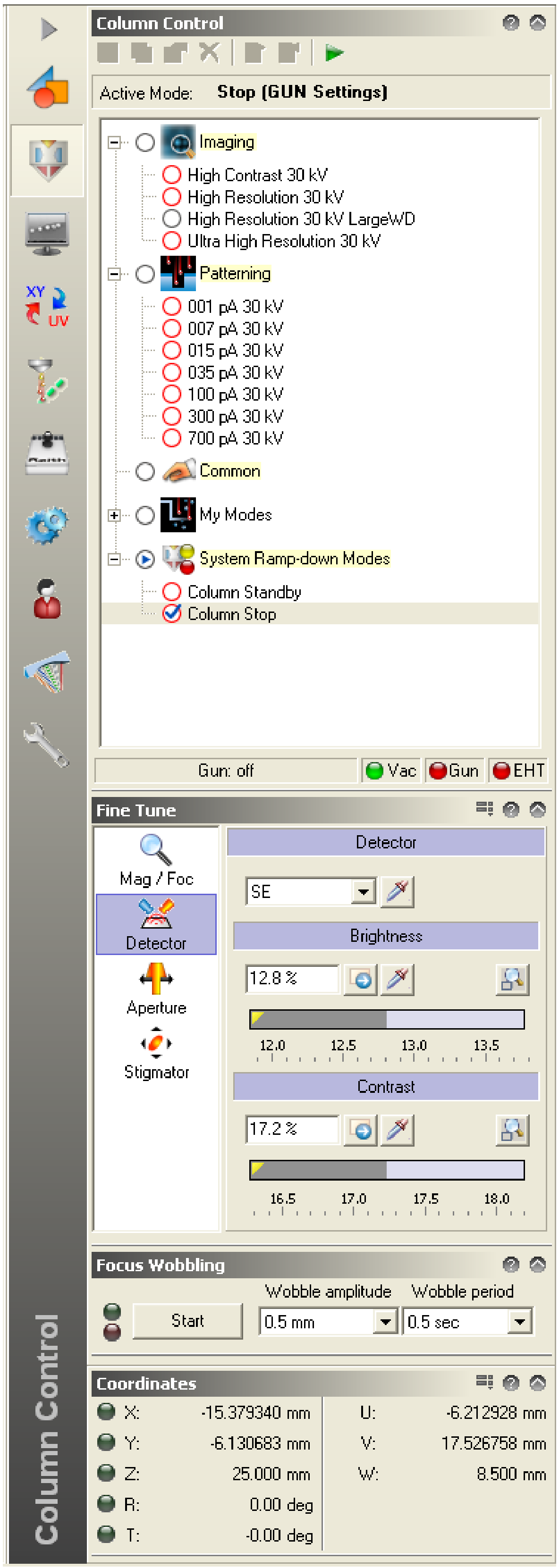


Figure 7-1: Column Control

A Functional Description

1 Group access and color coding

There are a number of **Groups** within the **Column Control** dialog, each containing several parameter sets. All groups and modes are indicated by color codes, to reflect their accessibility, by various users.

The groups with yellow backgrounds in the column control dialog are the public group settings, accessible to all users. These groups may also contain black datasets, which can be edited by the user, depending on the user access rights.

Red circle indicator: Settings within **Images** and **Patterning** marked with a red circle indicator, have been set up by Raith can only be edited by either Raith specialists or SERVICE-level users.

Those settings can, of course, be used by any user, but the parameters for those settings are locked and only accessible by users with the appropriate status.

For example, any user may use these setting, but if the user then adjusts any parameter values during fine tuning, it will not be possible to save those settings under the same name, as the **Save** button is disabled and grayed out.
The user can, however, save the revised settings under a different name, using the **Save as** button. The user may also choose a different **Group** for the new optimized settings. The new settings will now be saved in normal mode and marked by a **black circle indicator**.

Black circle indicator: These parameter sets are accessible to all users and can be edited and saved under the same name. Public access groups, such as **Patterning**, will normally include several such settings, marked by the black circle. These setting can also be used by any user. If, however, the user adjusts any settings, permission to save the new setting under the same name depends on the hierarchy level of the user login, as well as the hierarchy level of the user who created this setting.

Active mode: The active mode indicates the current setting and group. The currently active setting has a blue checkmark against it and the group in which the active setting is situated has a blue triangle in front of the group name.

1.1 User hierarchy

There are three user log-in hierarchy levels, listing from the highest level down.

- SYS
- Expert
- User

Only for the IONLINE system, an additional SYS-level has been created, since for some service tools, the SYS-level is not sufficient to be permitted certain aspects of software access. Therefore a SERVICE level has been created to permit this access.

If, for example, a User created the setting, another user will be permitted to adjust the settings and to save them under the same file name. Any SYS-level user can, of course, also edit and save these settings.

The general rule is that a setting may only be edited and saved under the same name by a user in the same hierarchy level as the creator of the setting, or in a greater hierarchy level than the creator of the setting.

1.2 User's own groups

Any user is permitted to create a new **Group**. A user may, for example create a Group named **My modes**, for ease of identification. This Group name does not have the yellow background, since it is not a public group. Only the user who has created the new group can view and edit any settings listed in this group.

2 Software tools within Fine Tune dialog

Software tools within the **Fine Tune** dialog enable the fine adjustment of the parameters, **Magnification**, **Focus**, **Detector**, **Aperture** and **Stigmator**.

The software tools consist of one-dimensional slider bars, e.g. for **Contrast** and **Brightness** or **Focus** setting, as well as a two-dimensional field, used for the **Aperture** and **Stigmator**.

The functional principle of the software tools is described below.

2.1 One-dimensional slider bar

The one-dimensional slider bar is displayed as a bar on a blue-gray background. The slider bar itself is filled with a dark gray color, depending on the current slider position. In addition, the value of the current position is displayed. When the slider position is changed, the value will change dynamically accordingly. When the user lets go of the ruler, it goes back to the center position, but the scale (shown as ruler) has now been modified according to the new value.

These slider bars are different to the standard Windows sliders, in which the slider bar will stop at the position at which the mouse has been released. For the Raith NANOSUITE software, these slider bars are dynamically adjusted, so that they are always positioned in the center of the slider bar. When the slider bar is being adjusted, click on the slider tool point and drag it to the new position. The cursor does not need to be positioned very precisely to the slider point, since a larger catchment area is built around the slider point to ensure ease of use.

After dragging it to the new position, the mouse button is released and the slider tool point will jump back into the center of the slider bar. The values of the slider bar will be adjusted to the new range.

The user can also move the slider tool point outside the width of the slider box to initiate a larger change in values.

B Tasks

1 Creating a new Group or Dataset

Proceed as described below, to create a new **Group** or a new **Dataset**:

- Open the context menu of an element in the working area.
- Select **New Group** or **New Dataset**, a new dialog will open. (⇒ *Edit Standard Values* on page 7-31).

You can use the **Fine Tune** dialog to establish settings for the column and adopt the current parameter setting into the dialog **Edit Standard Values** or **Edit Dataset Values** by clicking on the button and **Load Current Values** button.

- Adjust the parameters and click **OK**.

Your changes to the selected dataset are saved automatically.

1.1 Changing Dataset groups

The parameters of previously defined dataset groups can be adjusted in the **Edit Standard Values** subdialog.

- Select the dataset group entry that you wish to change in the **Column Control** dialog.
- Click the **Edit Dataset Values** button in the **tool bar**.

The **Edit Standard Values** subdialog opens.

- Make the required changes and click **OK**.

1.2 Replacing the Dataset group pictogram

To replace the dataset group pictogram, proceed as described below:

- Select the dataset group entry that you wish to change in the **Column Control** dialog.
- Place the mouse over the group name and right mouse click. Select **Image** from the dropdown list. The **Select Image** dialog opens.

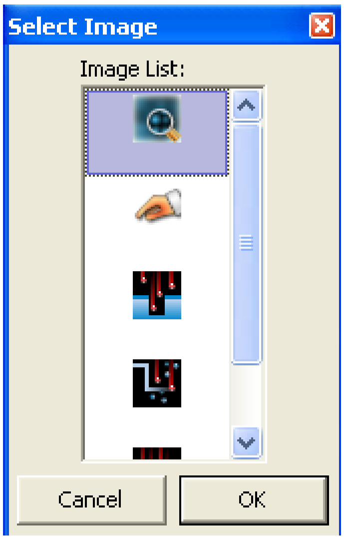


Figure 7-2: **Select Image** subdialog

- Select a new pictogram.
- Click on **OK**.

1.3 Changing Datasets

The parameters of previously defined datasets can be edited in the **Edit Standard Values** subdialog.

- Select the dataset entry that you wish to change in the **Column Control** dialog.
- Click the **Edit Dataset Values** button in the tool bar.

The **Dataset Values** subdialog opens (⇒ *Edit Dataset Values* on page 7-26)

You can use the **Fine Tune** dialog to define settings for the column and adopt the current column parameters by clicking on the **Current Values** button.

- Make the required changes and click **OK**.

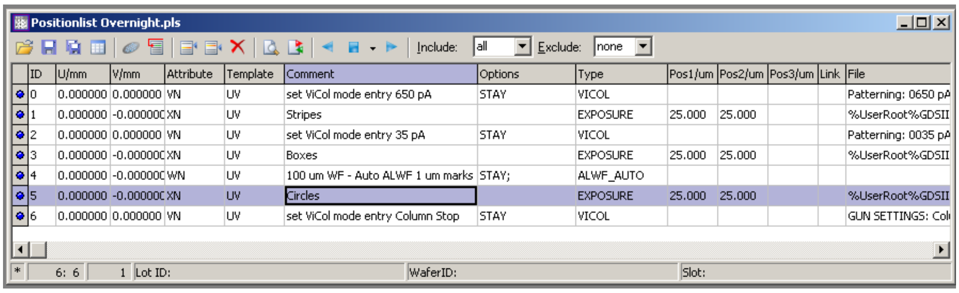
The SERVICE-level user can unlock the lock to gain access for editing all parameter sets, even those which are locked for the normal user, such as the parameter sets indicated in red. The SERVICE-level user can unlock any parameter set to edit it and after saving the new values, the parameter set can be locked again, by placing a checkmark next to **Locked**.

The philosophy of the software is that a normal user never has to access the **Edit Dataset** dialog to edit an existing locked column parameter set. The user can choose any suitable existing parameter set from the column control groups. The user can then edit an existing parameter set, for example by selecting a different aperture. After the fine tuning has been performed, on an image, the user can then save all new settings under a new parameter set in the same column group via the **Save as** button.

In order to change the high tension voltage to a new setting, the **Edit Dataset Values** dialog must be called up. However, a user should be able to select a suitable high tension voltage from one of the different column groups.
In the **Fine Tune** dialog, it is not possible to change the high tension voltage. The voltage is only displayed for information purposes for the IONLINE system.

1.4 Drag and drop of Datasets

You can add datasets to the Positionlist, using drag and drop, before a patterning task, to set up complex batch nanostructuring jobs, each requiring different column operating modes. The figure below shows an example of a Positionlist completed in this way:



ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type	Pos1/um	Pos2/um	Pos3/um	Link	File
0	0.000000	0.000000	VN	UV	set ViCol mode entry 650 pA	STAY	VICOL					Patterning: 0650 pA
1	0.000000	-0.000000	XN	UV	Stripes		EXPOSURE	25.000	25.000			%UserRoot%GDSII
2	0.000000	0.000000	VN	UV	set ViCol mode entry 35 pA	STAY	VICOL					Patterning: 0035 pA
3	0.000000	-0.000000	XN	UV	Boxes		EXPOSURE	25.000	25.000			%UserRoot%GDSII
4	0.000000	-0.000000	WN	UV	100 um WF - Auto ALWF 1 um marks	STAY;	ALWF_AUTO					
5	0.000000	-0.000000	XN	UV	Circles		EXPOSURE	25.000	25.000			%UserRoot%GDSII
6	0.000000	0.000000	VN	UV	set ViCol mode entry Column Stop	STAY	VICOL					GUN SETTINGS: Col

Figure 7-3: Section of a Positionlist

1.5 Switching to Column Standby or Column Stop

The column is switched to standby or stopped by selecting the corresponding entry under **System Ramp-down Modes**. You can find these entries in the tree structure below the dataset.

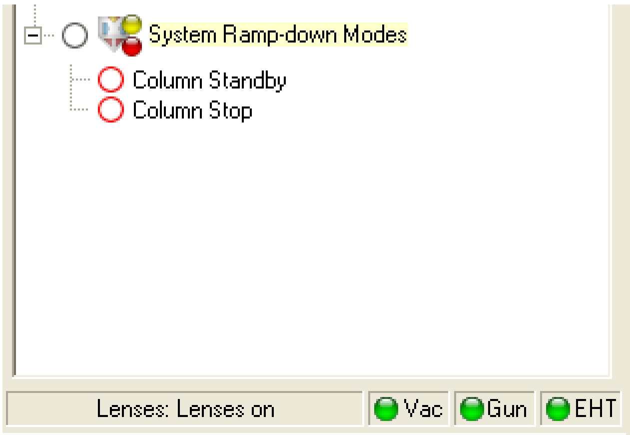


Figure 7-4: Column Standby

- Select **Column Standby** or **Column Stop**.

The column switches to the selected status. You can see the current status indicated by the messages and LEDs of the status row.

1.5.1 Switching on the gun

The gun can be switched on by either clicking on **Column Standby** or by initiating one of the column parameter sets.

Use the **Gun\Start** and **EHT\EHT ON** commands in the status bar of the **Column Control** window to switch on the gun and EHT.

To switch on the power supply monitor, click on the context menu in the **Fine Tune** dialog. Selecting the **Power Supply Monitor** will open a small dialog displaying hardware information about the column, such as condenser lens, objective lens, gun source and gun extractor. The progress on changing the gun status can therefore be monitored.

Note that the continuous read-back of the hardware information via the power supply monitor is a significant system load. Only turn on the polling of the power supply monitor for special requirements and for short periods.

- The gun extractor is applied to the source.
- When the instrument is in **Column Stop** mode and is switched to operating mode, e.g. by selecting a column parameter set, the procedure to transfer the system from **Column Shutdown** to operating mode is performed automatically, incorporating the **Column Standby** mode as an intermediate step during the start up procedure.

- Alternatively, the user can select **Column Standby** and once the system is in Column Standby, either select **Activate selected mode** from the Context menu via the right mouse button, or click the green **Play** button in the **Column Control** tool bar.
- The status of the start-up procedure can be monitored. The example below shows the IONLINE initialization.

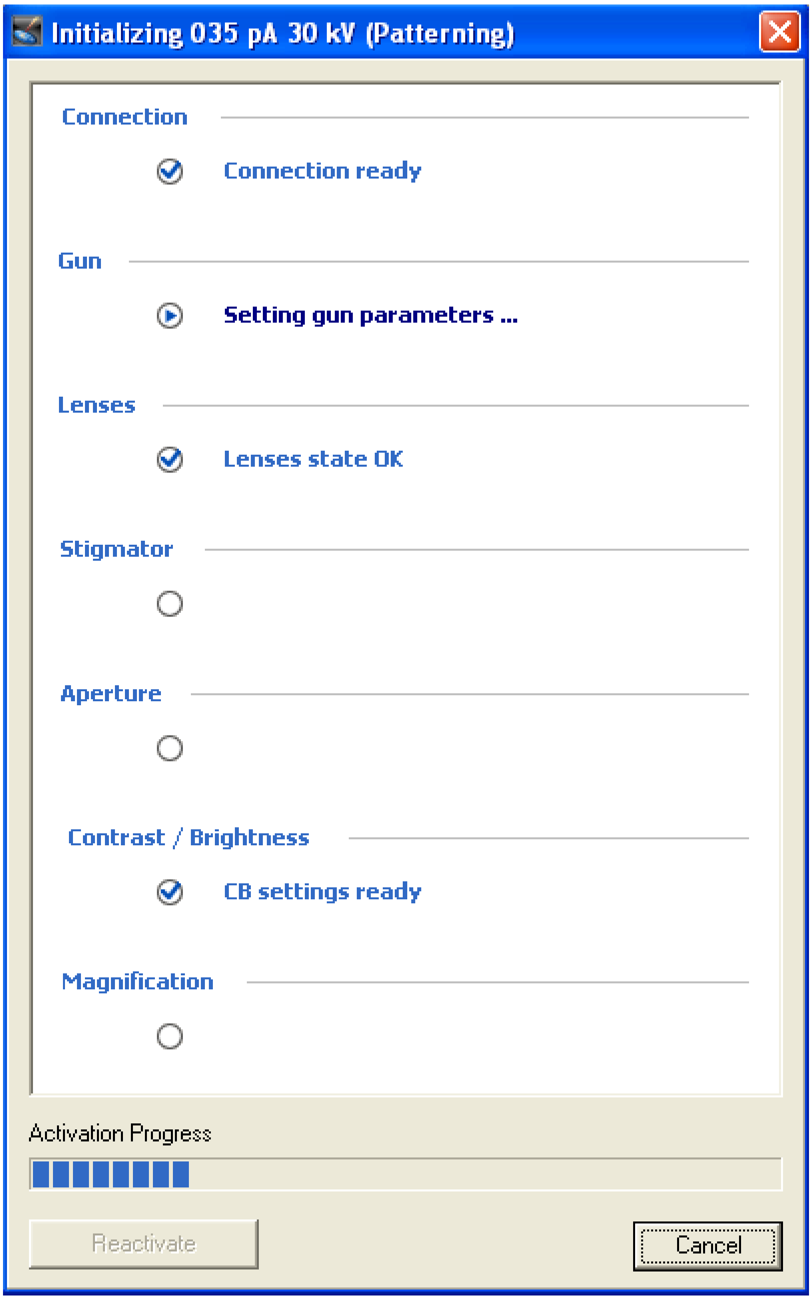


Figure 7-5: Initialization process activated, showing the IONLINE status

When Column Standby is activated, the following steps will be automatically performed and the status can be viewed.

- Gun source power supply is set to high emission current, the Ga emitter is switched on, applying this emission current.
- When the Ga emitter starts emitting, the resulting high voltage is typically 12 kV.

The voltage depends on the Ga emitter properties. It will vary slightly between different Ga emitters. It is not possible to set both the current and the voltage, as both parameters are inter-dependent. The current is typically defined and the voltage will result from the defined current, depending on the individual Ga emitter characteristic, it will be typically 12-14 kV.

- Once the gun source power supply has been set to the required current, a few seconds delay will enable the Ga emitter to stabilize.
- The current is then reduced step by step to 3.2 µA.

▶ The system is now in **Column Standby** mode. Everything is switched off apart from the gun source, which is set to the default emission current, set at the emission threshold value.

▶ The **Status** bar of the **Column Control** dialog shows green **vacuum LED** and **gun LED**, but the **EHT LED** is still red, as the system is in **Column Standby** mode.

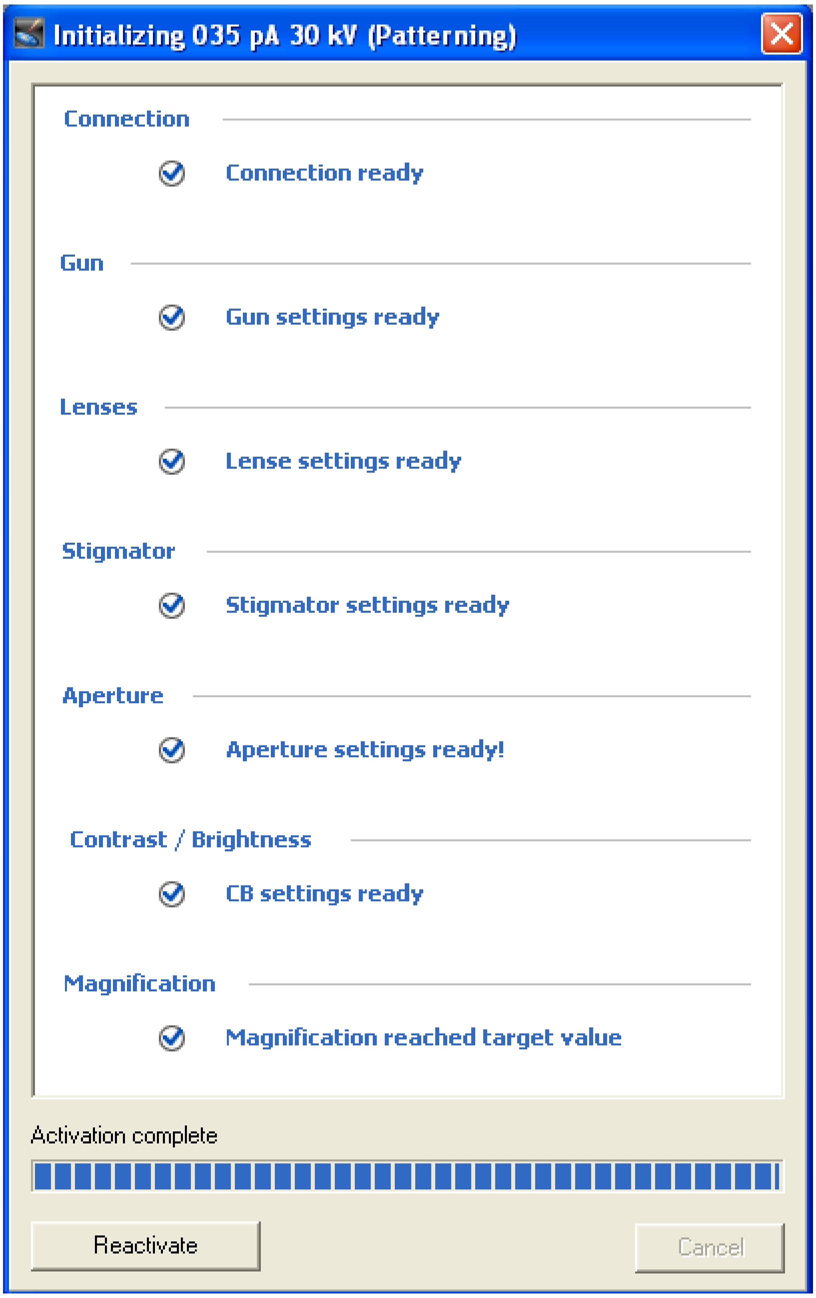


Figure 7-6: Initialization process complete

1.5.2 Recommended start-up procedure

Column parameter sets can be activated either from **Column Shutdown** or **Column Standby** modes. In either case, the system will automatically follow the start-up procedure. It is recommended to always start the column by activating a column parameter set.

▶ Select a column parameter set and click on **Play** button in the **Column Control** tool bar.

- The **Initializing** window will be displayed.
- The initialization will automatically go through check routines, including appropriate values and checking that the connections to the various power supplies are all ok.
- The values for the gun, lenses, stigmator, apertures, contrast, brightness and magnification, as defined in the selected column parameter set, are then established.

- Once a step in the initialization is successfully completed, a check-mark is placed next to it, e.g. connection ready.
- When a step is currently active, for example the gun setting values are being initiated, a **Play** button in a circle is displayed next to it.
- When a step can not be performed successfully, a red cross will be displayed as a warning signal.

- At the bottom of the **Initialization** window, a bar is displayed to illustrate the progress for the column start-up procedure. The complete initialization process takes typically 1-2 min. Once completed, the initialization window will close, when the chamber vacuum is within specification.
- To cancel the start-up procedure, the **Cancel** button stays enabled, while the initialization is in progress. When the Cancel button is clicked, the EHT ramp up will be paused and all electronics are transferred to safe default values, firstly ramping down the current to the Ga emitter source to the default threshold value.

1.5.3 Procedure for failed initialization

- If the initialization could not be completed successfully, the **Initialization** window will stay open, with a red cross displayed next to the step at which the error occurred. Clicking on the **Re-activate** button will start the initialization process again.

It is also possible to re-activate the initialization process after an error has occurred, by clicking on the **Display** button. All values will be checked again.

---

2 Adjusting the sensitivity range in the Fine Tune dialog

It is possible to adjust to different sensitivity ranges, indicated by a range of colors.

Red = coarse setting

Yellow = medium setting

Green = fine setting

- To change between the three settings, use the mouse wheel to toggle between **Coarse**, **Medium** and **Fine** settings.
- Move the cursor to the element for which you wish to change the sensitivity setting. The mouse wheel will become selected for this element.
- Moving the mouse wheel upwards will change to a finer setting, moving it downward will change to a coarser setting.
- When the setting is set to **Fine**, moving the slider bar will change the values only over a small range.

You may save the controller sensitivity as the new default. To do so, click on the right-hand side of the dialog title bar and select the command **Save Controller Sensitivity** in the context menu.

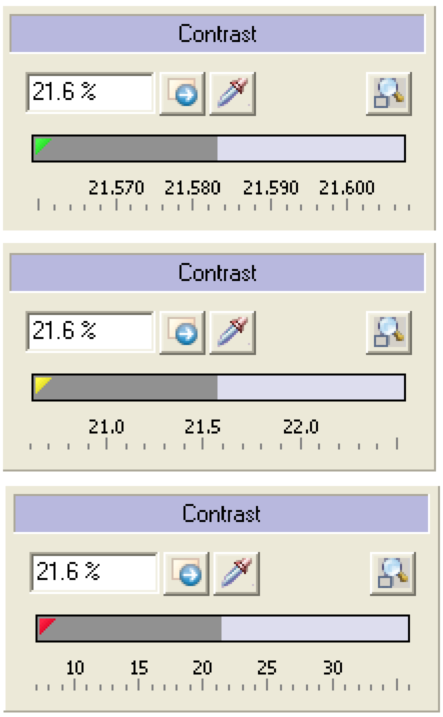


Figure 7-7: Changing the sensitivity range

The red triangle color in the slider bar has been chosen for the coarse setting, to give the user a warning indication, that the movement of the slider bar will result in large changes in the parameter.

2.1 Description of control functions

The control element of the slider bar can be adjusted via the slider bar or via the keyboard. For the functional description of the slider bar, please refer to the **Functional Description**, (⇒ *Functional Description* on page 7-3)

2.1.1 One-dimensional slider bar control

▶ It is recommended to monitor the numbers underneath the slider bar, as they give an indication of the status of the parameter, within the overall available range. For example, when changing the working distance, the user can observe by how much, in mm, that the working distance has been adjusted, when moving the slider bar.

It is always recommended to take extra care at the low and high ends of the movement range. The number should be used to gain a feel for the sensitivity of parameter to the movement of the slider bar.

Most of the parameters vary over a linear scale. The magnification setting, however, has a logarithmic scale.
It is recommended that the user should experiment with the slider bar and monitor the change in the number values underneath, until familiar with slider bar sensitivity in the various ranges.
The logarithmic scale for the magnification is very useful, as a magnification of 300,000x only requires very fine adjustments, whereas a low magnification of e.g. 2000x would require larger adjustments.

2.1.2 Keyboard control

The control element of the slider bar can also be achieved via the keyboard. This can be used for the one-dimensional, as well as two-dimensional tuning.

Prior to using the keyboard control, the control function has to be assigned by hovering with the cursor over the control function to be selected.
While using the keyboard control, it is recommended to keep the cursor stationary. If the cursor is accidentally moved to a new control function, this new control function would be assigned and the keyboard control would now control the new control function.

- The sensitivity range between **Coarse**, **Medium** and **Fine** can be toggled with **Page up**/**Page down** keys on the keyboard.

2.1.3 Control tips for advanced users

- Once the cursor is placed over a control element, this control element remains assigned to the mouse/keyboard controls, even if the mouse cursor is moved away. The currently assigned control element is displayed graphically.
- For example, **Brightness** and **Contrast** can be changed via the keyboard arrows. The keyboard enables very defined movements, e.g. three steps to the left and then three steps back to original position again.
- Each control element also has a text input field, in which the user can enter specific values. Clicking on the arrow or pressing **Enter** on the keyboard will activate the entered value.
- Using the **Pipette** button, the current value can be read.

When the one-dimensional slider bar has been activated by clicking it, a value may be entered and confirmed by pressing **Enter** on the keyboard or by using the shortcut **E** on the keyboard. E = Enter, to confirm the value. The user can also use a **left mouse button** double click on the slider bar to activate the setting.

2.1.4 Adjusting the two-dimensional tuning field

The two-dimensional tuning field functions use a similar principle to the one-dimensional slider bar.

The marker within the two-dimensional field will be displayed differently, depending on the function. For example, **Aperture** is shown as cross-hairs, **Stigmator** is shown as a blue dot.

The marker inside the two-dimensional field displays the current XY values. The mouse wheel changes the sensitivity ranges, as described for the one-dimensional slider bar. A catchment area is also implemented into the full field. The user can click anywhere in the field to move the marker into any direction by dragging the mouse left/right or up/down.

Clicking inside the field will not move the marker to the position, as otherwise accidental movements of the marker over large ranges might occur, e.g. the stigmator value could be moved by mistake over an excessive range.

- The first mouse click will not adjust the value at all, it simply enables the control element for dragging.

- In the two-dimensional tuning field, there is an additional side sliderbar. The user can click on the small black triangles, which will change increment values in defined small steps along one axis only.
- The user can also click in between the black triangle and the slider bar. This will adjust the value in larger steps.
- In addition, the current **XY%** data are displayed, with a text input field, in which the user can enter a value. Using the **Pipette** button, the current value can be read.
- The two buttons in front of the text input field normally have checkmarks, indicating that they are active. If a red cross is displayed, e.g. next to the X value, then the X axis would be disabled. When the user wishes to move the control element, the X value would remain constant and only the Y value can be adjusted. This can be a useful feature, if only one axis needs to be tuned.
- Only one of the axes can be disabled at any one time. Clicking on the checkmark of the Y axis for example, would toggle it to the red cross, but at the same time, the X axis will be enabled again, showing the checkmark, since it would not make sense to have both axes disabled at the same time for tuning.

2.1.5 Keyboard control for a two-dimensional field

The keyboard control functions in a similar way to the one-dimensional slider bar. This time, however, all four arrows are available on the keyboard for the two-dimensional tuning field. The control element can be moved via the arrows **up/down** and **left /right**.

2.2 Optimizing the Aperture Alignment

The apertures can be selected via the dropdown list. In the IONLINE system, the aperture must be physically driven to a position at which it is situated under the beam. Typically, 12 apertures are used in the system.

For each aperture, the size of the aperture is displayed, followed by a number, in brackets, which corresponds to its physical position on the carrier.

- The user can select the aperture. For the IONLINE system, the software sends a position information signal to the motor for the aperture position. This is the default position of the aperture center and the motor will drive the aperture to this position. For the electron beam (Raith EO) system, the electron beam will be positioned in such a way that the beam will be directed through the center of the selected aperture. The position for both the Raith EO and the IONLINE system will not be the optimum position, therefore the astigmatism must be compensated and optimized by setting the **Aperture Alignment**. The Aperture Alignment depends on the following factors: acceleration voltage, lens settings, working distances etc.

- Using focus wobbling, you may adjust the optimum position of the aperture, depending on the current operating conditions.
- The XY values are the values in X and Y axes and the % values are the position relative to the overall driving distance.
- If you do not know the current position, it is recommended to enter zero, as this corresponds to the system service position.
- Perform fine tuning, starting from the zero position. Fine tuning is always performed on a live image, using focus wobbling.

2.3 Associating Writefields to Column Control parameters for automated patterning

- Within a column control parameter set, it is possible to link to an associated writefield, using the **Edit Dataset Values** dialog. (⇒ *Edit Dataset Values* on page 7-26).

The writefield can only be assigned to a column parameter set via the **Edit Dataset Values** dialog.
The **Fine Tune** dialog enables the setting of Working Distance, Stigmator Alignment and Aperture Alignment.

- You will already be familiar with the **Writefield Manager** dialog, in which the parameter sets are listed with various magnifications and writefields.
- The same list is shown as a dropdown list in **Associated writefields**, where a predefined parameter set may be selected. This writefield is then assigned to the column control parameter set.
- Once a writefield is assigned to the column control parameter set, you can drag and drop the selected **Column Control** parameter set into the Positionlist.
- As a writefield was assigned to the Column Control parameter set, the writefield and magnification will now be automatically set, as soon as the Positionlist is executed.
- When the Positionlist is scanned, the Column Control parameter set will be adopted, such as acceleration voltage, aperture setting etc.
- It is possible to include a number of Column Control parameter sets, each assigned to a different writefield in the Positionlist.
- When the Positionlist is scanned, the positions are scanned in sequence. For each position, the specific Column Control parameters and writefield assignments can be executed.

- The patterning can therefore be run automatically.
- For a pattern in which a large structure is to be written, select a Column Control parameter set with a high beam current and a large writefield.
- For a pattern in which a small structure is to be written, select a Column Control parameter set with a low beam current and a smaller writefield.

C Software Reference

Clicking on the Column Control icon in the control bar, opens the following dialogs: **Column Control**, **Fine Tune**, **Focus Wobbling** and optional **Signal Mixer**.

Column Control consists of the following three dialogs:

- **Column Control** dialog for:
  - generating, saving, loading and adjusting column datasets,
  - controlling the gun and the high voltage (EHT). These are monitored via the **Status** LEDs,
  - pumping and ventilating the chamber.
- **Fine Tune** dialog to adjust the current column parameters.
- **Focus Wobbling** dialog to switch focus wobbling on and off with user-defined amplitude and period.

1 Column Control dialog

You can use the **Column Control** dialog to:

- group column parameters into datasets, which you can call up again at any time
- group similar datasets into **Groups**
- define defaults (standard values) for the corresponding datasets.

1.1 Column Control Tool bar

You can use the tool bar to select the commands for modifying datasets and groups.

You can access the current executable commands via the context menu.

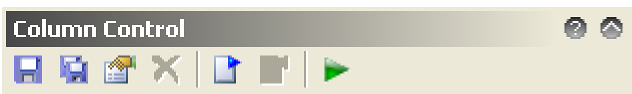
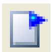


Figure 7-8: Tool bar of the **Column Control** dialog

Control element	Function
	To save the settings. This is only enabled when parameters have been changed and the user is permitted to edit the dataset.
	To save the settings under a new name.
	To change the settings in the selected dataset. A new dialog will open, in which parameters can be edited.
	To delete a parameter set. When the **Delete** button is pressed, a warning prompt is displayed by the software, which must be confirmed by the user, before a setting is deleted.
	To export data as .txt files
	To import data as .txt files
	The **Play** button will execute the selected setting, e.g. it will ramp up the column, if it is in Standby mode, or if it is already in operation, it will adjust the parameters, as defined in the selected setting.

1.2 Column Control working area

The working area is divided into two subsets:

- **Active Mode** subset which indicates the active operating mode
- **Tree View** subset which displays the available datasets and groups. Individual groups can be opened and closed.

The groups and datasets are freely definable, with the exception of the group **System Ramp-down Modes**.
You can use the functions of the group **System Ramp-down Modes** which have the same name, to switch the column to **Standby** mode or to switch it off.

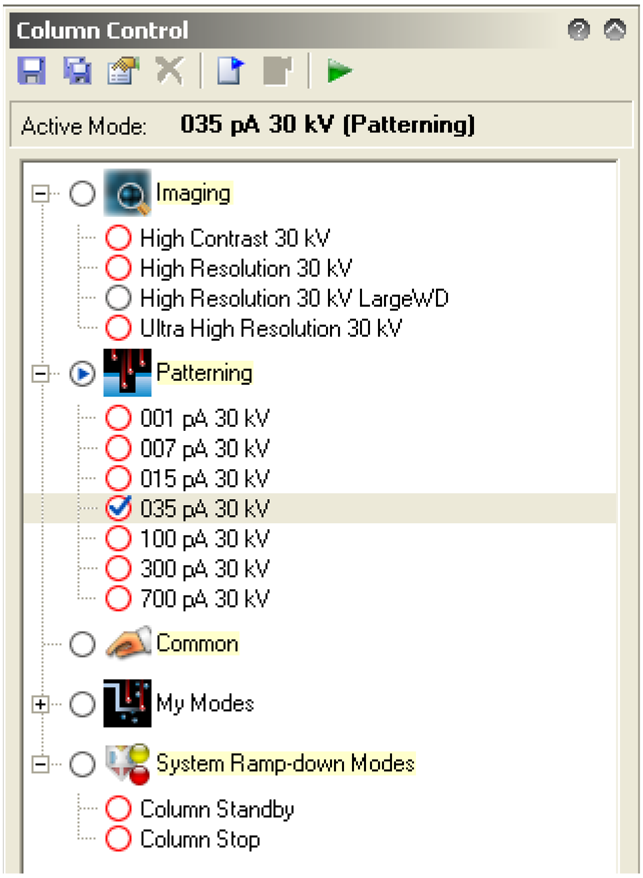


Figure 7-9: Working area of the **Column Control** dialog

There are several groups within the **Column Control** dialog. Each of the groups contains several parameter sets. All groups and modes are indicated by color codes, to reflect their accessibility, by various users. (⇒ *Functional Description* on page 7-3).

Global groups: The yellow background groups in the **Column Control** dialog are the global group settings, accessible to all users. Only the black parameter sets inside the **Global** group can be edited, depending on the users access rights.

Imaging group: All modes associated with **Imaging** present useful tools for imaging.

Patterning group: This group contains all the required parameter sets for beam current settings, with their associated lens settings.

Common group: This group is intended to be used as an exchange forum between users. A user can insert a parameter set into the Common group to share it with his colleagues.

System Ramp down Modes: These are located at the bottom of the **Column Control** window. They are also global settings which can not be edited. They consist only of **Column Standby** and **Column Stop** parameter sets.

This 'Work in progress; sign is displayed after the apertures have been exchanged and the alignment procedure needs to be carried out.

This 'Check' sign is displayed when the aperture alignment procedure has been carried out after the aperture exchange.

1.2.1 System Ramp-down Modes group

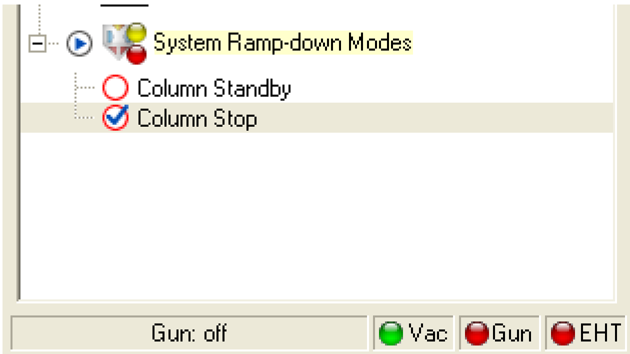


Figure 7-10: Column Stop and Standby

Column Stop: All high voltages are switched off, including the Ga emitter.

Use **Column Stop** if the instrument will be out of operation for several hours, as this will extend the life of the Ga source emitter.

Column Standby: All high voltages are switched off, except the voltage for the Ga source emission. The aperture driver is moved to a position at which there is no aperture, in between two apertures. This will extend the life of the apertures, as they are not exposed to any milling.

Use **Column Standby** when the instrument will not be in operation for a short period of time, of the order of an hour. Another reason for leaving the instrument in **Column Standby** instead of **Column Stop** would be if the application demands highest stability of the Ga emitter. The user may then choose to leave the Ga emitter switched on.

Column Stop: The acceleration voltage and the emitter are switched off. Only use the Column Stop for the Raith EO (thermal field emission) systems, when the system is not in use for more then a week. It is recommended, to only carry out the Column Stop for service work, e.g. to exchange the filament, vent the gun etc.

Column Standby: Only the high voltage will be switched off. It is recommended, to switch off the high voltage whenever the system is not in use. This will protect the apertures from any dirt, as the beam blanker will place the beam onto the aperture holder between the apertures, which can lead to contamination over time.

1.3 Status bar

The status bar is divided into two subsets:

- On the left-hand side, messages regarding current system processes of **Vac**, **Gun** and **EHT** are displayed (**System pumping** and **gun busy**, for example). These message display changes in hardware settings.
- The right-hand side is used to indicate the status of the following

Status bar	Function
Vac	vacuum status
Gun	represents the status of the gun source
EHT	Electrical High Tension. The EHT includes all high voltage settings for the acceleration and optics. For details, please refer to the system manual.

The displays resembling LEDs can have the status green, yellow or red to indicate operating mode.

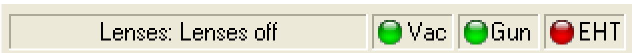


Figure 7-11: **Column Control** status bar

To switch **Gun** or **EHT** on or off manually, or to pump or ventilate the chamber, click on the corresponding command in the context menu.

Table 7-1: Colors of the Vac, Gun and EHT LEDs

Color	Vac	Gun	EHT
Red	No vacuum	Gun switched off	EHT switched off
Yellow	Pump running	Gun being switched on/off	EHT being switched on/off
Green	Vacuum OK	Gun on	EHT on

The green light for **Vacuum** indicates that the vacuum is sufficient to ramp up the column.The green light for **Gun** indicates that the gun settings have been reached.

The red light for **EHT** indicates that all high voltages are switched off, except for the voltage supplied to the gun, which is indicated in the **Gun** LED.

The gun can be switched off if the instrument is not being used for several hours, as this will extend the lifetime of the Ga emitter.

1.3.1 Context - Status bar

Place the cursor over the **Status** bar and click on the right mouse button to display the context menu of the status bar. The context menu displayed depends on the placement of the cursor, over **Vac**, **Gun** or **EHT.**

Figure 7-12: Status bar

NOTICE

Note that when **Vent** is pressed, the Vent will be activated and the chamber will be vented. For sample changeover, the load lock should be used instead.

Status bar context menu items	Function
Vac Pump/Vent	When the instrument is in pumping mode, only the **Vent** option will be available and when the instrument is being vented, only the **Pump** option will be available.
Gun Start/Shutdown	These **Start** and **Shutdown** options will perform the same procedure as **Column Standby** and **Column Stop**. Only one of the options is available at any one time, depending on the instrument status. Selecting **Start** in the Gun context menu will set the instrument to Column Standby. When the instrument is in **Column Standby** and the user selects **Shutdown** in the Gun context menu, the instrument will be transferred to **Column Stop**.
EHT On/Off	For the IONLINE, the **EHT On** button is grayed out, since the EHT can only be activated via the **Column configuration** command. **EHT Off** is a only available when the system is in operation mode. **EHT On** is only available when the gun is on. When the EHT is switched off, the same procedure as **Column Standby** will be carried out.

1.4 Column Control subdialogs

The following subdialogs are available for the **Column Control** tool bar.

The **Arrow** button sets the values from the text field to the column hardware and the **Pipette** reads the current values for all parameters and enters them into the fields. Basically, the arrow button reads from the software and sets and activates the values in the hardware, whereas the pipette reads all current dataset values from the hardware and enters them into the software.

The objective lens is defined by the working distance in mm, whereby the working distance is the distance between the sample surface and the lower edge of the column.
The condenser lens is defined by a high voltage, which will be automatically calculated by the software for the user. The condenser lens voltage is dependent on the acceleration voltage and the aperture size used. Only the Service user is permitted to adjust the condenser lens voltage setting.

You can use the **Fine Tune** dialog to change settings for the column and adopt the current parameter setting by clicking on the **Current Values** button.

1.4.1 Edit Dataset Values

To view or change the values of a parameter set, mark the parameter set and click on the **Edit Dataset Values** button. You can also double click on a parameter set name to open the same dialog.

The definition of a new dataset is based on the standard values of the group. To view or change the values of a dataset, mark the dataset and click on the **Edit Dataset Values** button.

The following dialog is for EO systems only.

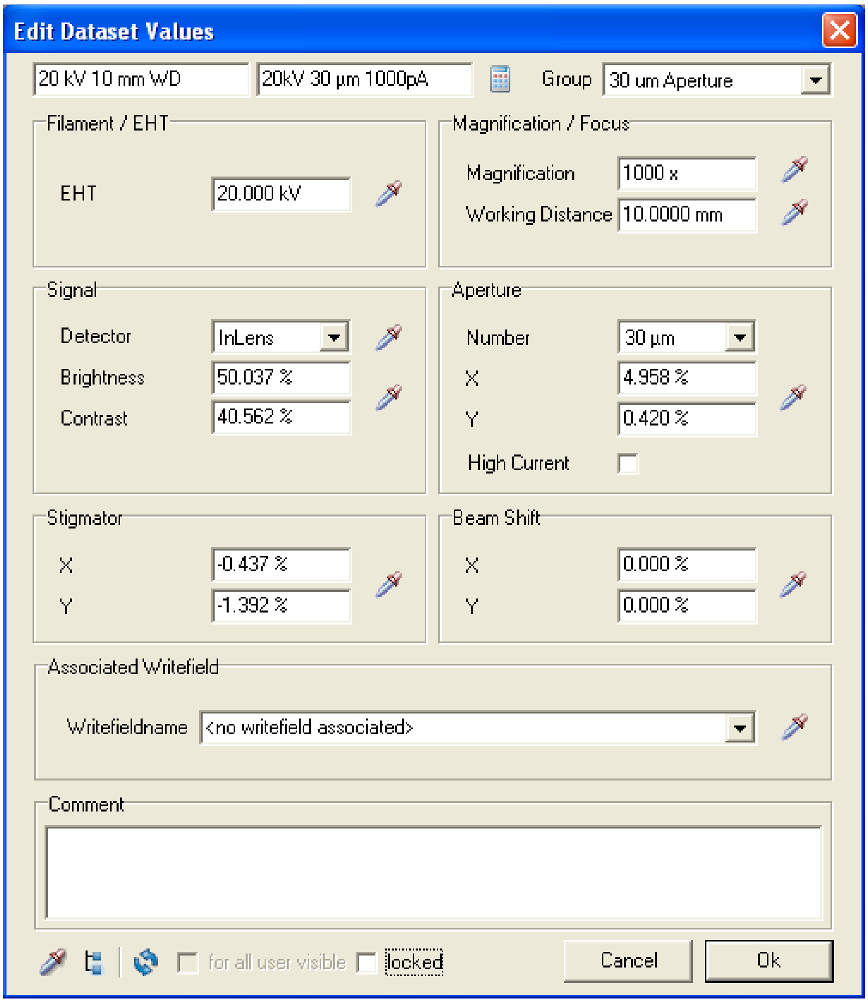


Figure 7-13: **Edit Dataset Values** subdialog for Raith EO

Control element	Function
Name and group	The name of the selected parameter set (EHT and WD) and the group will be displayed and can be edited.
	To create the suffix of the name of the current parameter set. The suffix is the second part of the name. The first part is typed in by the user.
Group	Group selection input field, to which this dataset is assigned. *A dropdown list will display a listing of all existing groups. The user can define the group to which the new parameter set should be allocated.*

Filament / EHT subset:

EHT	The user can enter the EHT value.
	To read in and use the current setting.
Magnification / Focus subset:	
Magnification	The user can enter the magnification setting.
Working Distance	The user can enter the working distance.
Signal subset:	
Detector	To select the detector from the dropdown list
Brightness	To enter the brightness [%]. The current brightness/contrast can also be read in.
Contrast	To enter the contrast [%]
Aperture subset:	
Number	To select an aperture from the dropdown list
X, Y	To enter aperture alignment values for **X, Y** [%].
High Current	Check the checkbox to enable **High Current** for the EO systems.
Stigmator subset:	
X, Y	To enter beam astigmatism correction values for **X, Y** [%]
Beam Shift subset:	
X, Y	The X, Y coordinates are given. The current **Beam Shift** values can also be read in.
Associated Writefield subset:	
Writefieldname	Select a predefined writefield name from the dropdown list (⇒ *Associating Writefields to Column Control parameters for automated patterning* on page 7-18). The current writefield name can also be read in.

Comment	Free text field for comments and remarks. The user can insert comments, such as hints about the application for which this parameter set should be used.
	When the **Pipette** button is clicked, the currently selected parameters are read into the column dataset. The pipette button initiates the reading of all parameters.
	Resets to previous values
	The **Tree** button reads and converts the values from the column control group default parameters.
for all user visible	Check the checkbox if you wish to make the parameters visible to all users. This can only be checked at the SYS or Master level.
Locked	Check the checkbox to lock the parameter set. All parameters will be grayed out and will be inactive.The **Pipette**, **Arrow** and **Tree** buttons will not be displayed.

The following dialog is for IO systems only. Only the parameters which differ from the EO systems will be explained.

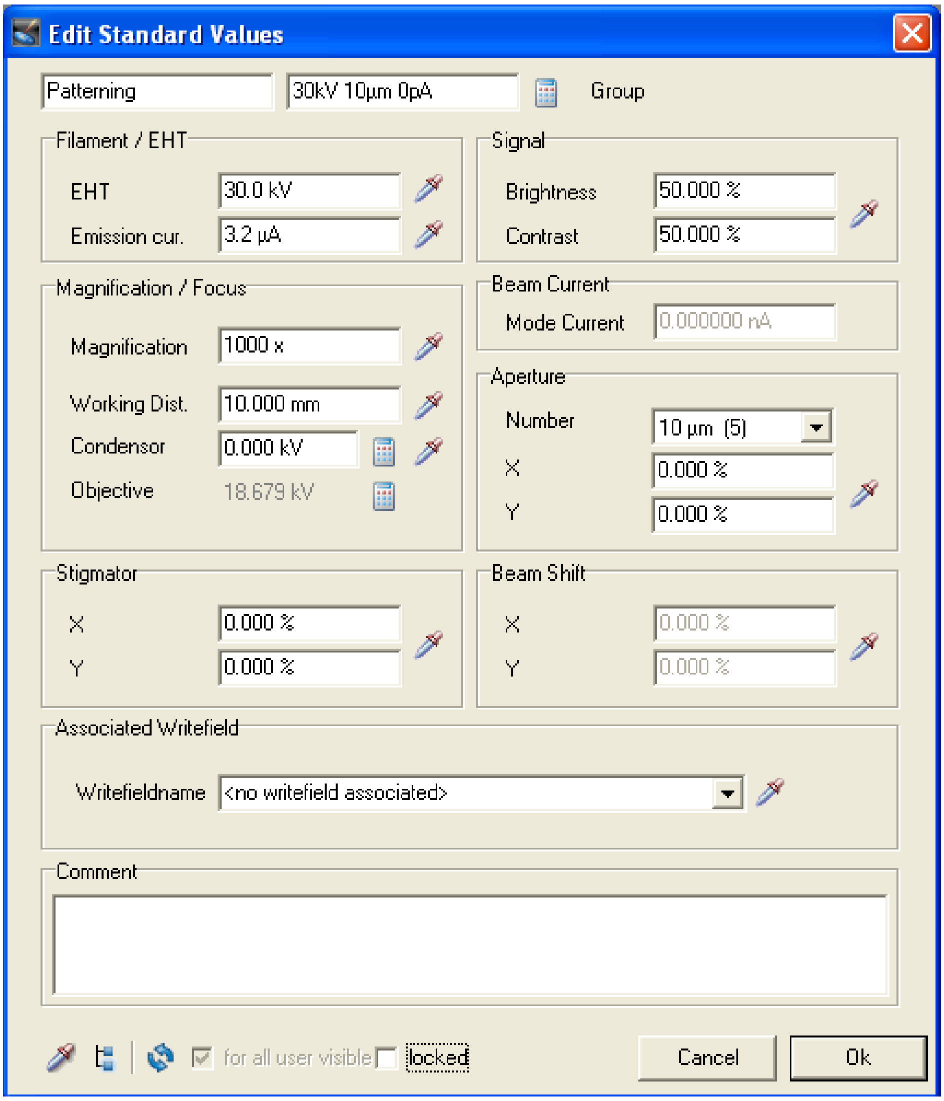


Figure 7-14: **Edit Standard Values** subdialog for IO systems

Control element	Function
Filament / EHT subset:	
Emission cur.	The emission current for the IO system is displayed for information only. The current emission current can also be read in and used.
Magnification / Focus subset:	
Condensor	To set the condensor lens voltage

Control element	Function
Objective	To display the corresponding objective lens voltage

Whenever a beam current measurement is performed, the reading will be saved as the mode setting for the selected parameter set. This mode current value is a good starting point and is therefore used as the default value for the **Beam Current** module as well as for the patterning dialog. The mode is then used for the calculation of the patterning parameters.

Within the IONLINE system, there are three detectors employed. The SE detector and two optical detectors, the CCD camera setting and the macroscope. The macroscope and CCD camera settings are not controlled via the software.
The contrast displayed in the software for TV mode (CCD camera) is set to 50%. This value can not be changed.

You only have to enter the appropriate number values in the text fields. The correct units are completed automatically.

Assigning a writefield to **Column Control** parameter sets is recommended, when automated patterning via a Positionlist will be carried out. Since the selected column control parameter sets can be inserted in the Positionlist via drag and drop, the assigned writefield will be automatically set when the Positionlist is scanned, (⇒ *Drag and drop of Datasets* on page 7-8).
Alternatively, the writefield can be inserted manually into the Positionlist via drag and drop from the **Writefield Manager**.

1.4.2 Edit Standard Values

To view or change the default values of a group, mark the group and click on the **Edit Dataset Values** button. You can also double click on a group name to open the same dialog.

The **Edit Standard Value** dialog is similar to the **Edit Dataset** dialog, except that it is for Group selection. New Groups can be created.

The group datasets can be viewed as master templates for the individual parameter sets within that group. The group parameters can be edited in the **Edit Standard Value** dialog.

You only need to enter the appropriate number values in the text fields. The correct units are completed automatically.

A group standard could be either a particular voltage or aperture. For example, to create a group where all 30 kV settings are stored, enter 30 kV for the group name. Alternatively a group can be created for a particular aperture.

It is possible to select a batch of settings and set the batch to the group standard. This would set all settings in this group to the selected group standard, for example to a particular selected voltage or aperture.

The following fields are also available:

Control element	Function
Group	A new group can be created.

1.5 Context menu - Group

Via the right mouse click, the context menu becomes available. The context menu will change according to the cursor position in the **Column Control** dialog, when the right mouse button is clicked.

Placing the mouse on the group name, a right mouse click will display the group context.

Group context menu item	Function
New dataset	This will create a new dataset in **Column Control**. A new dialog, **Edit Dataset Values** will open.
New group	A similar dialog to **Edit Standard Values** will open. A new text field will be overlaid, in which the group name can be entered.
Edit	A new dialog, **Edit Standard Values** will open, in which the parameters can be edited. This command is also available via the **Column Control** tool bar, (⇒ *Column Control Tool bar* on page 7-20)

Group context menu item	Function
Delete	To delete the group
Rename	To rename a group
Image	An image can be inserted to characterize a group. The user can select an image which is used as a symbol to represent the group.
Import Dataset List	This will open a new dialog, in which a list of all available datasets will be displayed. The file format of the datasets is a txt file.

1.6 Context menu - parameter set

Placing the mouse on a parameter set inside a group, a right mouse click will display the parameter set context.

Parameter set context menu item	Function
New dataset	This will create a new dataset in Column mode. A new dialog, **Edit Dataset Values** will open.
New group	A similar dialog to **Edit Standard Values** will open. A new text field will be overlaid, in which the group name can be entered.
Edit	A new dialog, **Edit Dataset Values** will open, in which the parameters can be edited. This command is also available via the **Column Control** tool bar, (⇒ *Column Control Tool bar* on page 7-20)
Delete	To delete the dataset
Save	To save a dataset
Save as	To save the dataset under a new name.
Rename	To rename a dataset. The text field of the name is activated and the user can enter a new name.

Parameter set context menu item	Function
Export Dataset	This will open a new dialog, in which the user can determine the folder to which the dataset should be exported. The file format of the datasets is a txt file. This command is also available via the **Column Control** tool bar, (⇒ *Column Control Tool bar* on page 7-20).
Activate Last Configuration	When a setting has been modified by the user, but the settings have not been saved, the user can revert to the last configuration, even if it had not been saved.
Activate Selected Mode	To activate the selected parameter set. It has the same function as the **Play** button. This command is also available via the **Column Control** tool bar, (⇒ *Column Control Tool bar* on page 7-20).

The command **Activate Last Configuration** is a very useful tool. For example, if the user has put the column into Standby mode over a lunch break and wishes to activate the last configuration, which had not been saved, clicking on **Activate Last Configuration** will reset the values to the last set values even if they have not been saved. This last parameter set is stored in the software, even if the software has been exited, for example when the instrument is switched to Shutdown mode. The last dataset will not be listed when the software is re-started, if it has not been saved, but it still can be recovered by the **Activate Last Configuration** button.

2 Fine Tune dialog

You can use the **Fine Tune** dialog to set the column parameters during operation.

There are two ways of entering values in this dialog:

- Using text fields: The values are only adopted when you click on the **Send** button.
- The slider bars or the control marker can be adjusted by dragging the cursor to the new position. The further the cursor is moved, the more the value will be changed.
- By scrolling with the mouse wheel, the sensitivity range of the slider bar is changed, from coarse to medium to fine. These changes are adopted immediately.

The **Fine Tune** dialog displays **Magnification**, **Focus**, **Detector**, **Aperture** and **Stigmator**.

The functional principle of the software tools within the **Fine Tune** window, such as the one-dimensional slider bar and the two-dimensional fields are explained in the **Functional Principle**, (⇒ *Software tools within Fine Tune dialog* on page 7-5).

You only need to enter the appropriate number values in the text fields. The correct units are completed automatically.

In addition to the tuning dialogs explained below, there is also a **Toolbox** for the main **Column Control** parameters available, whenever an image is opened. This toolbox enables very quick access to the tuning parameters, to adapt the tuning between focus and stigmator for example. This can be performed faster using the Column Control toolbox in the Image tool box, than by selecting the individual tuning dialogs in the **Fine Tune** dialog. The toolbox is explained in great detail in the chapter Linescans and Imagescans, (⇒ *Column subset* on page 10-36).

2.1 Magnification and Focus

You can read and set the high voltage, magnification and working distance in the **Mag / Foc** tab.

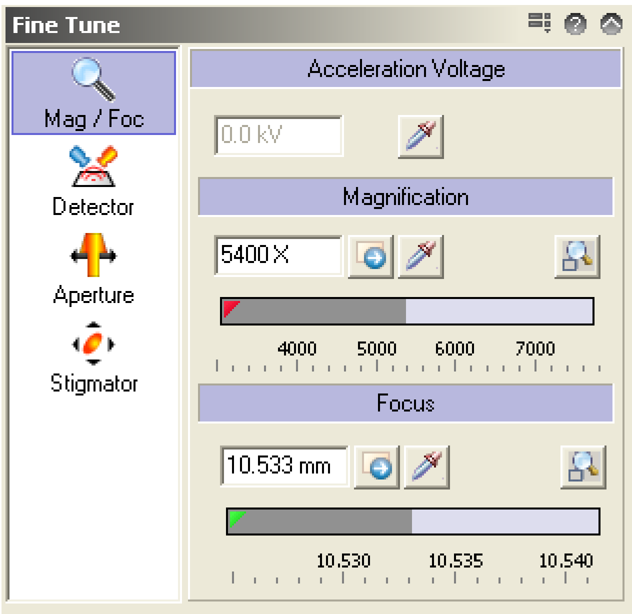


Figure 7-15: **Fine Tune** dialog – **Mag / Foc**

Control element	Function
Acceleration Voltage	The acceleration voltage is displayed for information only [kV]. The current setting can be read in via the **Pipette** button.

Control element	Function
Magnification	To set the magnification [x].
Focus	To set the working distance [mm].
	Click on this button to send the current value to the column.
	Click on the Pipette button to read the current value from the column.
	Clicking this button will enable the full range after using the mouse wheel.

2.2 Detector

Select the detector in the **Detector** tab and choose the settings for **Contrast** and **Brightness**.

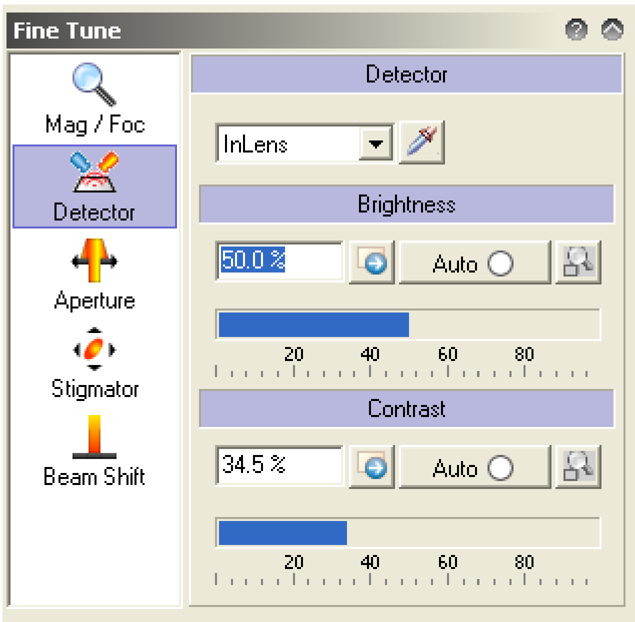


Figure 7-16: **Fine Tune** dialog – **Detector** tab

Control element	Function
Detector	To select the detector from the list: – **SE** – **InLens** – **TV** (Chamberscope) – **Macroscope**. The Macroscope will show an optical macroscope image. This window will close when the SE detector is selected.
	To read in the currently active detector

When an image is continuously scanned, and the user switches to TV mode, the software will pause the scan and the video window will be opened.
Since a scan is always performed with an electron or ion beam and the SE or InLens detector, the scan is interrupted while the user is viewing the chamber with the TV or macroscope. As soon as the SE detector is selected again, the continuous scan will be continued.

Control element	Function
Brightness	To enter the brightness [%]
Contrast	To enter the contrast [%]
	To send the current value to the column
	To autoselect the brightness and contrast
	To enable the full value range after using the mouse wheel. The coarse (red triangle) setting will be used.

The sensitivity ranges for **Brightness** and **Contrast** are inter-related, when one of them is adjusted, it will also change the other parameter range. The parameters Brightness and Contrast are therefore both either in **Fine**, **Medium** or **Coarse** range at any one time.

2.2.1 Signal Mixer (Raith EO only)

The **Signal Mixer** enables the mixing of the signals from two different detectors and can be installed on the Raith EO as an option.

For the **Signal Mixer**, only select the InLens and SE detector. IT is not possible to select the TV detector for signal mixing.

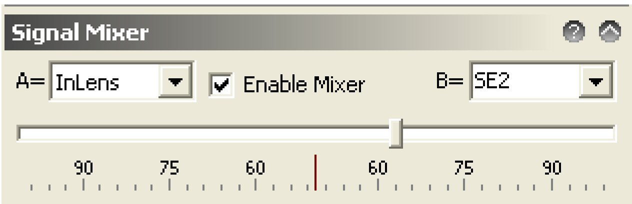


Figure 7-17: Signal mixer dialog

Control element	Function
A	Select one of the detectors from the dropdown list for signal A.
Enable Mixer	Check the checkbox to enable the mixing of both detectors.
B	Select another detector from the dropdown list for signal B.
slider bar	The sliderbar enables the user to vary the signal either for signal A or B between 50-100%. For example, setting signal A to 80% will automatically set signal B to 20%. The sum of both signals will always amount to 100% total signal. Moving the sliderbar from the center to the left hand side will increase the signal from detector A and decrease the signal from detector B. Moving the sliderbar from the center to the right hand side will increase signal B and decrease signal A.

The mixing of the InLens and SE detector is particular useful in obtaining the optimum image. The InLens detector, for example, has often less background noise, but does not display the material contrast as well as the SE detector. Contrary, the SE detector displays good material contrast but more background noise compared to the InLens detector.

Mixing both detector signals can therefore improve the image quality. Optimize for maximum contrast while keeping the background noise as low as possible.

2.3 Aperture

Select the aperture diameter in the **Aperture** tab and set the correction values for the penetration point of the beam.

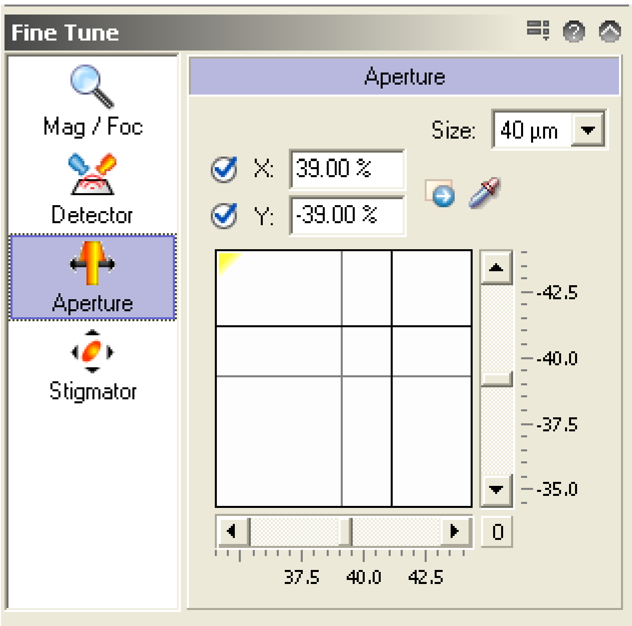


Figure 7-18: **Fine Tune** dialog – **Aperture** tab

Control element	Function
Size	To select the aperture diameter [µm] from the dropdown list.
X, Y	To enter Aperture Alignment correction values for **X, Y** [%] or to make the setting using a slider.

It is recommended to change the aperture only via a column control parameter set, since a change of aperture also requires a change of lens settings.

The **Aperture** is tuned on a real image, using focus wobbling focus wobbling. The aim is to fine tune the aperture so that it is situated in the center of the beam. The aperture is moved in X and Y directions to place it into the center of the beam. When optimized, the image should not shift during focus wobbling. The image will become sharp and less sharp during focus wobbling, but should not shift in X or Y direction.

It is recommended to perform **Aperture Fine Alignment** with a reduced scan, as the refresh rate is faster and therefore it is easier for the user to monitor changes during the tuning procedure.

2.4 Stigmator

You can set the values for correcting the beam astigmatism in the **Stigmator** tab.

The **Stigmator** can be tuned on a real image using focus wobbling. Whilst focus wobbling is on, the same effect should be observed in X and Y axes. The distortion caused by focus wobbling should be symmetrical.
Some users prefer to monitor the stigmator via an imagescan, to check if e.g. a spot is symmetrical, round and sharp.

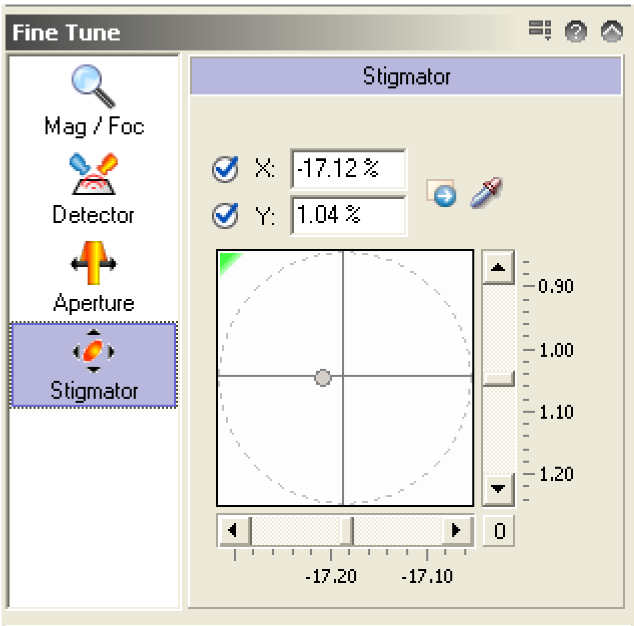


Figure 7-19: **Fine Tune** dialog – **Stigmator** tab

Control element	Function
X, Y	To enter Stigmator Alignment correction values for **X, Y** [%] or to make the setting using a slider.

2.5 Fine Tune file menu

The **Fine Tune** menu will open the following menu options:

Clicking on the **menu** button the **Fine Tune** menu is displayed.

Fine Tune menu items	Function
Power Supply Monitor	The **Power Supply Monitor** displays the status of the important power supplies for the column.
System Status Report	This is an overview monitor window. A new dialog opens.
Safe Controller Sensitivity	It will save the currently selected sensitivity ranges for all control elements as their new default values.

2.5.1 Power Supply Monitor subdialog (Raith IO only)

Selecting the **Power Supply Monitor** will open the following dialog, the access is only permitted for Master level users.

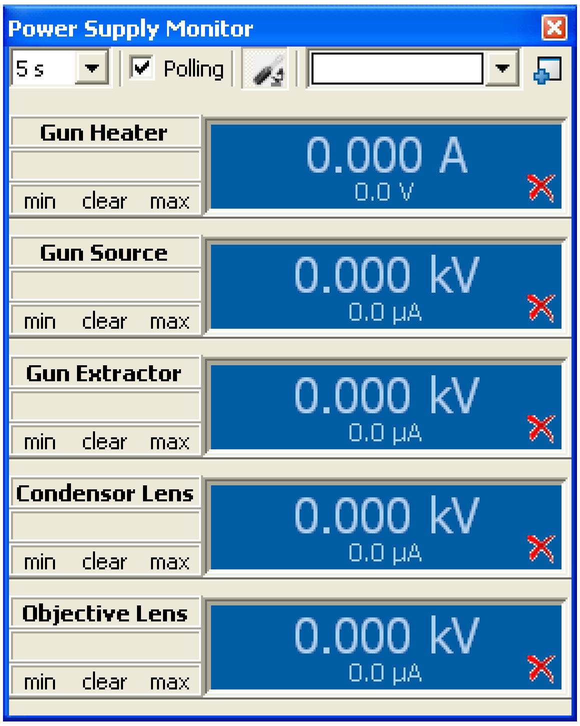


Figure 7-20: **Power Supply Monitor** dialog

Control element	Function
Timescale	Select the timescale for the measurement intervals from the dropdown list. It can be selected in the range of 1 - 60 s.

Control element	Function
Polling	When **Polling** is checked, the supplies are now selectable for polling. The supplies displayed in the monitor will be measured automatically at a set interval, e.g. every second.
	To take a measurement directly
Power Supply Monitor drop-down list	A dropdown list will display a list of all available power supplies to be selected for monitoring. The **Condenser Lens**, **Objective Lens**, **Gun Source** and **Gun Extractor** are typically the principal ones to be monitored.
	Via the button **Add to monitor**, other supplies can be added, for example, the gun heater. Up to five supplies can be monitored at any one time. Once the **Add to monitor** is pressed, the selected supply will be displayed. The current value as well as the minimum/maximum values will be displayed.
Display setting	The monitor display shows a blue field with a large number and a smaller number situated underneath. The more important value is the larger one, e.g. either the current or voltage setting for the display is chosen as the large letter. For each parameter, the default configuration is set. Placing the mouse cursor on the display field and clicking with the right mouse button enables the user to change the **Display Setting**. The priority between the large and the small display can be toggled via **Switch U/I** and the units for the display can be selected. The software will automatically calculate the value in the selected units. For example, the gun meter has a large **Ampere** displayed and a small **Voltage**.
	The red **Delete** button after each displayed parameter will delete this parameter from the monitoring list. Please note that it will be only deleted from the monitoring. This button is the opposite to **Add to monitor**, it simply means **Delete from monitor**.

The supplies that have not been selected for monitoring will not be measured. Only if the system status is changed will the software then automatically check and set the values to all required supplies.

It is recommended only to monitor the most important supplies. If the software has to communicate with all hardware supplies for a frequent update, the system might slow down, as large amount of data needs to be communicated between hardware and software. If the system is slowed down, it might affect the patterning process.

The polling, in particular, requires a lot of data transfer. As soon as the **Polling** window is closed, even if the checkmark is still present next to **Main polling**, polling will be stopped.

The power supply monitor window should only be switched on if the user needs to monitor the supplies, for example if any problems are suspected. As all supplies are checked and indicated during the initialization process, the Power Supply Monitor window should be used as an additional monitor resource, when required.

Power Supply Monitor context menu items	Function
Switch U/I	This enables the user to toggle between voltage and current as the main display parameter.
Toggle units for U/I	The user can toggle between the units, e.g. V to kV or A to mA, nA and µA.

2.5.2 System Status report (Raith IO only)

A new dialog will open, giving an overview of the monitor window. The checkmarks indicate when the components have the status **On** and the connectivity is functioning.

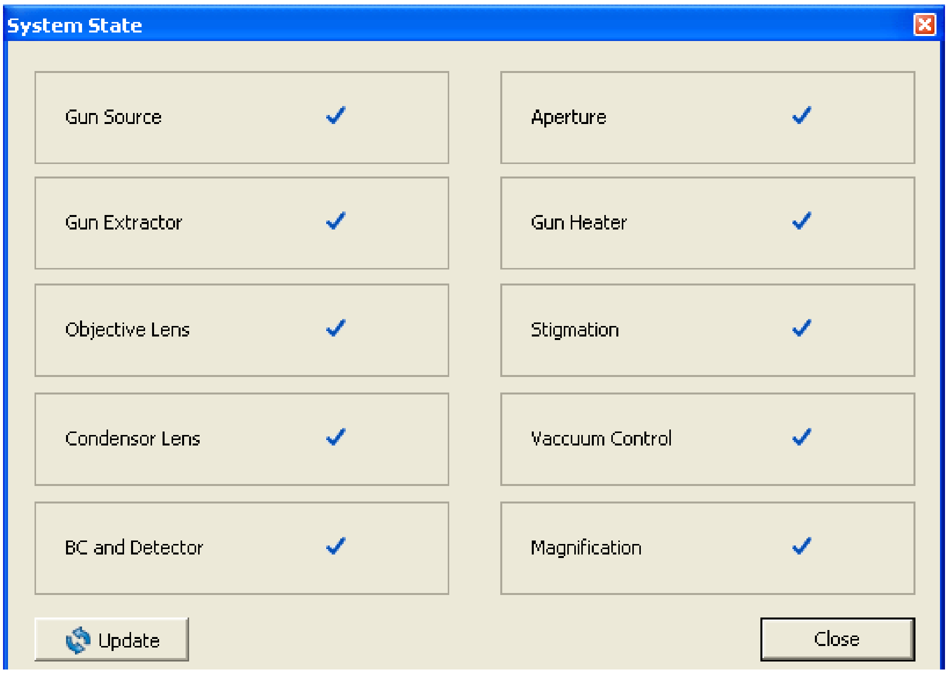


Figure 7-21: **System State** dialog

Clicking on the **Update** button will check the connectivity, for example ensuring that all supplies are plugged in. This can be used as a coarse diagnostic tool by the user.

2.5.3 Save Controller sensitivity

This will not open another dialog. It will save the currently selected sensitivity ranges for all control elements such as Brightness, Contrast, Working Distance, Stigmator, Aperture etc to their new default values.

Generally, the following sensitivity ranges are recommended:
Magnification: Coarse sensitivity range
Brightness/Contrast: Medium sensitivity range
Aperture/Stigmator: Fine sensitivity range
Focus: Fine sensitivity range.

---

3 Focus Wobbling dialog

You can use the **Focus Wobbling** dialog to switch focus wobbling on and off with selectable amplitude.

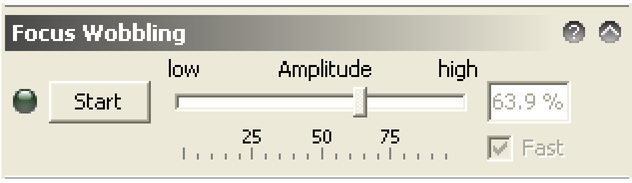


Figure 7-22: **Focus wobbling** dialog

Control element	Function
Start/Stop button	Starts and stops focus wobbling with the set amplitude.
Wobble amplitude	The wobble amplitude in mm can be selected via the sliderbar, or entered as %.
Fast	Checking the checkbox for **Fast** wobble period will increase the wobble frequency.
LED top/bottom	When focus wobbling is not selected, both LEDs are off. When focus wobbling is initiated, the first LED is green to indicate that focus wobbling is active.
	The bottom LED is blinking. The faster the focus wobbling speed, the faster the blinking.

Focus wobbling creates a sinusoidal form of the alternating voltage. During this change, the bottom LED will change its status between gray and red, depending if the voltage range is in the positive or negative range. The focus wobbling voltage is varied around the focus setting on the objective lens. The blinking LED speed therefore indicates the focus wobbling period.

The recommended settings are between 0.2 and 0.5 mm for the amplitude.

4 Vacuum Levels dialog (Raith IO only)

The **Vacuum Levels** dialog includes blue data fields, in which all vacuum readings are displayed, such as **Chamber vacuum**, **Column vacuum** and **Load lock vacuum**. In addition, the status of the valve between the column and the chamber, the **CCV (Column Chamber valve)**, is given. The valve can be opened and closed via the **CCV open** button.

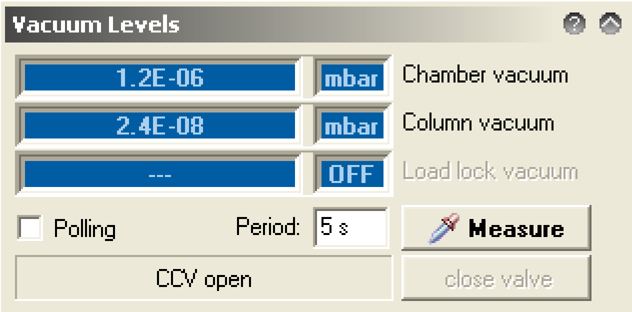


Figure 7-23: **Vacuum Levels** dialog

Control element	Function
Chamber vacuum	The vacuum in the chamber is displayed.
Column vacuum	The vacuum in the column will be displayed.
Load lock vacuum	The vacuum in the load lock will be displayed.
Polling	To enable polling, which will measure the vacuum in the **Period** specified.
Period	Enter the time period, after which the vacuum readings will be obtained again, e.g. every 5 seconds.
	Click on the **Measure** button to obtain the current reading of the vacuum levels.
CCV open/CCV closed	Click on the **CCV open/CCV closed** to toggle the status of the Column Chamber Valve.The valve can be opened and closed via this button. The button description will be changed accordingly to the current status.

The **CCV** valve is normally open under operating conditions. It will be closed during sample changeover via the load lock to maintain the high vacuum for the ion emission source on the IONLINE system while the load lock is operated.

The **Polling** option is a very useful feature when working with the GIS option to check the required vacuum pressure regularly during the GIS procedure.

The overall vacuum can also be check via the **Vacuum** LEDs in the **Column Control**, (⇒ *Status bar* on page 7-24). When the vacuum LED is green, the vacuum is ok.

Chapter 8
Stage Control

This chapter gives you an overview of how to control the stage using the **Stage Control** function.

The following topics and tasks are included:

- Background information on the stage axes
(⇒ *Stage axes overview* on page 8-4),
- Listing of available drive commands to define a stage movement and manually drive the stage over the defined path
(⇒ *Description of Drive commands for stage movement* on page 8-5),
- Background information on Eucentric drive mode (⇒ *Eucentric drive mode* on page 8-8),
- Functional description of so called W-limits monitoring (⇒ *W-limits* on page 8-10),
- Introduction of a new method of backlash compensation (⇒ *Anti Creep extension for auto backlash* on page 8-13).
- Explanation of the editing of a position in the Stage Control dialog
(⇒ *Editing a position within Stage Control* on page 8-14),
- Detailed description of the Coordinates dialog and display options
(⇒ *Coordinates dialog* on page 8-16),
- Description of the four tabs - Drive, Step, Positions and Stage Lock - within the Stage Control dialog
(⇒ *Stage Control dialog* on page 8-18),
- When to use the Find Home Position dialog
(⇒ *Find Home Position* on page 8-23),
- Explanation of the Laser Stage Control dialog
(⇒ *Laser Stage Control* on page 8-25),
- Explanation of the Touching Alarm dialog
(⇒ *Touching Alarm* on page 8-26).

You can call up **Stage Control** by clicking the (Stage Control) icon in the control bar.

- **Coordinate display** shows the current coordinates.
- **Stage Control** dialog is made up of the following four tabs:
  - **Drive** tab to drive to relative or absolute positions in XY and UV coordinates
  - **Step** tab to set the step size of the stage movement
  - **Positions** tab to drive to pre-defined positions or to edit and save new positions
  - **Stage Lock** enabling the locking of any stage axis
- **Find Home Position** dialog carries out the stage coordinate initialization.
- **Laser Stage Control** dialog enables the change or verification of the current stage mode.
- **Touching Alarm** dialog can be either armed or disarmed.

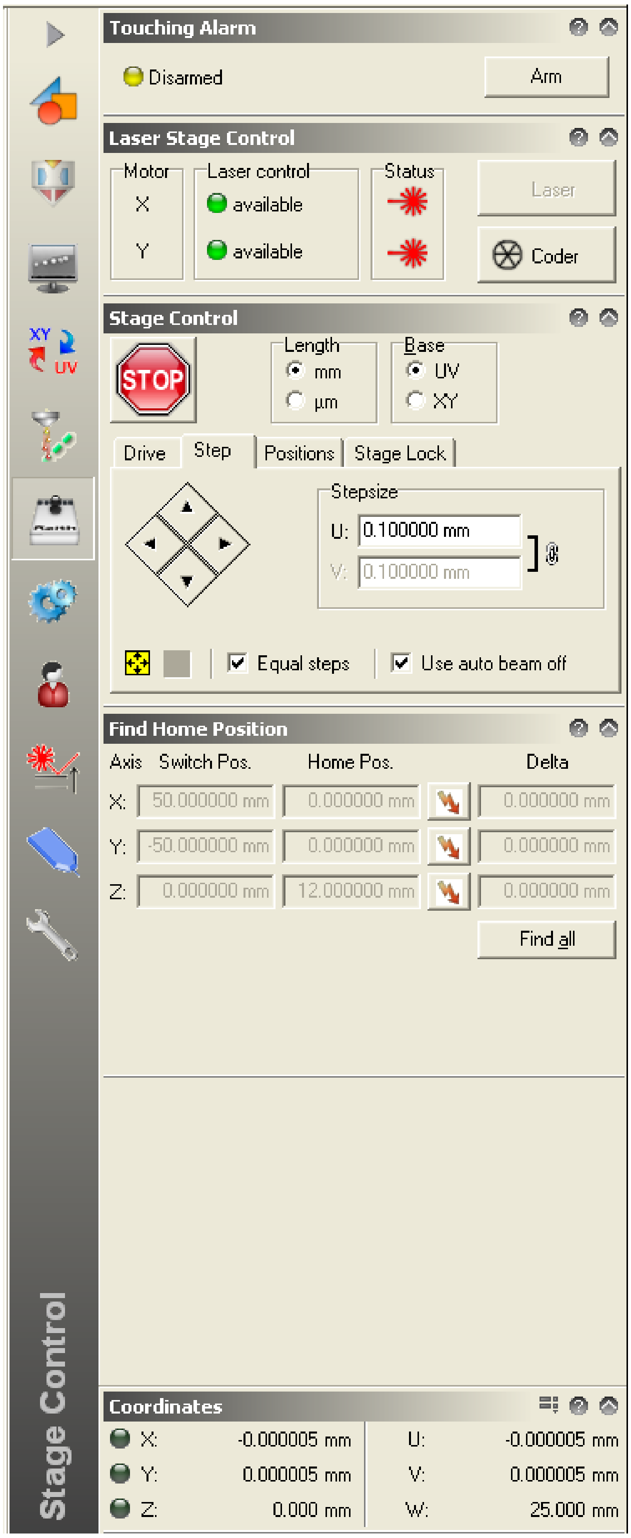


Figure 8-1: **Stage Control**

A Functional Description

1 Stage axes overview

Normally, the system is equipped with a stage to position the sample. If this stage is motorized then it is possible to control the stage movement by means of the software. The axes that are available depend on which stage is installed. In the following, all supported axes are listed.

Axes	Function
X	One of the physical axes of the stage. This axis is normally oriented perpendicular to the beam.
Y	One of the physical axes of the stage. This axis is normally oriented perpendicular to the beam and perpendicular to X.
Z	One of the physical axes of the stage. This axis is normally parallel to beam.
R	If available, this is the rotational axis of the system.
T	If available, this is the tilt axis of the system.
U	One of the axes of the sample coordinate system. This axis is in the plane of the sample surface
V	One of the axes of the sample coordinate system. This axis is in the plane of the sample surface and perpendicular to U.
W	One of the axes of the sample coordinate system. This axis is perpendicular to the sample surface and, if corrected for the sample height, is equal to the column working distance.

2 Description of Drive commands for stage movement

The **Drive** tab allows control of stage movement, by sending single commands to the motor controller. The following digital commands can be sent via the **Command line**.

Syntax	Function
<coordinate><lowercase-axis-letter>	absolute addressing
<distance><uppercase-axis-letter>	relative addressing
<axis#>b	backlash compensation
<Space>b	backlash on all axes
<line#>g	absolute Positionlist line number
<line#>G	relative Positionlist line number
i<id#>g	absolute Positionlist ID
i<id#>G	relative Positionlist ID
e<coordinate> <lowercase-axis-letter>	absolute eucentric movement
e<distance> <uppercase-axis-letter>	relative eucentric movement

The functionalities of these commands are as follows:

Syntax	Function
Absolute addressing	To move one axis to a requested position in stage coordinates or sample coordinates. Syntax **<coordinate><lowercase-axis-letter>** is used, where the positive or negative <coordinate> value must be entered in mm. Available **<lowercase-axis-letter>** values are **x**, **y**, **z**, **r**, **t**, **u**, **v**, **w**. To move to **0,0**, simply type the letter of the axis.

Example:
The command **-5.5x** moves the stage to the X coordinate -5.5 mm.

Syntax	Function
Relative addressing	To move one axis relative to its current position over a requested distance in stage or sample coordinates. Syntax **<distance><uppercase-axis-letter>** is used, where the positive or negative <distance> value must be entered in mm. Available **<uppercase-axis-letter>** values are **X**, **Y**, **Z**, **R**, **T**, **U**, **V**, **W**.

Example:
The command **2.5X** moves the stage over 2.5 mm in the positive X direction.

Syntax	Function
Backlash compensation	Backlash compensation is essential for exact positioning with non-Laser stages. It is done by moving the stage away from the final position and returning to it always from the same direction, normally against the spring force. The syntax is **<axis#>b**, where <axis#> is **1** for the X axis, **2** for the Y axis, **3** for the Z axis, **4** for the R axis, **5** for the T axis. Use **Space** to perform a backlash on all axes.

Example:
The command **2b** moves the Y motor for backlash compensation.

Syntax	Function
Positionlist addressing	Within the current open Positionlist, any position can be addressed by data line number or by data line identification number (**ID**). Each of these commands can be specified to be absolute or relative to the currently selected data line. The syntax for absolute Positionlist data line number addressing is **<line No.>g**, where <line No.> is the number of the corresponding line.

Example:
The command **15g** selects data line with number 15 and moves the stage to the position specified in that data line.

The syntax for relative Positionlist addressing is **<line No.>G**, where the <line No.> value has to be entered as the difference between the number of the currently selected line and the number of the line to be selected.

Example:
If the data line with number 10 is currently selected, the command **15G** selects data line with number 25 (=10+15) and moves the stage to the position specified in that data line.

The syntax for absolute Positionlist ID addressing is **i<line ID>g**, where <line ID> is the ID of corresponding line.

Example:
The command **i15g** selects the data line with ID 15 and moves the stage to the position specified in that data line.

The syntax of relative Positionlist ID addressing is **i<line ID>G**, where the <line ID> value must be entered as the difference between the ID of the currently selected data line and the ID of the data line to be selected.

Example:
If the data line with ID 10 is currently selected, the command **i15G** selects data line with ID 25 (=10+15) and moves the stage to the position specified in that data line.

Syntax	Function
Eucentric drive mode	When you launch an eucentric drive command, the stage will move in a special predefined sequence of single axis movements. Eucentric drive mode suppotes the drive commands for R and T axes, only. Drive command can be specified to be absolute or relative to the current position and are represented by a prefixed **e**.

Example:
The command "e30r" moves the stage rotation-eucentrically to the R coordinate 30°.

3 Eucentric drive mode

During sample tilt or rotation with a Cartesian stage, the previous observed position of the sample moves away from the beam position in horizontal and vertical direction. This leads to a different UVW-sample position of the beam after the movement.

When activated, the eucentric drive mode moves the three Cartesian axes X, Y and Z, to compensate the sample displacement, due to the R and T movement. This will result in “fixed” UVW-sample coordinates and is often termed as “Compucentric“ movement.

In addition, the eucentric drive mode initiates a Z-axis correction for a eucentrical UV-movement in a tilted sample plane to hold the W position.

The position correction of the Z-axis is related to a tolerance, which checks the amount of the vertical correction displacement. If the vertical position shift is below this tolerance, the Z axis will not be driven for working distance correction. The tolerance is defined in the **Module Status > Virtual Motor Control > Compucentric tilt WD tolerance**.

A eucentric movement can be addressed via the command window or within the position list.

The mode is limited to tilt angles up to 85°. Above this angle no eucentrical or UV movement is possible.

Eucentric UV movement and automatic **Working Distance Correction** interfere when activated at the same time.

When you launch a eucentric drive command, the stage will move in a special predefined sequence of single axis movements.

The axis order in this sequence depends on your starting and target position and is optimized to decrease the risk of collision during the movement. Table below summarizes which axes are driven during a specific movement.

Drive template	Stage axes involved	Comment
eUV	X, Y, Z	Z only for ΔW > **WD tilt Tolerance**.
eT	X or Y, Z	Z only for ΔW > **WD tilt Tolerance**, X if Y is parallel to tilt axis and Y if X is parallel to tilt axis.
eR	X, Y, Z	Z only for ΔW > **WD tilt Tolerance**.

It is possible to activate an information window which lists the intended drive sequence before the first movement is performed. The sequence starts with the axis shown on top of the drive table.

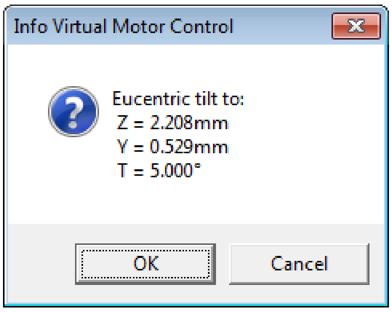


Figure 8-2: Eucentric tilt drive table

The drive table can be enabled in the Module Status settings, go to **Module Status > Virtual Motor Control > Compucentric tilt drive table**.

Tilt and rotation axis must be calibrated during the final system integration of RT module. For this reason, Raith software provides a service dialog, which allows you to perform several calibration procedures described in **Chapter 19 Service**.

4 W-limits

The **W-limit** function defines two W-axis warning zones, shown in ⇒ Figure 8-3. The upper zone is given by the absolute W position **Critical W**. All W positions, which are smaller than or equal to **Critical W** belong to this zone. The lower zone is entered first and defined as **W Warning range** before **Critical W**. Its lower boundary is reached at **Critical W** + **W Warning range**.

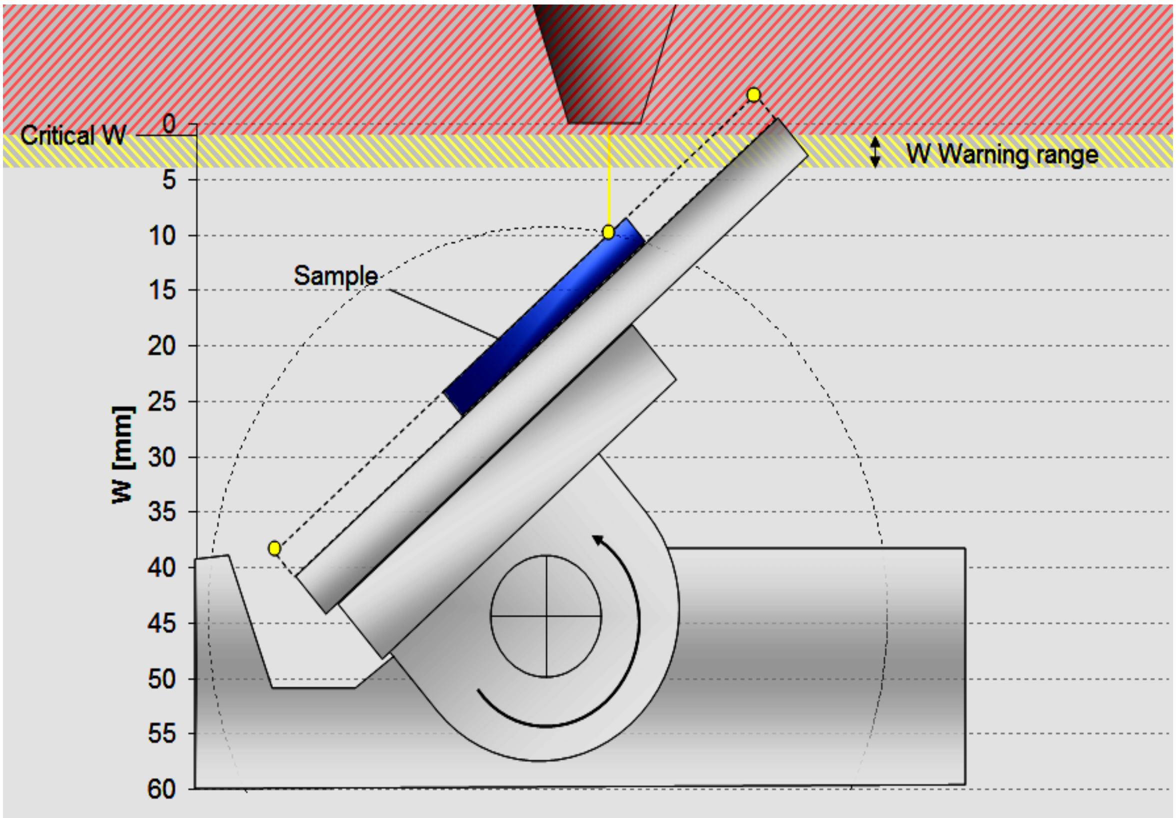


Figure 8-3: W-limits zones

Three points of the sample and sample holder are monitored during operation. They are marked as yellow dots in ⇒ Figure 8-3. The first one is given by the current sample position. The other points are defined by the outermost extensions of the sample holder or tilt arm projected in the plane of the sample surface.

The **W-limit** function can be enabled or disabled by sys-level users under:
Module Status > Virtual Motor Control > Working distance limits.

The **Critical W** value can be edited under:
Module Status > Virtual Motor Control > Critical W.

The **W Warning range** can be defined under:
Module Status > Virtual Motor Control > W Warning range.

When enabled, the **W-limit** function checks before each digital drive command, if the target position will be within a zone. In this case it generates an Info message and a Warning flag; as shown in ⇒ Figure 8-4 and ⇒ Figure 8-5.

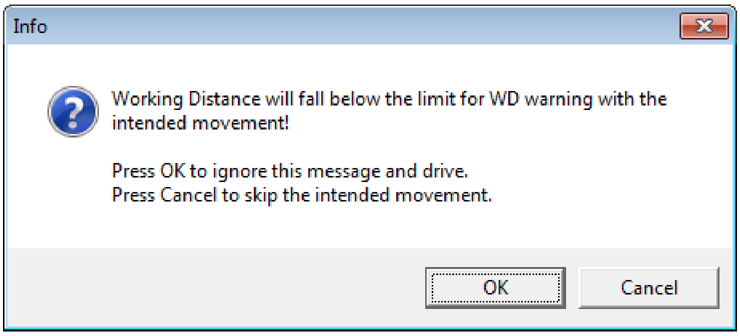


Figure 8-4: W-Limit info before movement: **W Warning range**

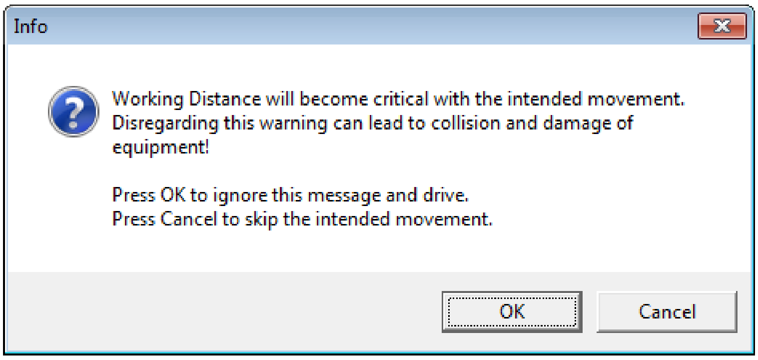


Figure 8-5: W-Limit info before movement: **Critical W**

If the intended movement consists of several single axis movements, each axis will be checked individually and can cause a message.

During the stage movement a working distance warning message will appear in the moment a zone is entered.

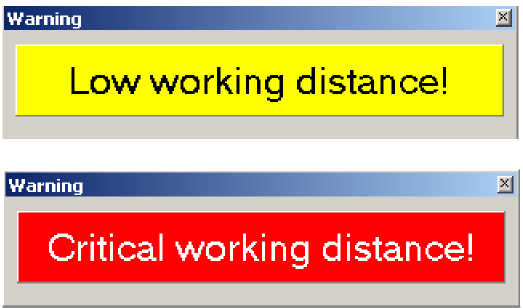


Figure 8-6: W-Limit zone entering Warning windows

NOTICE

The w-limit monitoring is limited to tilt angles up to 86°. Greater angles will be processed as an 86° tilt angle.

NOTICE

All points monitored by the **W-limits** function are related to the current sample height. Therefore a proper W adjustment is essential for proper operation.

A working distance warning does not strictly indicate, that the intended movement will lead to a collision: E.g. it is hardware depending if, there is still enough horizontal clearance before the sample holder is hitting the pole pieces, as it is shown in ⇒ Figure 8-3 on page 8-10.

Understand the working distance warning as indication to act very carefully, e.g. use smaller movement steps and monitor the movement with the TV chamber camera.

5 Anti Creep extension for auto backlash

Some tilt mechanics suffer under backlash and a strong position drift directly after axis movement. The direction of this special kind of drift is always inverted to the former movement direction.

The Anti Creep function is an extension to the Auto Backlash function for tilt axes and addresses this drift issue by performing a single compensation movement of the motor to remove mechanical tension.

After the general **auto backlash** routine, the **Anti Creep** extension adds a small overshoot to the intended target position for the motor and corrects the axis position afterwards. Thus the last movement of a drive command is always inverted to the main movement direction while the target position is reached within the same accuracy.

As the **Anti Creep** function is an extension of the **backlash compensation** routine, it is only available, if the auto backlash routine is activated for the tilt axis.

The Anti Creep extension can be configured under:
Module Status > Virtual Motor Control > Anti creep amplitude.

It is disabled if **Anti creep amplitude = 0** and enabled for **Anti creep amplitude > 0**.

NOTICE

In general, only very small **Anti creep amplitudes** are effective. Do not increase the **Anti creep amplitude** yourself without consultation Raith. The function causes a temporary overshoot of the target position and can increase the risk of collision, when used improperly.

B Tasks

1 Editing a position within Stage Control

To edit a position, proceed as follows:

- Select the position to be edited from the dropdown list box.
- Choose the **Edit** button. The **Edit User Defined Position** dialog box opens.
- Enter a suitable name for this position.
- Enter the position coordinates into the text boxes **X**, **Y**, **Z**, **R**, **T** or choose the **Read** button to adopt the coordinates of the current stage position.
- Enter the sequence of axis movements into the text box, e.g. **zXY** for movement first in Z- and then in the XY-direction. Only the axes which are listed within this text box will be driven. If the user enters capital letters, the stage moves in these axes simultaneously. If small letters are entered, the stage moves sequentially in one axis after the other.

This may be important if a tilted wafer is moved to a position near the pole piece. In this case it is recommended first to drive the tilt axis to zero and the Z axis to a low position before moving X and Y axes.

- For information purposes, enter a comment within the respective text box.
- Choose **OK** to use the settings.

2 Eucentric mode movement

NOTICE

It is prerequisite for the **eucentric drive mode** as well as for an accurate coordinate transformation that the position of the rotation and the tilt axes are properly calibrated. (For more information refer to ⇒ *Tilt and rotation axis calibration dialog* on page 19-21).

STEP 1: W adjustment

Before using the **eucentric drive mode**, you have to adjust the W-coordinate.

- Go to the **Adjust UVW** dialog and select the **Adjust W** tab.
- Focus the beam on the sample.

- Select the **Pipette (Read)** button to read the current working distance setting of the column.
- Press **Adjust** button to apply new W adjustment.

It is recommended to adjust the W coordinate when the sample is not tilted
(T = 0°).

It is recommended to read the chapter ⇒ *Adjust W tab* on page 12-18 to enhance understanding of W adjustment functionality.

STEP 2: Launch a drive command

- Use the **Stage Control** dialog or a **Positionlist** entry to launch an eucentric movement.
- If enabled, survey the displayed Drive Table and press **OK**.

The eucentric drive mode is not available for UV windows.

NOTICE

Risk of collision: The knowledge about the current sample height is essential for coordinate processing. An inadequate **W adjustment** will lead in incorrect eucentrical movements and **W-limit** calculations.

NOTICE

The eucentric drive mode handles only the sample position defined at the beginning of the movement. It does not prevent collisions at other areas and positions of the sample holder.

C Software Reference

You can call up **Stage Control** by clicking the (Stage Control) icon in the control bar.

This will open up the following dialogs: **Coordinates**, **Stage Control**, **Find Home Position**, **Laser Stage Control** and **Touching Alarm**.

1 Coordinates dialog

The number of displayed coordinates depends on the number of motorized axes of the system. It may vary between 2 axes, i.e. X and Y, and up to 5 axes, i.e. X, Y, Z, R, and T. Any combination is possible. In most cases however, the only motorized axes are the X and Y or the X, Y and Z axes. All displayed values are refreshed periodically. The difference between stage X, Y and Z coordinates and sample U, V and W coordinates is defined by a complete three-dimensional transformation.

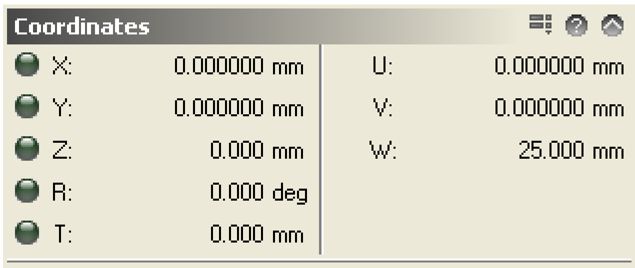


Figure 8-7: **Coordinates** dialog

Within **Coordinates**, a series of light indicators, next to each stage axis, show the status of the stage:

- Gray: stage is in standby

- Green: stage is moving

- Red: stage is locked

To obtain an overview of the current relationship between the stage and the global and local coordinate systems, select **Menu➡Coordinates Overview** via the button in the **Coordinates** dialog

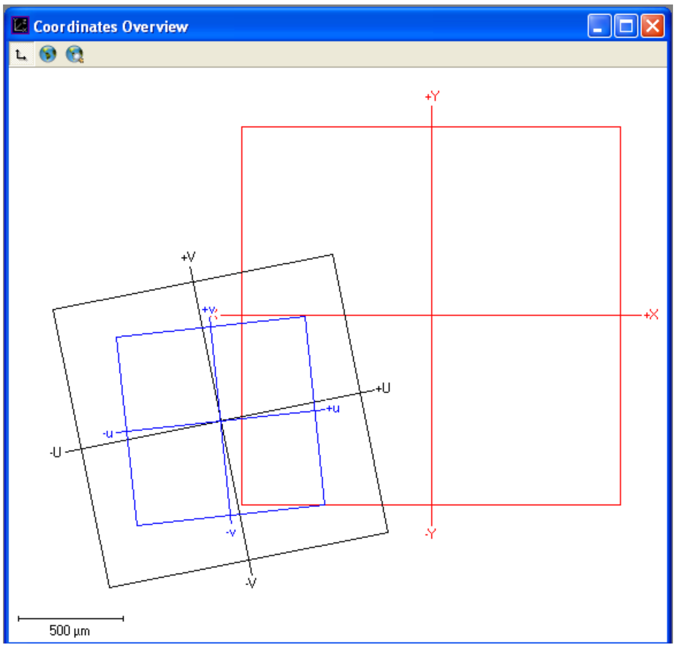


Figure 8-8: **Coordinates Overview** window

1.1 Display options

Basic display option settings can be chosen using the command **Coordinate Display** via the menu or by **right mouse click** anywhere in the **Coordinates** section. These settings are assisted by the **Coordinate Display Options** dialog.

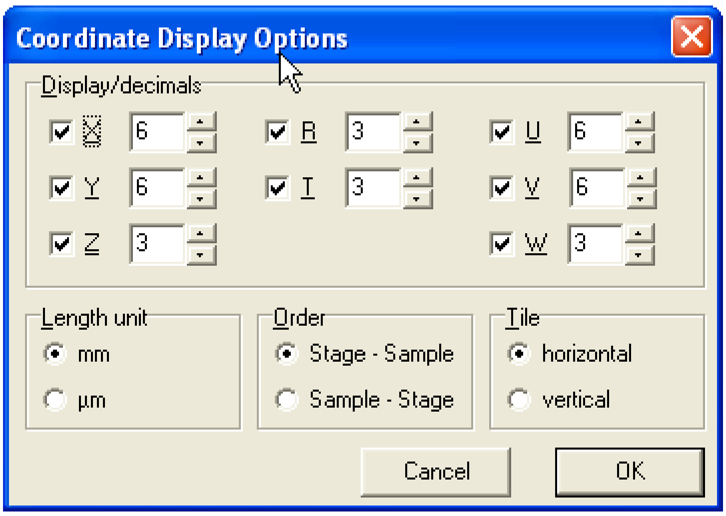


Figure 8-9: **Coordinate Display Options** dialog

Control element	Function
Display decimals	Select the checkbox for each axis to be displayed, i.e. if one of the axes should not be displayed, the related checkbox must be cleared. The display resolution, i.e. number of decimal places, can be entered for all axes individually within the text boxes.
Length unit	The metric unit setting is valid only for metric axes X, Y, Z, U, V and W and not for R and T. Units can be set to mm or µm. Depending on this setting, the number of digits will change, but the position data remain the same. Change the number of decimals accordingly.
Order	Use this option to select which coordinates are displayed first, either **Stage** or **Sample**.
Tile	Select the option **horizontal** to display stage coordinates and sample coordinates side by side. By selecting the option **vertical**, they will be displayed one beneath the other.

2 Stage Control dialog

The **Stage Control** allows control of stage movements, by sending single commands to the motor controller. Parameters may be entered in the tabs **Drive**, **Step**, **Positions** and **Stage Lock.**

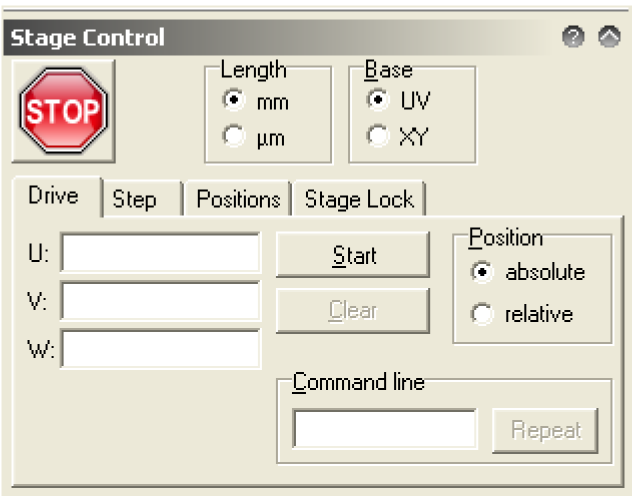


Figure 8-10: **Stage Control** dialog

The top of the dialog box includes the following:

Control element	Function
Stop	To stop all motors immediately. The command is used as an Emergency Stop to cancel any stage movement or to avoid damage in an unexpected situation. For some third party motor controllers, this button may not be available.
Length	To select the unit basis for all drive commands, except drive commands in the **Command line**. You may select **mm** or **µm**.
Base	Choose the coordinate system to be used for addressing, i.e. sample coordinates in **UV** or stage coordinates in **XY**.

2.1 Drive tab

The **Drive** tab enables the user to enter the stage destination values

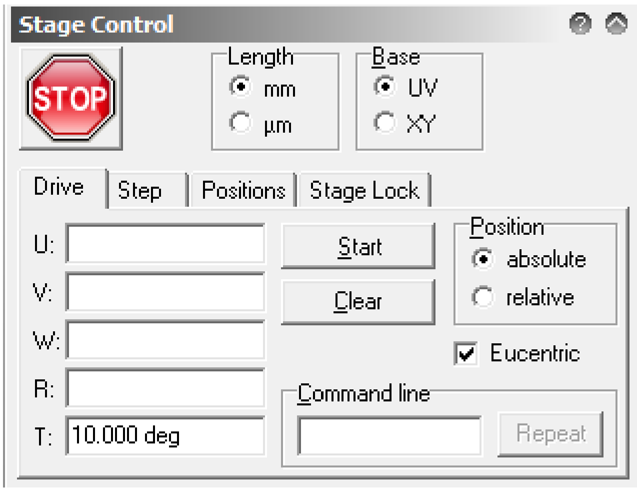


Figure 8-11: Stage Control - **Drive** tab

Control element	Function
Axis input fields	Enter the destination values for **U**, **V** and **W** or for **X**, **Y**, **Z**, **R** and **T**, depending on the Base selection.
Start	To start the movement
Clear	To clear all text boxes
Position	Choose the addressing mode, i.e **absolute** movement or **relative** movement.

Control element	Function
Command line	Digital command input is done via this text box. To repeat the last command, choose the **Repeat** button next to the text box.
Eucentric	When checked, R and T as well as U and V drive commands from the **Stage Control** dialog are performed in eucentric mode.

It is not recommended to mix X, Y or Z with R or T movements when the **Eucentric** check box is checked. This will combine eucentric and non-eucentric movements in one driving task. In this case, the software will start with the eucentric movements and will then perform the non-eucentric movements.

Eucentric drive mode commands for R and T are also supported by the command line. They can be specified to be absolute or relative to the current position and are represented by a prefixed “e”.
EXAMPLE:
The command "e30r" moves the stage rotation-eucentrically to the R coordinate 30°.

2.2 Step tab

The **Step** tab enables the user to define the **Stepsize**.

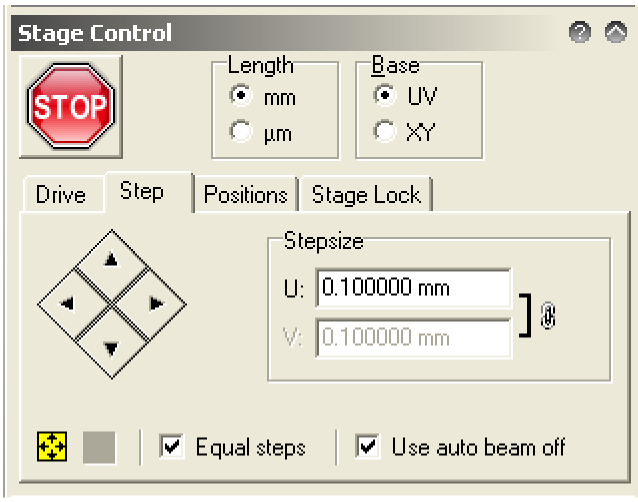


Figure 8-12: Stage Control - **Step** tab

Choose one of the arrow buttons to move the stage in the corresponding direction, using the distance given in the input field **Stepsize.**

Control element	Function
Base	To select the base, either **UV** or **XY**, for the subsequent drive commands.
	To obtain the current writefield size.
Stepsize	Type in the distance for subsequent drive commands. Select **Equal steps** to link the movement in the two orthogonal directions. Choose one of the two buttons to read the current writefield or the current distance between chips.
Equal steps	If the checkbox **Equal steps** is checked, the step size in U and V directions will be equal.
Beam	If the checkbox **Use auto beam off** is selected, the beam will be blanked during stage movements, for example when the beam moves from one line to the next in a larger writing pattern.

If the checkbox **Use auto beam off** is checked, the beam will be blanked during stage movement. In order to enable this function, the Autobeam has to be enabled in the **Module Status** settings. It will be set up correctly upon installation. If, for any test and development purposes, the **Autobeam** function should be disabled, for example to observe the sample without any patterning taking place, go to the main menu **Extras**➡**Settings**➡**Virtual Beam Control**. Double click on **Virtual Beam Control**, a dialog will open. Select **Autobeam off**.

2.3 Positions tab

A set of 10 positions can be defined by the user and recalled on request.

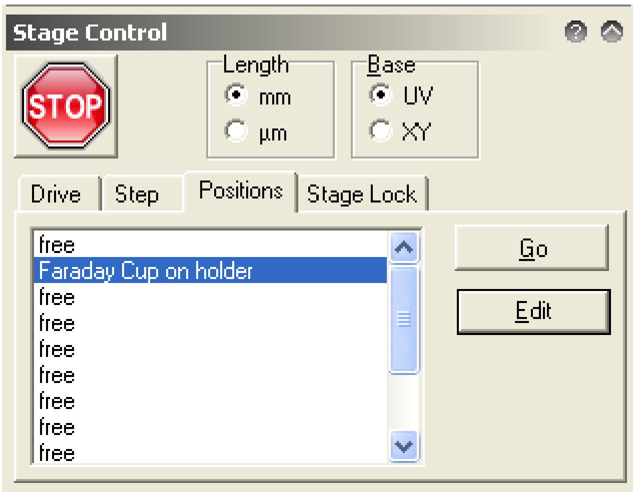


Figure 8-13: Stage Control - **Positions** tab

Control element	Function
Go	To drive the stage to a position that has been selected from the dropdown list.
Edit	To edit a position that has been selected from the drop-down list. An **Edit User Defined Position** dialog opens.

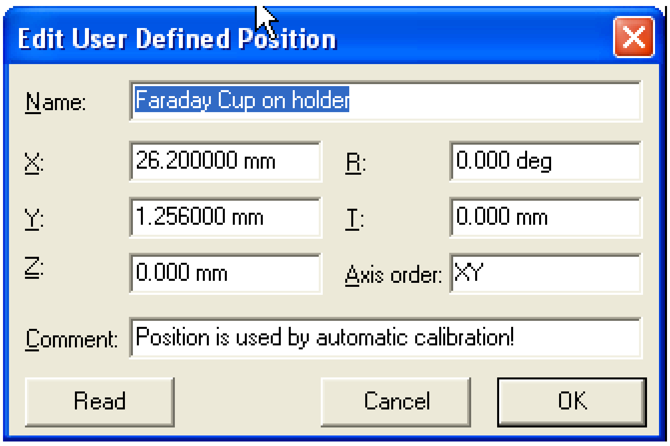


Figure 8-14: **Edit User Defined Position** subdialog

Control element	Function
Name	Enter the file name for the user-defined position.

Control element	Function
X, Y, Z, R, T	Enter the destination values for **U**, **V** and **W** or for **X**, **Y**, **Z**, **R** and **T**, depending on the Base selection.
Axis order	To define the order of the axis
Comment	To enter a description of the user-defined position
Read	To read in the current position

2.4 Stage Lock tab

This functionality is only available for user level SYS and for Raith motor control or for demo motor control. Here it is possible to lock individual axes to prevent them from moving. Use the menu **Extras**➡**Module Status**➡**Stage Control**➡**Enable Stage Lock Control** = **ON**. Restart the software to update the **Stage Lock** status.

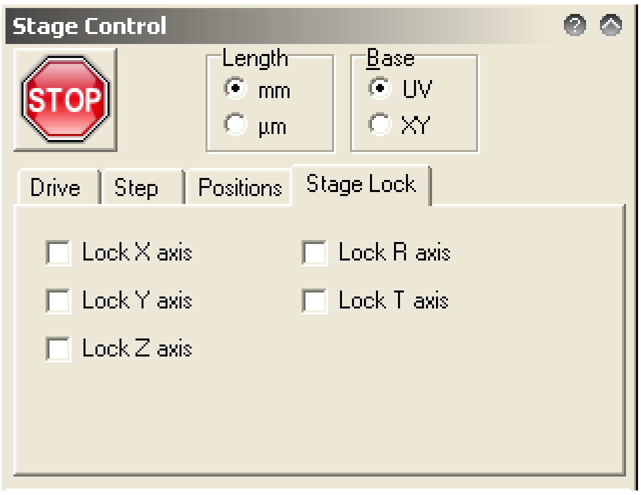


Figure 8-15: Stage Control - **Stage Lock** tab

Control element	Function
Lock X, Y, Z, R, T	To lock one or more axes individually, check the checkbox for the relevant axes.

3 Find Home Position

If you accidentally lose the coordinate system, the **Find Home Position** routine recovers and resets the setup.

The definitions of limit switches and home positions are required for the relevant axes within the initialization routine. The **Find Home Position** routine can therefore be performed only for motorized axes which are equipped with limit switches.

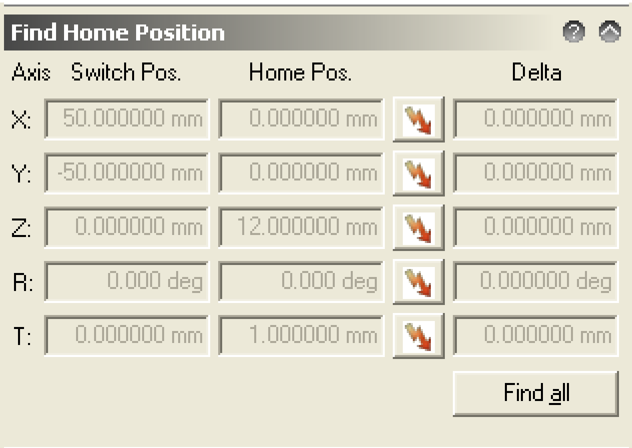


Figure 8-16: **Find Home Position** dialog

Control element	Function
X, Y, Z, R, T	The values for the axes **U**, **V** and **W** or for **X**, **Y**, **Z**, **R** and **T**, depending on the Base selection, are displayed
	Click on the Flash button next to any single axis to initiate the axis. The stage then moves towards the limit switches and the system sets the corresponding **Switch** position. The stage then moves to the corresponding **Home** position.
Switch Position	The **Switch** position will be displayed for the selected axis.
Home Position	The **Home** position will be displayed for the selected axis.
Delta	The **Delta** value between the current position and the **Target** position at limit switch will be displayed.
Find all	Click the **Find all** button to initiate all axes.

4 Laser Stage Control

Note that this module is only available for stages which can move under Laser or encoder control.

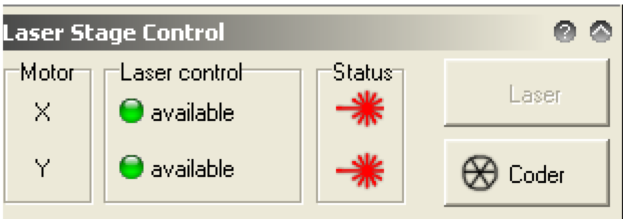


Figure 8-17: **Laser Stage Control** dialog

Control element	Function
Laser control	A green LED indicates that the laser signal is available for the corresponding axis.
Status	Indicates the mode that is currently used.
Laser	Switch to Laser mode.
Coder	Switch to encoder mode.

Please be aware that when working in encoder mode, it is not possible to perform any writefield alignments. In encoder mode, the accuracy of the stage is in the micrometer range and it can therefore not measure accurately enough for alignment.

5 Touching Alarm

The **Touching Alarm** can either be **Armed** or **Disarmed**. It will be activated to warn the user prior to a possible collision between the sample holder and the pole piece. The stage will be stopped, an alarm message will be displayed on the monitor and a beep signal will sound.

The touching alarm is only available on the RAITH150^{TWO} systems.

Figure 8-18: **Touching Alarm** dialog

Control element	Function
Status LED	The Status LED indicates the status of the touching alarm. Green - Armed Yellow - Disarmed
Arm, Disarm	To toggle between **Armed** and **Disarmed**

Chapter 9
Positionlist

This section describes the creation and execution of the Positionlist, which functions as a job list, in which various procedures for Patterning, Adjustment and Alignment, Positions etc can be listed and run automatically.

This chapter details the following:

- Functional description and background (⇒ *Functional Description* on page 9-2),
- Positionlists and how to open them (⇒ *Opening a Positionlist* on page 9-4),
- Tasks describing how to edit a Positionlist (⇒ *Editing a Positionlist* on page 9-5),
- Effects of changing values in Scan Manager (⇒ *Changing values in Scan Manager after insertion into Positionlist* on page 9-5),
- Property dialog description (⇒ *Using the Properties dialog* on page 9-6),
- Applying filters to previously acquired data (⇒ *Using the Matrix filter* on page 9-8),
- PLS preprocessor options and how to apply them (⇒ *PLS-preprocessor* on page 9-21),
- Complete description of the dialogs, menus and control elements of the module (⇒ *Software Reference* on page 9-12),
- Positionlist window (⇒ *Positionlist working area* on page 9-14),
- Positionlist tool bar description (⇒ *Tool bar* on page 9-12),
- Menu commands (⇒ *Main menu* on page 9-30),
- Mouse and keyboard commands (⇒ *Mouse and keyboard commands* on page 9-64),
- Drag and drop concept (⇒ *Drag and drop* on page 9-65),
- Configuration (⇒ *Configuration* on page 9-66).

You may call up the Positionlist via **File➡Open Positionlist** or **New Positionlist.** Alternatively you may use the tool bar button in the **Positionlist** window.

A Functional Description

A Positionlist is a spread sheet-orientated tool within the Raith NANOSUITE software, which performs the execution, generation, administration, and documentation of various tasks:

- Adjustment and alignment tasks, i.e. performing and then evaluating a complex sequence of mark scans.
- Stage control tasks, such as driving the stage sequentially to multiple positions in stage or sample coordinates.
- Patterning tasks, performing patterning of one or more selected structures at different sites on the sample, with different parameter options.
- Performing scans of images or of linescans and automated tasks.

The various tasks defined in a Positionlist can be configured by the user in any selected sequence and combination. Positionlists required for standard adjustment and alignment procedures are generated automatically.

A Positionlist is defined and stored as an ASCII data file, which normally is indicated by the extension ***.pls**. The Positionlist concept of the Raith software allows very flexible configurations and the handling of different list formats. They are adaptable, therefore, to almost any application. Even though for normal operation, a single Positionlist is often used and fully sufficient for the task, it is possible to load and operate multiple Positionlists simultaneously.

1 Positionlist window

Open an existing Positionlist with extension *.pls, by choosing the command **File➡Open Positionlist** or define a new one by choosing **File➡New Positionlist**.

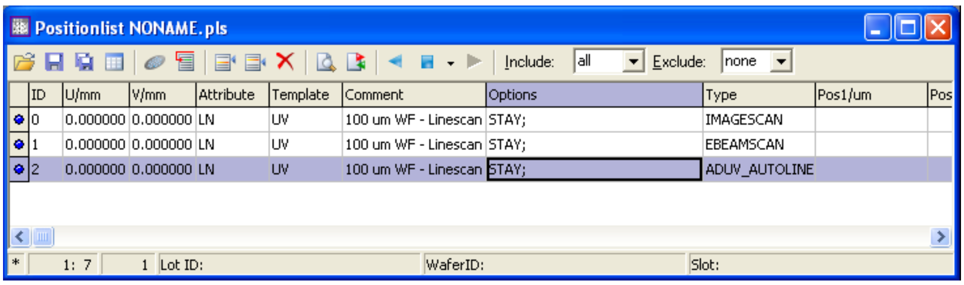


Figure 9-1: **Positionlist** window

Control element	Function
Task list (1)	The task list displays each task on a separate line. Each task is defined by various data which are stored in separate fields. Thus the task list can be seen as a spread sheet.
Tool bar (2)	The most frequently used commands are directly available from the tool bar. Using the list boxes, it is possible to include or exclude some tasks according to their attributes.
Information line (3)	Information associated with this Positionlist is displayed here. It displays the following from left to right: - The line currently selected and how many lines exist. - A star indicates that this Positionlist was modified and contains some data which has not been stored so far. If the Positionlist is used in connection with the defect finder, **Lot ID**, **Wafer ID** and **Slot** can be used to identify a specific wafer.

A Positionlist can be opened and used in several ways. Procedures from a number of windows such as **Column Control**, **Writefield Manager**, **GDSII Editor**, **Automation** as well as the **Scan Manager** window can be transferred via drag and drop into the Positionlist.

Positionlist execution is fundamental to the Raith software, as so many tasks can be executed automatically from the Positionlist. The procedures within the Positionlist will be executed sequentially.

The Positionlist consists of a table. In the first column, the positions in XY or UV are displayed. The Positionlist can initiate stage movement in UV or XY coordinate systems, in either relative or absolute values. Any position can be executed and the stage will drive to the specified position.

The position will then be scanned. A number of images can be acquired and a number of filters can be applied.

Procedures from Writefield Alignment, Adjust UV, Linescans, Images etc. Can all be executed from the Positionlist. Patterning of GDSII structures can be written, macros and scripts can be executed etc.

B Tasks

1 Task list

Each line contains the data set of a task. Tasks may include alignment procedures, Imagescans, Linescans, structures to be exposed, line width measurements, or simply stage positions to be addressed.

1.1 Opening a Positionlist

▶ To open a Positionlist, click on the menu bar, **File➡New Positionlist**. This command is always available, independent of the windows currently open via the control bar.

▶ A Positionlist is now opened. It is possible to open several Positionlists at the same time.

In the previous software version, each row of the Positionlist had a specific buffer capacity of approximately 1 kb per row. The number of characters available for each column was limited. Consequently, if long file names were used, perhaps from the **GDSII Editor** or **Scan Manager**, the file path was sometimes corrupted when the file name contained too many characters. Due to the buffer capacity, the number of rows in the Positionlist was also limited, with the maximum default number of 10,000 rows, which was sufficient for most applications.
Now the buffer capacity for each Positionlist is dynamic, the number of characters in each column is no longer limited, nor is the number of lines in a Positionlist.

▶ Once a Positionlist is open, any procedure from the **Scan Manager** can be included, via the drag and drop function, such as Linescans, Images, Writefield and Beam Tracking Alignments.

▶ A GDSII design can also be included via a drag and drop function.

▶ When a GDSII structure contains several layers, a dialog will open automatically, in which the user can select layers from the GDSII structure.

When the selected GDSII structure only contains one layer, the dialog for different layers will not open, and the GDSII structure is automatically entered into one row of the Positionlist.

▶ Select the layers that you wish to pattern. Each selected layer of the GDSII structure will be entered into different rows of the Positionlist, if the **entry for each selected layer** option is active. Otherwise the GDSII structure is associated with one row in the Positionlist and all selected layers are stored in the **Layer** field.

Defining the number of layers for patterning prior to executing the Positionlist has the great advantage that each selected layer is already defined in the Positionlist. The Positionlist can therefore be executed without the user being present, since no further pop-up dialogs will be displayed that require the user to select the layers.

Using a right mouse click, the context menu will open. Selecting **Properties** from the dropdown list will display the properties corresponding to the currently selected type, e.g. for Linescans and Imagescans, the Scan properties will be called up and for GDSII structures, the Patterning properties will be called up etc.

1.2 Editing a Positionlist

The Positionlist can be edited column by column.

In theory, the columns can also be directly edited in the Positionlist, but editing them via the **Edit Position** dialog has the advantage that all columns are visible, as sometimes columns might not be displayed.

- Using **Edit**, it is easy to see and to understand how the Positionlist is structured. At first, an ID (Identity) is displayed, as well as a line number. The ID number can be edited. The ID does not have to be equal to the line number.
- It is possible to use the same ID number several times, although it does not make any sense to do so.

The ID number is useful when using filters or when searching for an ID. The ID has no influence on the patterning.

1.3 Changing values in Scan Manager after insertion into Positionlist

Within the **Scan Manager**, the user may define all the key parameters such as dimensions, direction, rotation etc. As soon as the Scan Manager procedure is inserted in the Positionlist via drag and drop, these parameters are then stored within the Positionlist. If the user changes any of these parameters afterwards in the Scan Manager, these values will not be updated subsequently in the Positionlist.

It is important to remember that the parameters contained in a Scan Manager procedure are stored in the Positionlist at the time of the drag and drop. Any other alterations in the Scan Manager afterwards will be disregarded in the Positionlist.

- Whenever the user changes the values in the Scan Manager, the corresponding line in the Positionlist should be deleted and the edited Scan Manager procedure should be inserted again via drag and drop into the Positionlist.

This is valid for all dimensions defined under the **Main** and **Advanced** tabs in the **Scan Manager**.

Any parameters entered in **Post Processing** and **Evaluation** can be edited after the procedure has been included in the Positionlist, as these two tabs deal with the processing and evaluation after the procedure has been performed. The post processing and evaluation tasks are not therefore initiated from the Positionlist, they are performed via macro commands.

The commands will be carried out according to the latest setting of the **Post Processing** and **Evaluation** tabs in the **Scan Manager**.

1.4 Using the Properties dialog

- The **Properties** dialog can be called up via the context menu using a **right mouse** click.
- You can edit the values in the Properties dialog to change the settings.

It is recommended to use the **Properties** dialog instead of the **Edit** dialog to change the values, since the Properties dialog is more user friendly, as it only displays the relevant parameters for the type of procedure selected.

For example, if a **Patterning** procedure is selected in the Positionlist, the parameters for database and structure are shown and the corresponding procedure is displayed in the **Comment** text field.

The **Select layer** dialog can be opened and the **Working area** can be edited. The starting and end points are shown. The **Start** position for U and V can be entered.

The Start position for any Patterning procedure is always defined as the center position of the first writefield.

The system may alternate between the **Global/Local UV** coordinate systems.

Within the **Patterning** dialog the user can change patterning parameters such as dwell time or step size for the currently selected patterning tasks.

- Clicking on **Patterning Parameters** will display additional parameters.
- Uncheck default values, in order to edit the values in this dialog.

- The **Calculator** can be opened, and the parameters can be calculated again, or a particular setting can be changed.
- Using **Times** in the **Patterning Properties** dialog, the time for the patterning procedure can be calculated.

1.4.1 Overview of Properties dialog depending on selected task

Various tasks can be inserted into the Positionlist via drag and drop. Depending on the originated module, a task type will automatically be assigned, (⇒ *Drag and drop* on page 9-65).

Clicking on the task using the right mouse button will open the context menu. Choose the menu item **Properties** to open the corresponding Properties dialog.

Drag and drop into Positionlist	Properties Function
GDSII Design	**Patterning Properties** dialog will be displayed.
Column Control	**Edit Position** dialog will be displayed.
Writefield Manager	**Edit Position** dialog will be displayed.
Scan Manager	**Scan Properties** dialog will be displayed.
Automation	**Edit Position** dialog will be displayed.

1.5 Assigning a Wafermap

- A corresponding wafer layout can be created and linked to the Positionlist. (⇒ *Wafermap window* on page 16-15).

This is often used for defect review applications. The Positionlist will contain the positions of the defects from the wafermap. When the button in the Positionlist tool bar is clicked, the wafer layout will be opened and the defect positions will be marked on the wafer.

- Use the **Edit➡List properties** menu item to open the **Positionlist Properties** dialog and choose the appropriate wafer layout file for the current Positionlist.
- Activate the **Save as default** option to assign the wafer layout to all Positionlists.

1.6 Using the Matrix filter

- **Matrix filter** is used, for example, for scanning images. For an image to be scanned over a large area, e.g. 100 µm x 100 µm, the user can define that this image should be scanned in a 10 x 10 matrix. These images will be stitched, comprising the required 100 µm x 100 µm image size.
- Set the **Element order**. Firstly, the image is scanned 10 times in the U direction, and afterwards in the V direction. It is also possible to define the order the other way round.
- A **Meander Scan** can also be selected. This will save overall scanning time, since the stage moves a shorter total distance during a meander movement.
- The **Dose factor** is not used for image scans, but it is used for GDSII structure patterning. A dose scaling for the same structure can be defined. The user can then afterwards choose the optimized dose for the selected position. The dose factor can be either added or multiplied.
- **Calculate automatic** for U can be chosen, after the user has selected the value for the U direction. For example, the user might choose to add a dose factor of 0.1 in the U direction. Starting from a dose factor of 1.0, it will then be increased to 1.1, 1.2, 1.3 etc. When the first GDSII structure is written in the V direction, the software will then calculate the dose factor automatically and will start with 1 + (number of columns *0.1).

1.7 Using Linescan Analysis filter

- The post processing and evaluation parameters for a linescan can be defined in the **Scan Manager** or using the **Linescan Analysis**, **Filter** dialog.
- Using **Threshold Analysis**, limits will be defined. **Position 1** is defined at the left edge of the peak, **Position 2** at the right hand edge of the peak. **Position 3** can either be the central position between **Position 1** and **Position 2** or the width of the peak, depending how the user has defined it.
- When **Edge detection** has been selected, the width of the edge will be measured.
- When **Pitch detection** has been selected, the distance between the adjacent flanks of two peaks will be measured.

When the **Post Processing** and **Evaluation** parameters are already defined in the **Scan Manager**, then post processing and evaluation will be carried out immediately when the linescan has been completed in the Positionlist. This has the advantage that the user can view the processed results straight away, while the remaining Positionlist is still being performed.

The disadvantage is that the overall time for the execution of the Positionlist will be greater, since the linescans will be post-processed and evaluated prior to executing the next task in the Positionlist.

An alternative method is to define the parameters in the **Linescan Analysis Filter**. The linescan post processing and evaluation will now be carried out after the full Positionlist has been completed.

- It is possible to combine several algorithms, but not all combinations are analytically meaningful. For example, **Noise reduction** and **Threshold** would be a good combination, as would **Offset correction** and **Threshold**. Threshold combined with Edge detection would, however, not be a meaningful combination.

1.8 Preprocessing a Positionlist for multi-field patterning without a laser stage

- Use the button in the Positionlist tool bar to enable the setup of the patterning of multiple fields via the Positionlist without a laser stage. If no laser stage is installed, the stitching accuracy will not be as high, but it enables the user to set up and perform multi-field patterning.
- Select **Multifield** patterning. The user now needs to define a working area, which must be larger than the writefield.
- Firstly, place a writefield in the lower left corner of the working area.
- Then fill the working area with writefields from bottom to top and from left to right. The Design coordinates will be applied.

In contrast, for single field patterning, it is only possible to write one structure, as defined in a row of a Positionlist. The maximum size of the structure is limited to the writefield size. If, in error, the structure is larger than the defined writefield, the software will not execute the patterning process.

Preprocessing enables the user to execute a ‘quasi-multi-field patterning’ without a laser stage and consequently without the advantages of the accuracy of real stitching.

Preprocessing for the preparation of multi-field patterning is possible. The software will automatically create several positions and each position in the Positionlist will be linked to a writefield containing the GDSII design elements.

1.9 Applying a Working Area Matrix

One way to prepare GDSII design for multifield patterning via the **Positionlist Preprocessor** is to apply the **Working Area Matrix**. The software generates equally sized working areas at each position in the Positionlist, which together correspond to the whole structure. This procedure is executed automatically by the software.

- The GDSII design can be dragged and dropped into the Positionlist with its total overall working area.
- Open the **Preprocessing** window, via the button in the Positionlist tool bar. Click on the **WA Matrix** tab.
- If, for example, the Design covers 9x9 writefields for stitching, the **Design Start Point** is normally located in the lower left corner. Other Design Start Points can be specified.
- Choose either the Default position in the lower left corner or define a selected position.
- Define the **Step Size** and the **WA size**. The step size is the distance to the next starting point.

Normally, the **Step Size** and the **WA size** should be identical.
For example, if larger overlays are required, the sizes for Step Size and WA size will differ. To achieve an overlay, the WA size must be larger than the Step Size.

The advantage of the WA Matrix is that the GDSII structure itself remains unchanged. This makes it easier for the user to work with the GDSII design. It is not necessary to start the PLS preprocessor again, if the original GDSII design has been edited.

If the GDSII design is very large, it would mean that for each scan executed from the Positionlist, the complete GDSII design structure would need to be loaded again to the software, which might be time consuming. In such cases, **GDSII Fraction** is recommended.

1.10 Using GDSII Fraction

If the user wants to minimize the patterning time of the Positionlist, it is recommended to use **GDSII Fraction**. This functions in a similar way to **WA Matrix**, but it actually cuts the GDSII design itself. This leads to new GDSII Design structures.

- Select a GDSII design and drag and drop it into the Positionlist.
- Open the **Preprocessing** window via the button in the Positionlist tool bar. Click on the **GDSII Fraction** tab.
- Select an appropriate fracturing area. The GDSII Design structure will be cut into several writefields inside the selected fracturing area.
- Check the checkbox for **PLS layer only**, if you wish to apply it only to the selected layer.
- Uncheck the checkbox for **PLS layer only**, if you wish to apply it to all layers.
- Click on **Apply**.

C Software Reference

1 Positionlist window

In the **Positionlist** window, you can insert and execute procedures from:

- **Scan Manager**, such as adjustment and alignment procedures, Linescans, Images, Image Linescans and Beam tracking.
- **Writefield Manager**, where magnification and field sizes are defined.
- **GDSII Editor**, to execute patterning of structures.
- **Column Control** to carry out some column settings, such as Standby or Shutdown.
- **Automation** to perform automated procedures.

These procedures can be inserted into the Positionlist via the drag and drop function.

You may call up the Positionlist via **File menu➡Open Positionlist**. Several Positionlists can be opened at the same time.

1.1 Tool bar

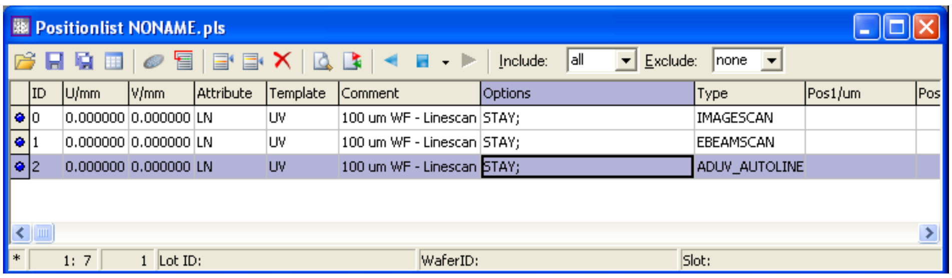


Figure 9-2: **Positionlist tool bar**

This bar contains some of the frequently used commands as buttons.

Control element	Function
	To open a new Positionlist file
	To save the Positionlist file
	To save the Positionlist under a different filename

Control element	Function
	To edit the list properties. A new dialog will be opened, in which all properties can be edited.
	To assign a wafermap to a position within the Positionlist. Clicking the **Open wafermap** button will display the wafermap. If the Positionlist includes some defects, these defects will be shown as well.
	To apply the **PLS Preprocessor**, if the user wishes to perform multi-field patterning without a laser stage. Clicking the button will open a new window, consisting of four tabs: **Matrix Copy**, **WA Matrix**, **Move Positions** and **GDSII Fraction**. It enables the patterning of multiple fields via the Positionlist.
	To insert a new position above the row of the current position
	To insert a new position below the row of the current position
	To delete selected positions
	To search for text. The search criteria can be entered, e.g. search from top to bottom or vice versa, by row or column, for all positions or for selected positions. The search can be started either from the current position or from the top.
	To find and replace text using the same search criteria as described above.
	To scan the position prior to the current position
	To scan the selected position. Using the arrow, a dropdown list will be displayed, where the either **Drive to Position**, **Scan Manager**, **Patterning**, **Execute Macro** or **Column Control** can be selected for the scan selection.
	To scan the position after the current position

Control element	Function
Include	To display or to select only a subset of the tasks, it is possible to include or exclude lines that contain a certain character in their **Attribute** field. By typing a character into the **Include** field, only the lines containing this character in their Attribute field are included in the display.
Exclude	Typing a character into the **Exclude** field prevents the lines containing this character in their Attribute field from being displayed.

One may enter several attributes without separation. Attributes are case-sensitive, so entry of lower case characters or upper case characters will achieve different results. In addition, including and excluding can be used at the same time and only lines fulfilling both conditions are shown. Default values are all for **Include** and none for **Exclude**. The attribute entry is stored within the related dropdown list as a history, but this history is available only until the Positionlist is closed.

The search restrictions do not change the content of the Positionlist. This means that those data lines that are not displayed are kept, staying in the Positionlist data file. They can be made accessible by changing the search restrictions.

1.2 Positionlist working area

The task list is represented in a common table format, so that the standard editing functionality is also available for a Raith Positionlist. This means that it is possible to adapt the column width using the mouse and edit the contents of each cell by placing the cursor into it.

Each Positionlist consists of some fixed columns as a serial number **No.**, an identification number **ID**, coordinates **XYZRTUVW**, characterization **Attribute**, addressing information, **Template**, individual remark **Comment**. In addition, each Positionlist can have some application-specific columns. The columns have their own properties, such as name, optional units, display width, data size, default value, etc. Setting of these properties is described in the **Configuration** (⇒ *Configuration* on page 9-66).

Control element	Function
No.	Serial numbers are set automatically by the Raith NANOSUITE software. They cannot be changed by the operator. They are used for addressing commands and to select a data line for scanning or editing purposes. The layout of serial numbers can be changed by setting the variable **Extras**➡**Settings**➡**Positionlist**➡ **Line Number Format**. The syntax is: Line Number Format = **%<0><digits>d** where **<0>** is optional to display leading zeros, and **<digits>** is the maximum number of digits for each serial number. The serial number is dynamically generated and may change due to another sorting of the lines, e.g. after a filter command, etc.
ID	ID numbers may be freely defined and changed by the operator. They are used for addressing commands and to select a data line for scanning or editing purposes. In contrast to the serial number, the ID number of a data line will not change due to display or filter restrictions.
X, Y, Z, R, T, U, V, W	Any Positionlist must have at least one or more coordinates from the range **X, Y, Z, R, T, U, V,** and **W**.
Attribute	Attributes are very important features of the Positionlist concept, since they may contain key codes for control purposes or other important information. Attributes are also used for display restrictions as described in the section **Including and Excluding**. Attributes should be used to qualify or classify individual objects or positions. Allowed inputs for attributes are all small letters **a-z**, all capital letters **A-Z**, single digit numbers **0- 9**, and all combinations of the above.

Control element	Function
Template	Templates are used to define how the addressing of the stage is done, i.e. which of the values given under **X, Y, Z, R, T, U, V**, and **W** are used and in which order. Within the Raith NANOSUITE software, template generation and handling are fully automated and require no interaction by the user. For special purposes, the user can use and modify the templates in order to define which coordinate combinations are to be used for driving the stage. Data lines without a template are ignored for stage addressing. The standard value for templates can be changed by setting the variable **Extras**➡**Settings**➡**Positionlist**➡**New Template**. This variable is used to set a standard entry when creating a new data line, e.g. by using the command **Edit**➡**New**.

Control element	Function
Optional fields	The following fields are optional.
Options	To define specific features of the task, e.g. settings defined in Template and U, V, W, X, Y, Z, R, T fields and option **STAY,** which specifies that the stage must stay at the current position.
Type	To define the type of the tasks: **EBEAMSCAN** (Linescan), **WRITEFIELD** (auto setting writefield), **EXPOSURE** (all patterning processes), **VICOL** (auto setting column parameter), **MACRO** (execute macro or script).
Size-U, Size-V	The size is displayed. When using Images, for example, the size of the image will be shown. When using linescans, the length of the line will be displayed in microns.
Points-U, Points-V	To specify the length and width for Image or Linescan in pixels (DAC steps). Also to specify which setting, Points-U or Points-V, correspond to width or length. This depends on the **Direction** setting, described below.
Direction	This is the defined direction. For example, for a Linescan it will be the main direction in which the linescan will be scanned.
Avg	This is the point average taken.

Control element	Function
Pos1, Pos2, Pos3	This is used, for example, for the evaluation of linescans, e.g. the left edge of a peak is Pos 1, the right edge of the peak is Pos 2, and Pos 3 can either be the width or the center point between Pos1 and Pos2.
Link	Link to other position in the Positionlist. This field is used by a small number of special procedures, e.g. Long Distance filter.
Layer	The layer for patterning will be selected from a GDSII structure.
Area	This is the working area.
Dose factor	This is the assigned dose factor. When a value other than 1 is entered here, then the clearing dose used for patterning tasks will be scaled by this dose factor.
Dwell time, Stepsize	These are used for the fine tuning of the patterning parameters.
Time	Time since 00:00:00 UTC on 1 January 1970 when the position was scanned.
Timestamp	The time that the position was scanned is displayed in the format MM:DD:YY hh:mm:ss.
Angle	Defines the angle relative to the main direction of the Linescan at which the linescan will be scanned.
Marksequences	Value specific to the **Scan Manager**, used for alignment and adjustment procedures. The value is automatically handled by the software.
Frame	Takes the frame average for the image post processing.

Example of Line Number Format:
Line Number Format = **%3d** displays 1, 2, 3,....
Line Number Format = **%03d** displays 001, 002, 003,...
Line Number Format = **%04d** displays 0001, 0002, 0003, ...

Depending on the type inserted into the Positionlist, the relevant columns for the selected type will be added. For example, if the Positionlist contains a GDSII procedure, the columns for Layer, Dose Factor, FBMS Area/ Line and Time will be shown.

The standard value for attributes can be changed by setting the variable **Extras➡Settings➡Positionlist➡New Attributes**. This variable is used to set a standard entry when creating a new data line, e.g. by using the command **Edit➡New**.

Generally, it makes no sense to have templates X and U at the same time. This is also true for Y and V, as well as for Z and W.

The **Information** line contains the following information from left to right:

NOTICE
For predefined tasks like a Patterning or a Mark Scan, the **Comment** field is used internally, e.g. to define which structure to expose or which scan to execute. Do not change its content.

- The line currently selected and how many lines exist.
- A star indicates that this Positionlist was modified and contains some data which has not been stored so far.

If the Positionlist is used in connection with defect finder **Lot ID**, **Wafer ID** and **Slot** can be used to identify a specific wafer. The information itself can be changed by using the command **Edit➡Sample information.**

1.3 Positionlist tool bar subdialogs

The Positionlist consists of the following subdialogs:

1.3.1 Edit List Properties

You may call up the dialog by clicking the button in the **Positionlist** tool bar.

This dialog enables the user to edit the column properties.

All available columns will be displayed. Each individual column can be defined as either visible or invisible. The default width of each column can be defined in pixels.

Some of the columns are fixed columns which can not be deleted. Other columns are user-defined, which can be deleted. The user can add columns and can also delete them. Some items or unit information can be made visible or invisible.

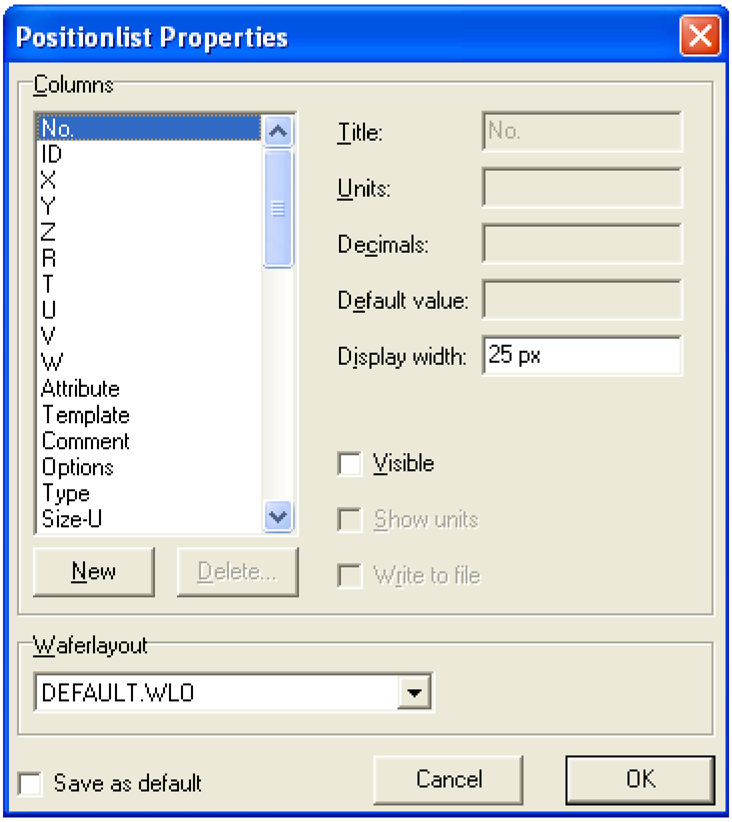


Figure 9-3: **Positionlist Properties** dialog

Control element	Function
Column	The list box on the left hand side contains the names of the **Columns**. The list starts with the fixed columns serial number **<No.>**, identification number **<ID>**, coordinates **<X, Y, Z, R, T, U, V, W>**, attribute, addressing information **<Template>**, **<Comment>**. Then additional application-specific columns follow.
New, Delete	To add a new application-specific column, click on **New**. To delete an application-specific column, select the column first and then click on **Delete**. Only application-specific columns can be deleted. To set or to change the layout properties of a column, the column name has to be selected within this list box. A selected column name appears highlighted.
Title	On the right hand side, the layout properties for each selected column name are displayed. The title can be changed only for user-defined columns but not for fixed columns.

Control element	Function
Units	The entered dimensions are displayed directly behind the column title, if **Show units** is selected. They serve for information purposes only and have no influence on calculation routines. The units can be made visible or invisible.
Decimals	The number of decimal points can be selected.
Default value	The default value can be set or changed only for application-specific columns but not for fixed columns. It is used to set a standard entry when creating a new line.
Display width	The graphical representation of the columns can be accommodated by assigning a display width for each of them.
Visible	If this checkbox is selected, the column will be displayed within the Positionlist. Units can be made visible or invisible.
Show units	Check the checkbox if you wish to display the units.
Write to file	If this checkbox is selected, the column data will be stored within the Positionlist file. The serial number will never be written to file. The application-specific columns will always be written to file.
Waferlayout	Select a file for the wafer layout from the dropdown list. Select the wafermap that should be assigned to the current Positionlist. The dropdown list contains all wafermaps stored in the user directory **Wafer.**
Save as default	To use the applied settings for all newly generated Positionlists

The **Display width** can also be set directly within the **Positionlist** window by using the mouse. To do this, select the separating line between two column names and move it. To set the optimum column width, double click the column name.

1.3.2 Wafermap

Call up the dialog by clicking the button in the **Positionlist** tool bar.

A wafermap can be linked to a Positionlist so that, by pressing the corresponding button, this wafermap is opened. The corresponding information is stored within the Positionlist, which serves as a master.

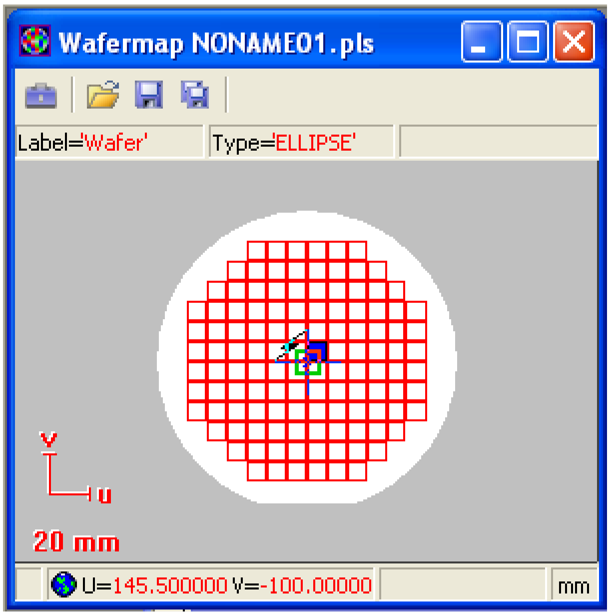


Figure 9-4: **Wafermap** dialog

To learn more about the wafermap, please read the chapter **Working with wafers**, where all functions are explained in detail, (⇒ *Working with Wafers* on page 16-1).

1.3.3 PLS-preprocessor

Call up the dialog by clicking in the **Positionlist** tool bar.

The **Preprocessing** dialog contains four tabs:

Matrix Copy, WA Matrix, Move Positions and **GDSII fraction**.

Before applying one of the functions, select either **All Positions** or the **Selected positions** to be processed.

Matrix Copy tab

The **Matrix copy** processor serves to easily generate a position matrix, consisting of a rectangular array of positions. The result of this operation is a list of newly-generated lines with the same setting but at different UV positions.

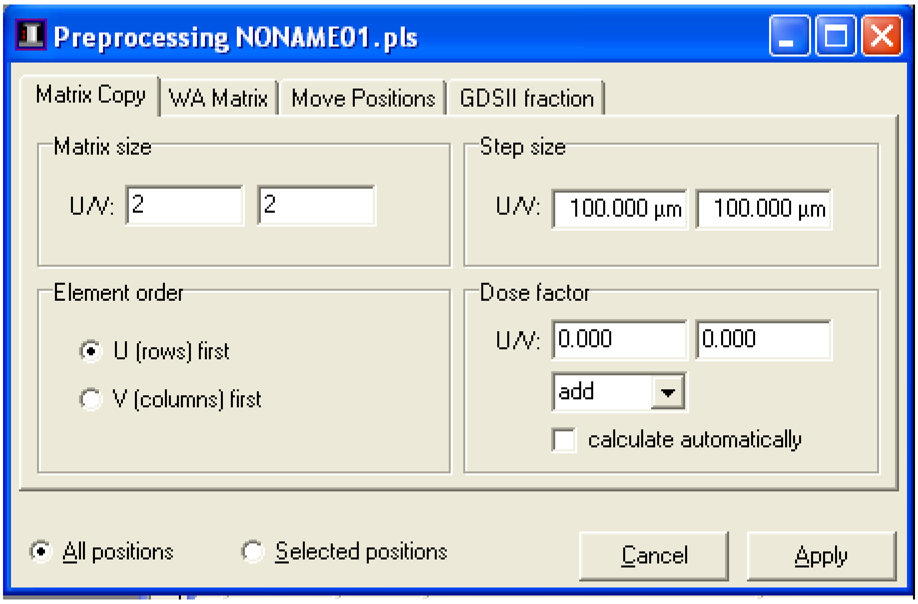


Figure 9-5: **Matrix Copy** tab

Control element	Function
Matrix size U/V	Enter the number of positions in U and V directions.
Step size U/V	Enter the distances between adjacent positions in U and in V directions.
Element order U (rows), V (columns)	Select the order of copying, either in the U direction (rows) or the V direction (columns) first.
Dose factor U/V	To change the dose factor for each position. This can be used to very easily generate a dose test. Set the dose change in U and V directions independently or select **calculate automatically** to have the same variation over the complete matrix. Select **add** or **multiply** to generate an arithmetic or geometric sequence.
All positions, Selected positions	Click on either **All positions** or **Selected positions** to apply the working area matrix.

Control element	Function
Apply, Cancel	To confirm and execute the settings for the working area matrix creation, or click on the **Cancel** button to exit without saving the settings.

WA Matrix tab

The **WA Matrix** processor serves to generate a rectangular array of working areas. The result of the operation is a list of newly generated lines with the same GDSII, at a new working area, using new UV positions.

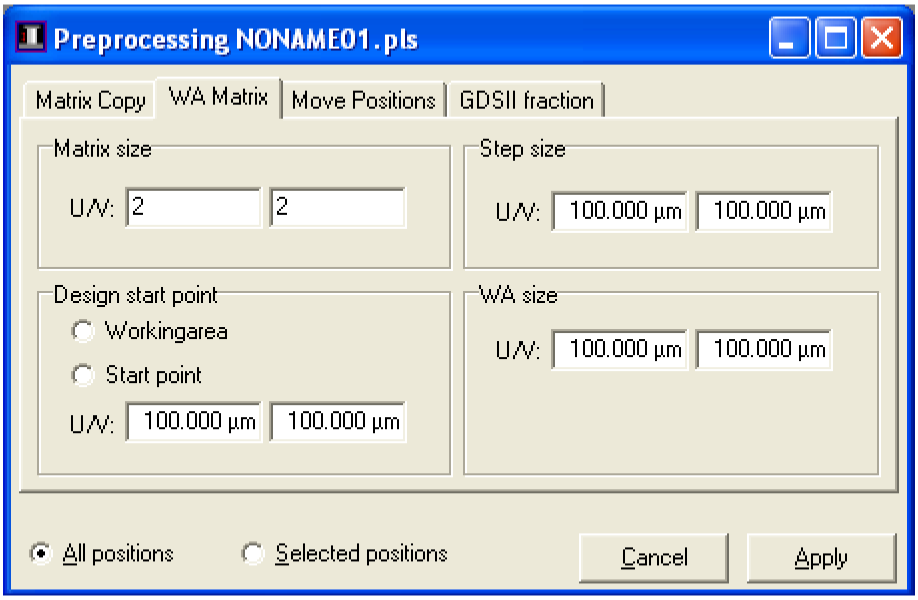


Figure 9-6: **WA Matrix** tab

Control element	Function
Matrix size U/V	Enter the number of positions in U and V directions.
Step size U/V	Enter the distances between adjacent working areas in U and in V directions.
Design start point	To start with the currently used working area. Select **Start point** to define the point independently.
WA size U/V	Enter the size of the working area in U and V.
All positions, Selected positions	Click on either **All positions** or **Selected positions** to apply the working area matrix.

Control element	Function
Apply, Cancel	To confirm and execute the settings for the working area matrix creation, or click on the **Cancel** button to exit without saving the settings.

Move Positions tab

The **Move Positions** processor serves to move positions to new UV coordinates.

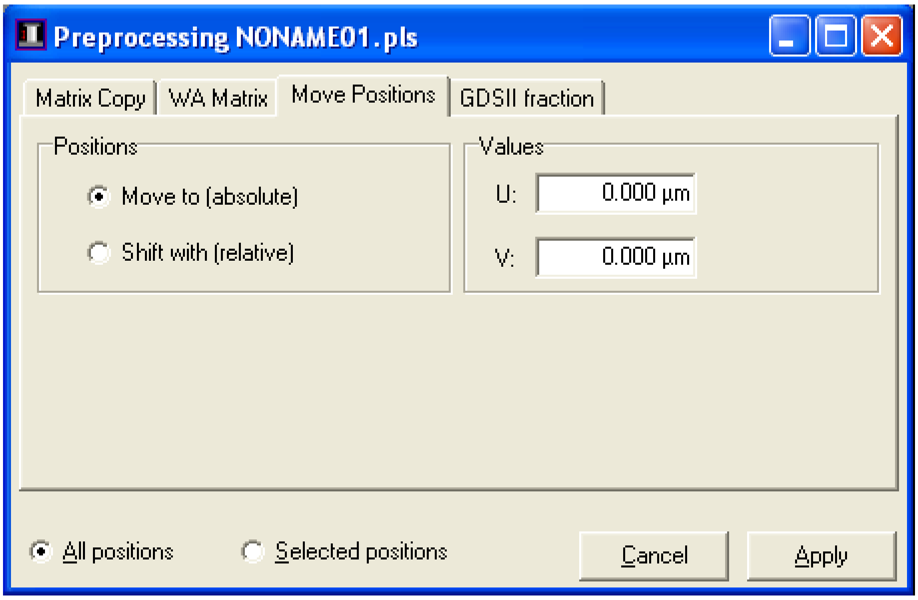


Figure 9-7: **Move Positions** tab

Control element	Function
Positions	Select either **Move to (absolute)** or **Shift with (relative)** positions.
Values U, V	Enter the values by which the position will be relocated.
All positions, Selected positions	Click on either **All positions** or **Selected positions** to apply the **Move Position** command.

GDSII Fraction tab

The **GDSII fracturing** preprocessor serves to fracture GDSII positions. The result of the operation is a series of GDSII positions in which each position is only a fraction of the original GDSII. At the same time, the UV coordinates are shifted to reproduce the way that the original GDSII would be exposed.

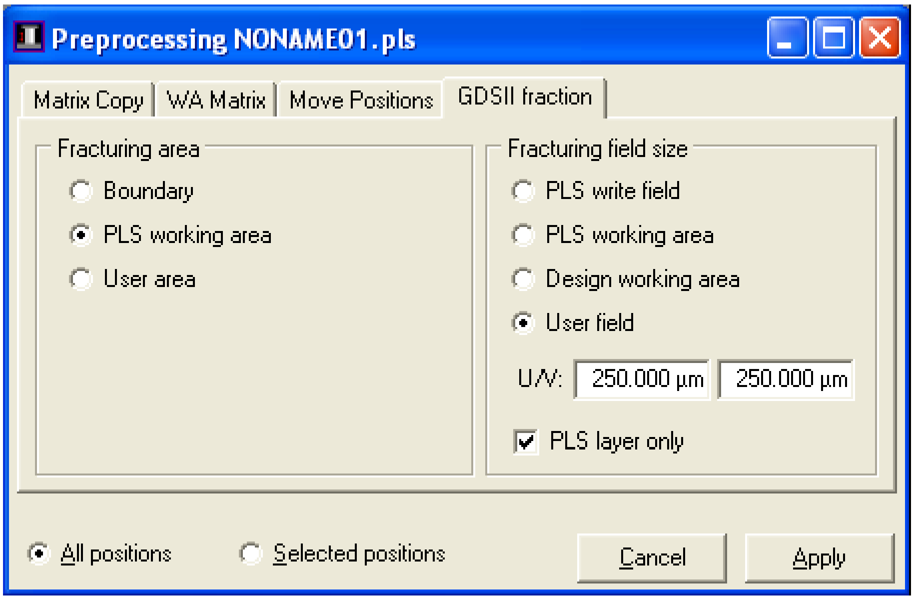


Figure 9-8: **GDSII fraction** tab

Control element	Function
Fracturing area	Select the area to be fractured into smaller parts.
Boundary	The boundary box of all GDSII elements in the corresponding structure
PLS Working area	Uses the working area currently defined within the Positionlist
User area	Allows the definition of a new area
Fracturing field size	Select the size of the fragments.
PLS write-field	Uses the writefield stored within the Positionlist
PLS working area	Uses the working area stored within the Positionlist. Please note that when using this option, setting **Fracturing area** to **PLS working area** cuts out the working area and stores the corresponding GDSII elements in a new database.
Design working area	Allows selection of a working area stored with the GDSII
User field	Enables the user to define the field

Control element	Function
PLS layer only	Keeps only the layers stored within the Positionlist on the fractured GDSII
All positions, Selected positions	Click on either **All positions** or **Selected positions** to apply the **Matrix Copy.**
Apply, Cancel	To confirm and execute the settings for the **GDSII fraction**, or click on the **Cancel** button to exit without saving the settings.

1.3.4 Find Text

You may call up the dialog by clicking in the **Positionlist** tool bar.

The **Find Text** dialog enables the user to search for specific text or a string of commands. This is particularly useful when complex Positionlists are being used.

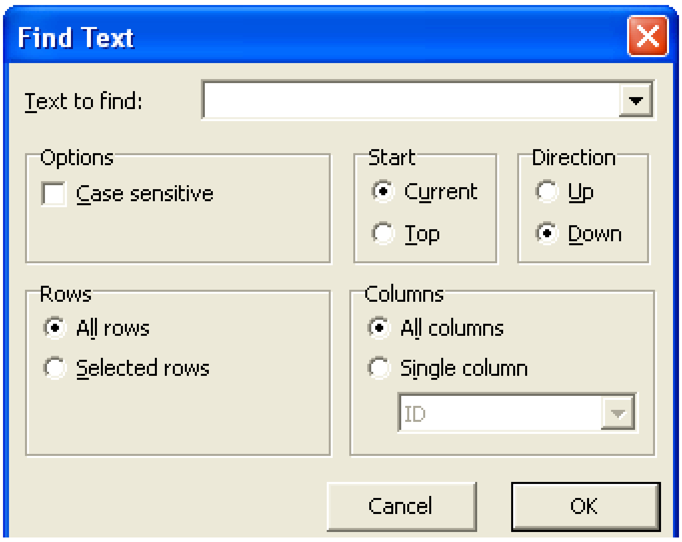


Figure 9-9: **Find Text** dialog

Control element	Function
Text to find	Enter the text to search for.
Options, Case sensitive	To perform a case-sensitive search
Start Current/Top	Select the search to either start from the current position or from the top of the Positionlist.

Control element	Function
Direction Up/ Down	Select the search direction, upwards or downwards.
Rows	Select either **All rows** or only **Selected rows** for the search.
Columns	Select either **All columns** or **Single column** for the search. When Single column is selected, any of the columns present in the current Positionlist may be selected from the dropdown list.

1.3.5 Replace Text

The **Replace Text** dialog allows the user to search for and replace specific text or a string of commands. This is particularly useful when complex Positionlists are being used.

Call up the dialog by clicking in the **Positionlist** tool bar.

The dialog shows the same properties as the **Find Text** dialog. For further information, please refer to Find Text (⇒ *Find Text* on page 9-27).

1.4 Context

Clicking on the **right mouse** button enables the following commands:

Context menu item	Function
Open	This command action is dependent on what type of task is connected with the selected position. For example, for Images, the Image window will open, for Linescans the Linescan window will open and for Patterning tasks, the GDSII Viewer will open, with the corresponding structure inside etc.
Edit	Open the **Edit Position** dialog (⇒ *Edit Position* on page 9-33)
Properties	The properties, depending on the type selected, will be displayed and can be edited. A new dialog will open.
Scan F9	To execute the selected tasks from the Positionlist

Context menu item	Function
New	A new task will be added to the Positionlist. - **Insert before/after**: To create a new line before or after the currently selected line in the Positionlist. The software will insert a task with a simple UV drive command to the current UV stage position. - **Stage position**: opens **Store New Position** dialog for advanced task definition by the operator (⇒ *Store new position* on page 9-32).
Delete	To delete selected positions
Select all...	To select all positions

The **Property** dialog will open a context-specific dialog, depending on the type of procedure in the Positionlist.
If a procedure from the **Scan Manager** is selected, the **Scan properties** dialog will open. (⇒ *Scan properties subdialog* on page 11-17)
If a GDSII procedure is selected, the **Patterning Properties** dialog will open (⇒ *Modify menu* on page 5-101).

1.5 Main menu

It is possible to handle more than one Positionlist on the desktop. If more than one Positionlist has been opened, all commands are only applied to the active one.

1.5.1 File menu

The File menu items have the following functions:

File menu item	**Function**
Close	To close the current Positionlist
Close all	To close all open Positionlists
Open	To load a Positionlist into the current Positionlist window. A dialog box allows the format to be kept, i.e. all list properties, of the already opened list. Answering **No** uses the format for the newly opened list. This command is different to **File➡Open Positionlist** which adds another Positionlist window to the desktop.
Save	To save the active Positionlist under its current name
Save as...	To save the active Positionlist under a freely selectable name
Open assigned Wafermap...	To open the Assigned Wafermap. See section **Assigned Wafermap** for further details (⇒ *Assigning a Wafermap* on page 9-7).
Open PLS-Preprocessor...	To open the Positionlist preprocessor window. See section **Positionlist Preprocessor** for further details (⇒ *PLS-preprocessor* on page 9-21).
New Positionlist	To open an empty Positionlist window
Open Positionlist...	To open a previously saved Positionlist

1.5.2 Edit menu

The **Edit** menu items have the following functions:

Edit menu item	**Function**
Cut	To cut out the currently selected lines and copy them to the clipboard
Copy	To copy the currently selected lines to the clipboard
Paste	To paste lines from the clipboard to the Positionlist before the currently selected line
Delete	To delete the currently selected lines
Select all	To select all lines of a Positionlist
New	New lines
Insert before	To insert a new line before the currently selected line. The new line contains only the current stage position, the default template as well as the default attributes.
Insert after	To insert a new line after the currently selected line. The new line contains only the current stage position, the default template as well as the default attributes.
Stage position...	To open the dialog **Store new position**, which already contains the current stage position, the default template as well as the default attributes. After editing the position properties and clicking **OK**, a new line is appended at the end of the Positionlist. Press **Cancel** to quit the dialog without adding a new position. The stage position can be entered, either in UV or XY absolute or relative. For example, template UV refers to absolute UV drive and dUV refers to relative drive. If no drive is required, choose Option = **STAY**.
Current	To open a window for editing of the selected line.
Line number\ID...	To select a line by using the line number or its ID. This command is useful to avoid scrolling through extremely long Positionlists.
Attributes	Each type of procedure has its own attribute. Choose this command to add attributes to or remove attributes from all lines or selected lines only.

Edit menu item	**Function**
List properties	To open a **Positionlist Properties** dialog to modify the properties of the columns and to select the assigned **Wafermap**. (⇒ *Assigning a Wafermap* on page 9-7).
Sample information...	To open the **Sample Information** dialog. You can enter the **Slot ID**, **Wafer ID** and **Slot** which will be displayed in the Information bar at the bottom of the Positionlist. (⇒ *Positionlist window* on page 9-2).

Store new position

The **Store new position** dialog can be called up via the menu **Edit➡New➡Stage position.**

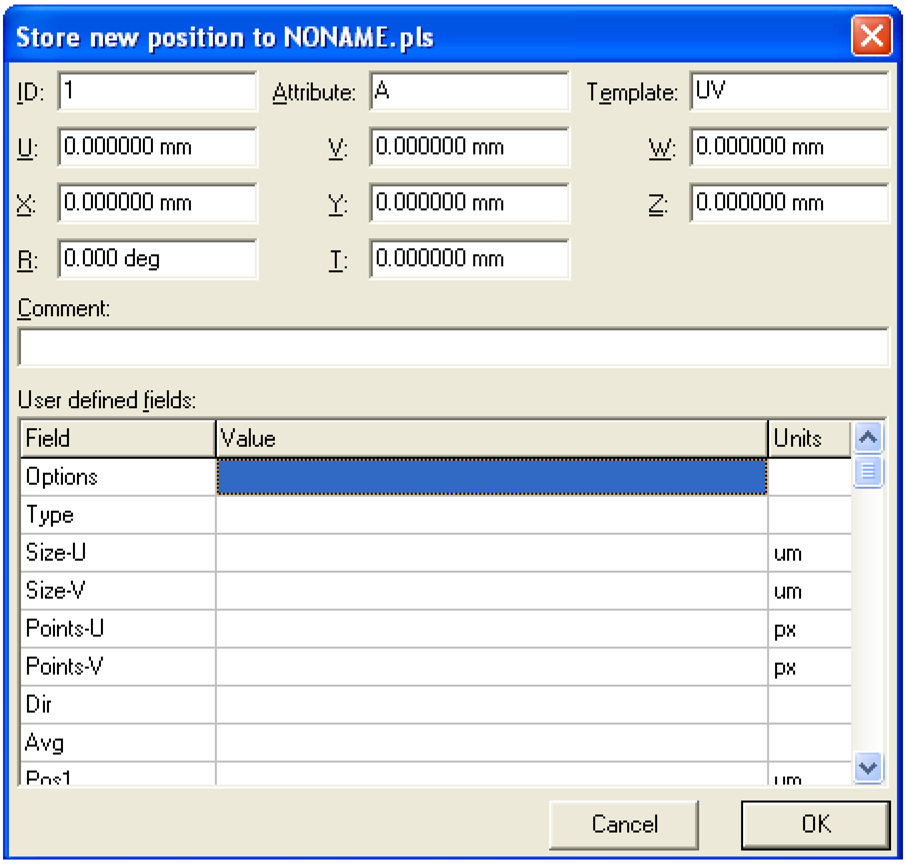


Figure 9-10: **Store new position** dialog

Control element	Function
ID	An **ID** can be entered.
Attribute	An **Attribute** can be allocated.
Template	The template can be defined in either **UV** or **XY**, which will set the coordinates system used for moving the stage. The UV or XY values can be entered, but the important factor is to either define the UV or XY coordinate system. Within the template, **R** (rotation) and **T** (tilt) can also be defined. Use prefix **d** for relative stage movements, e.g. **dUV**, **dXY**. Use prefix **e** or **E** for eucentric movements: **eUV**, **eR** or **eT**. It is possible to combine **e** and **d** in single drive template. Use prefix **c** to drive to the UV chip coordinate locally, within the current die, relative to the lower left edge of the chip. Use prefix **C** to drive to the UV chip coordinate relative to the center of the chip. Use prefix **-** to invert coordinates.
U, V, W, X, Y, Z, R, T	All axes can be defined in the template.
Comment	This is not simply a comment, it has a much more significant meaning. For example, when patterning is initiated, the comment will display the GDSII structure name and it will also save it. If, for example, a GDSII database is shown as the file in the text field, the patterning will be executed using the structure name listed in the **Comment** text field. For example, if a **Scan Manager** procedure is used in the Positionlist, the parameter set of the Scan Manager procedure will be displayed in the Comment text field.

Edit Position

The **Edit Position** dialog can be called up via the menu **Edit➡Current.**

This dialog is the same as the **Store New Position** dialog.

Select Position

The **Select Position** dialog can be called up by the menu **Line number\ID.**

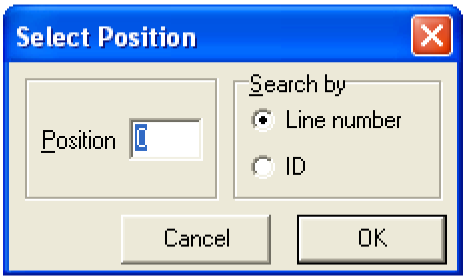


Figure 9-11: **Select Position** dialog

Control element	Function
Position	Enter the position that you wish to select.
Line number / ID	To perform the search via **Line number** or via **ID**

Attributes

The **Attributes** dialog can be called up via the menu **Edit➡Attributes.**

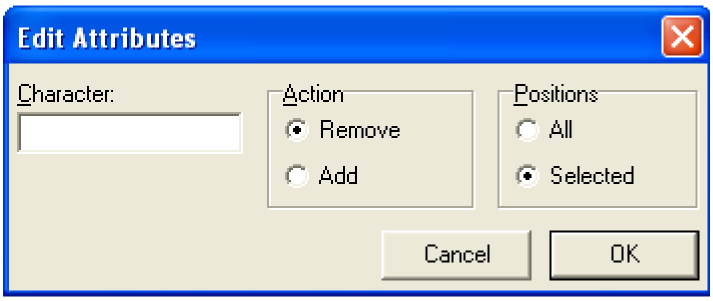


Figure 9-12: **Edit Attributes** dialog

Control element	Function
Character	Enter the **Character** that you wish to edit.
Action Remove/ Add	Select the editing action mode, either **Remove** or **Add**.

Control element	Function
Positions All / Selected	Specify to which **Positions** the modification should apply, either to **All Positions** or to **Selected Positions** only. Please note that the evaluation of the Attributes is case-sensitive and matches only whole words.

Sample Information

The **Sample Information** dialog can be called up via the menu **Edit➡Sample Information**

This is used for wafers. **Lot ID**, **Wafer ID** and **Slot** are defined. This information will be displayed in the status bar of the Positionlist.

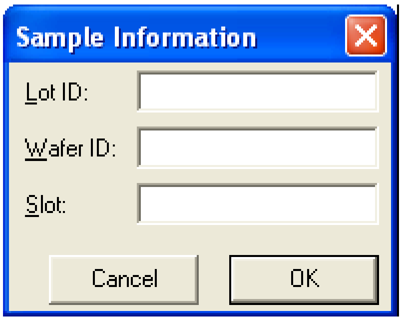


Figure 9-13: **Sample Information** dialog

Control element	Function
Lot ID	To enter the Lot ID for the sample information.
Wafer ID	To enter the Wafer ID for the sample information
Slot	To enter the Slot for the sample information

1.5.3 Search menu

The **Search** menu items have the following functions:

Within a Positionlist, it is possible to search for certain strings as well as to replace them.

Search menu item	**Function**
Find...	To open the **Find Text** dialog, in which the user can define if the search should be case-sensitive or not. The search can be started either from the **Current Selected Line** in the Positionlist or from the **Top** of the Positionlist. The direction of the search can be defined either top to bottom or vice versa. Either **All Rows** or **Selected Rows** can be chosen for the search. The search can also be selected for **All Columns** or a **Single Column**.
Replace...	To open a **Replace Text** dialog. This dialog offers, in addition to the **Find Text** dialog, the ability to replace single or all occurrences of the found string.
Search again	Using this command, the search will be continued after the first search item has been located.

The **Find and Replace** command is a very useful feature when handling large Positionlists.

1.5.4 Filter menu

The **Filter** menu items have the following functions:

Filter menu item	**Function**
Matrix Copy	The Matrix Copy filter serves to easily generate a position matrix consisting of a rectangular array of positions.
Linescan analysis	A Linescan analysis can be applied to previously acquired linescans.
Imagescan analysis	An Imagescan analysis can be applied to previously acquired images.
Line calculation	A line calculation can be performed on previously acquired linescans.

Filter menu item	Function
Long distance	A long distance measurement can be performed on previously acquired linescans.
Statistics	Statistical evaluations can be performed.
Query	This command selects all positions which fulfill the criteria specified by means of the **Positionlist Query** dialog.
Export	Export the content of the Positionlist to text, KLA or Candela file.
Statistics Results	Statistics results can be viewed and stored.
Calculate Patterning Time	The **Patterning Time** of selected patterning tasks can be calculated.

Create Position Matrix

The **Create Position Matrix** dialog can be called up via the menu **Filter➡Matrix Copy.**

The **Matrix Copy** filter serves to easily generate a position matrix, consisting of a rectangular array of positions in the Positionlist. The result of the matrix copy operation is a list of newly-generated lines, which are added automatically.

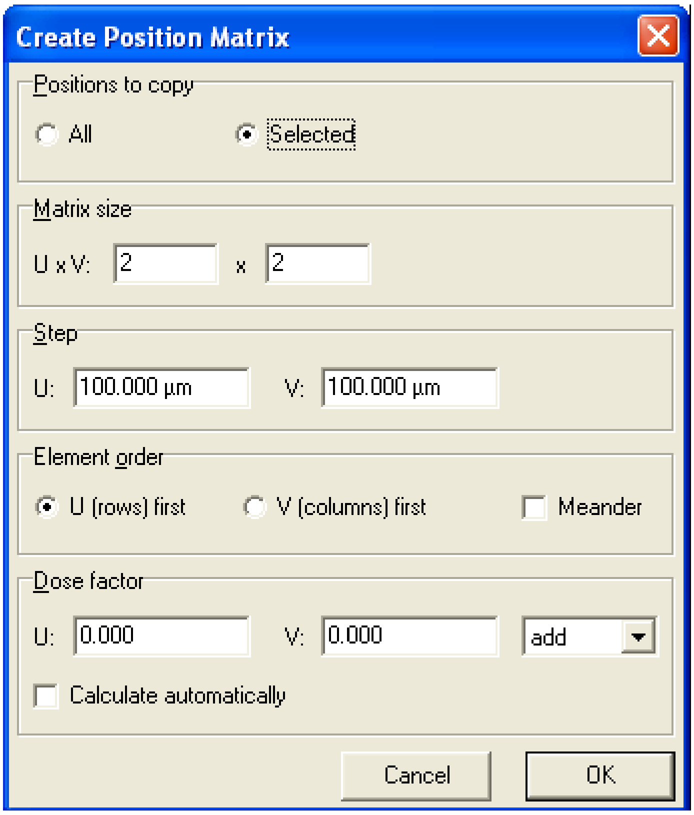


Figure 9-14: **Create Position Matrix** dialog

The command is not available if the active Positionlist is empty.

Control element	Function
Positions to copy	Choose whether all data lines are to be used as the base data lines or only the selected ones.
Matrix Size U x V	Enter the numbers of matrix elements in U and in V directions.
Step U, V	Enter the distances between adjacent matrix positions in U and in V directions.
Element order	Choose the element order from the following options:
U (rows) first	The element order can be row by row. i.e. **U (rows) first**
V (columns) first	The element order can also be column by column, i.e. **V (columns) first**
Meander	If the **Meander** option is active, the positions are ordered and the sequence of positions alternates from row to row or column to column. This optimizes the spatial movement and therefore decreases the stage movement time.
Dose factor	Choose the increase in the patterning dose factor from matrix element to element. This parameter is only valid for patterning tasks.
U, V	The factor can be set for U and V directions independently. The factor can by added to or multiplied by the previous matrix element.
Calculate automatically	By selecting **Calculate automatically**, it is possible to use a single factor for the complete matrix.

Linescan Analysis

The **Linescan Analysis** dialog can be called up via the menu **Filter➡Linescan Analysis.**

This dialog is used to apply one or more Linescan filters to previously scanned Linescans in a Positionlist. Its main use is retrospective data processing and analysis. Those filters which obtain a result will store this result as Pos 1 through Pos 3 in the Positionlist. Be careful with this function, since the old data are overwritten.

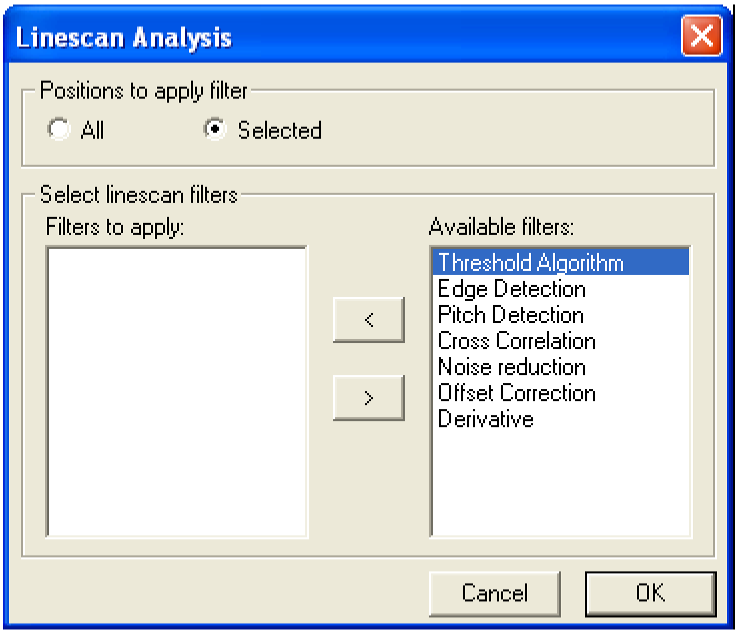


Figure 9-15: **Linescan Analysis** dialog

Within the dialog, choose firstly if the filters should be applied to **All** or only to the **Selected** Linescans. By using the **<** and **>** buttons, it is possible to copy the filters from the list **Available filters** to the list **Filters to apply**. The functionality of the filters is the same is described **Evaluation algorithms**. (⇒ *Calculation information for Threshold algorithm* on page 10-4)

Control element	Function
Positions to apply filter	Select if you wish to apply the filter to all positions or to the selected positions only.
Select linescan filters	A linescan filter can be selected from the database.
Filters to apply	Select one or more filters to apply to the previously acquired linescan.
Available filters	A list of available filters will be displayed.

Imagescan analysis

The **Imagescan** dialog can be called up via the menu **Filter➡Imagescan.**

This command is used to apply one or more Image filters to previously scanned images in a Positionlist. Its main use is retrospective data processing and analysis.

Within the dialog choose firstly if the filters should be applied to **All** or only to the **Selected** positions. By using the **<** and **>** buttons, it is possible to copy the filters from the list **Available filters** to the list **Filters to apply**. The functionality of the filters is the same as described in Threshold Algorithm (⇒ *Calculation information for Threshold algorithm* on page 10-4).

The same parameters as in the **Post Processing** tab within the **Scan Manager** can be selected, but when choosing the Imagescan analysis, the image processing will be performed after the full Positionlist is completed.

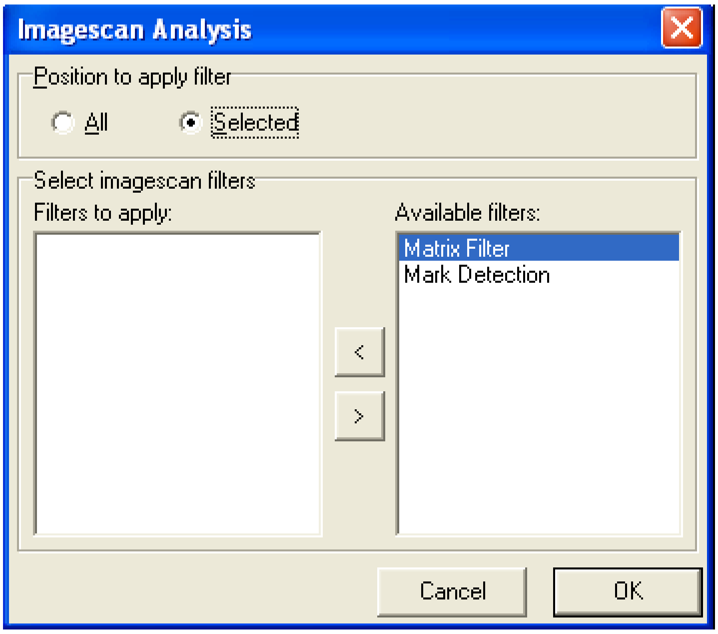


Figure 9-16: **Imagescan Analysis** dialog

Control element	Function
Position to apply filter	Select if you wish to apply the filter to **All** positions or to **Selected** positions only.
Select Imagescan filters	An Imagescan filter can be selected from the database.
Filters to apply	Select one or more filters to apply to the previously acquired Imagescan.
Available filters	A list of available filters will be displayed.

Line calculation

The **Line calculation** dialog can be called up via the menu **Filter➡Line calculation**.

This command is used to change the evaluation results of previously scanned and evaluated linescans. If, for example, a linescan was scanned and a peak detection had been applied to it, the results of the filter are stored in the Pos1 to Pos3 fields in the corresponding PLS line. The **Line Calculation** filter allows the recalculation of the result stored in the Pos3 field, based on data stored in fields Pos 1 and Pos2. Now the filter can determine the width or center position of the linescan profile.

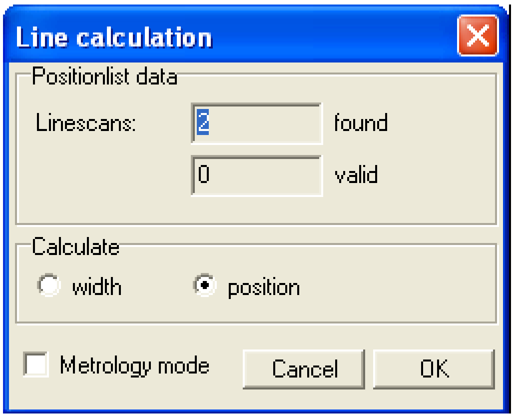


Figure 9-17: **Line calculation** dialog

Control element	Function
Linescans	The linescan can be selected.
found	The number of linescans found will be listed.
valid	The number of valid (evaluated) linescans with Pos1 to Pos3 fields not empty, will be listed.
Calculate	The Calculate parameters can be entered.
width	To calculate the width of the peak or edge
position	To calculate the position of the peak or edge
Metrology mode	To use the scaling previously defined in an **Adjust UV** procedure. Hence, the measurement is given in units of the stage.

Long Distance Measurement

The **Long Distance Measurement** dialog can be called up via the menu **Filter➡Long Distance**.

This command calculates the difference between Pos1, Pos2 or Pos3 of one line and Pos1, Pos2 and Pos3 of another line within a Positionlist. If Pos1, Pos 2, Pos3 are the results of position measurements, the result is the distance between them.

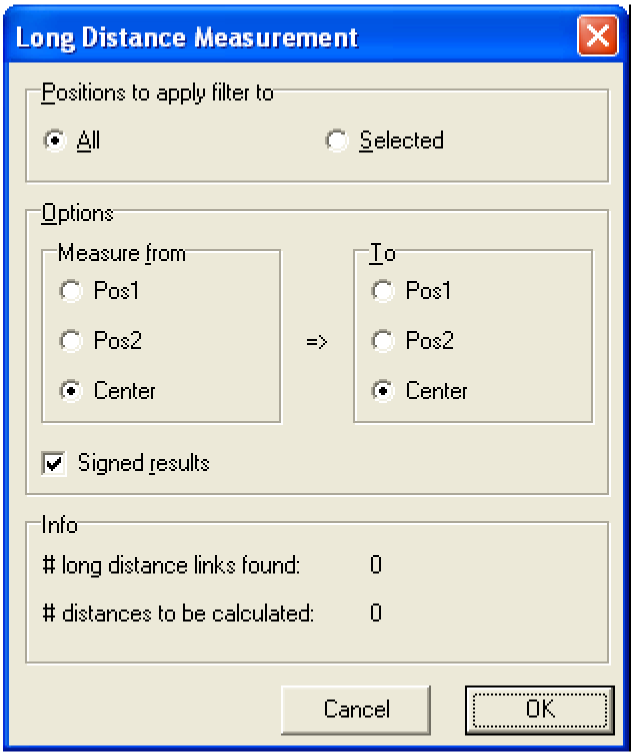


Figure 9-18: **Long Distance Measurement** dialog

Within the dialog, choose firstly if the filter should be applied to **All** or only to **Selected** positions. For distance calculation, the two lines must have a link assigned between them. To establish this link, type into the column **Link** of one of the two lines the ID of the other line. After applying the command, this line now contains the result.

Control element	Function
Positions to apply filter to	Positions can be selected
All, Selected	Select to apply the filter to either **All** or **Selected** positions.
Options Measure from	Options for the starting point can be selected.
Pos1, Pos2, Center	Select either **Pos1**, **Pos2** or the **Center** of a peak or edge as the starting point for the measurement.
To	Options for the end point can be selected.
Pos1, Pos2, Center	Select either **Pos1**, **Pos2** or the **Center** of a peak or edge as the end point for the measurement.
Info	Information can be displayed.

Control element	Function
# long distance links found	Total number of linked Positionlist entries.
# distances to be calculated	Number of linked Positionlist entries required for calculation, containing data in **Pos1** to **Pos3** fields.

Statistics

The **Statistics** dialog can be called up via the menu **Filter➡Statistics.**

This dialog calculates and displays the **mean**, the **minimum**, the **maximum** and the **standard deviation** values for all data within the columns Pos1, Pos2 and Pos3 of a Positionlist.

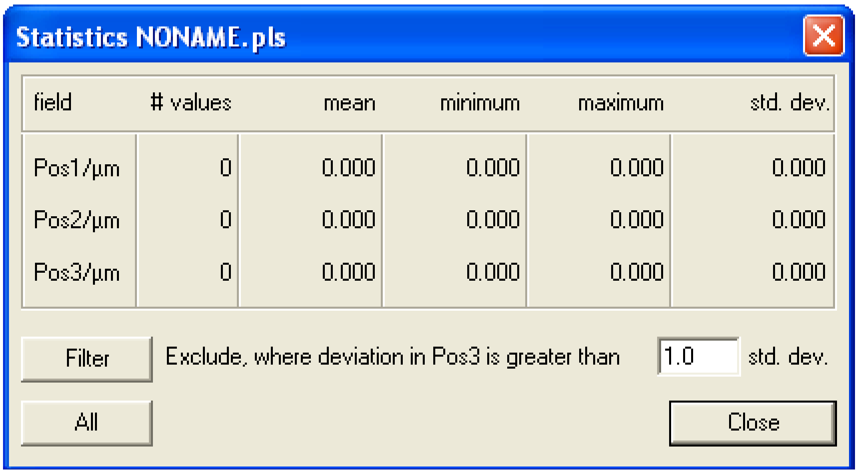


Figure 9-19: **Statistics** dialog

Control element	Function
Pos1, Pos2, Pos3	The values for **Pos1**, **Pos2** and **Pos3** are given, as well as the mean, minimum, maximum values and the standard deviation.
Filter	By using **Filter**, it is possible to exclude positions from being displayed and from use in calculation. To display and to use all, select **All**.

Query

The **Query** dialog can be called up via the menu **Filter➡Query.**

This command selects all positions which fulfill the criteria specified by means of the current dialog. This can be used to filter the Positionlist, which is useful for large Positionlists. It can be used to find certain items quickly, e.g. using the extendable regular expressions editor in this dialog.

Select firstly the name of the column, then decide if the comparison is to be made **as text** or **as number**. Select, in the following, the comparison condition.

In the case of a text comparison, the conditions **<** and **>** assume an alphabetical order. Then type in the data. The comparison can be negated by selecting the **not** option. To build a more complex comparison, press the **insert before** or **insert after** button and connect the sub-conditions by using **and**, **or** and **xor** operators. Each sub-condition can be deleted using the **Delete** button.

Press **Apply** to see the result of the selection. Press **Remove** to neutralize the filtering.

The whole comparison can be saved and recalled for later use, using the **Open** and **Save** buttons.

For example, a query filter could search for linescans with angles greater than 5 degrees and **Options** = **STAY** in the full Positionlist. Pressing the **Apply** button will initiate the query filter.

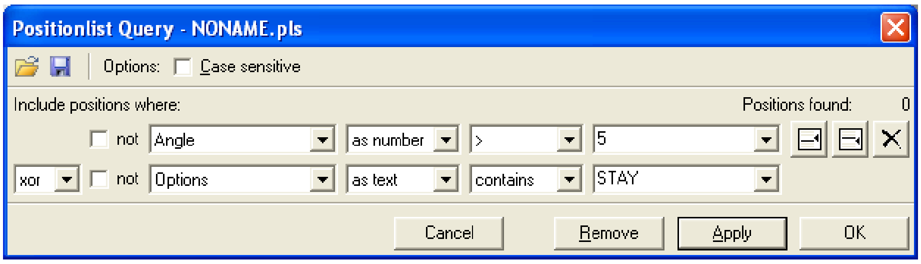


Figure 9-20: **Positionlist Query** dialog

Complex created queries can be created.

Table 9-1: Allocation table for logical operators

Logical operator	Function
and	1 and 0 = 0
	0 and 1 = 0
	0 and 0 = 1
	1 and 1 = 1
or	1 or 0 = 1
	0 or 1 = 1
	0 or 0 = 0
	1 or 1 = 1
xor	1 xor 0 = 1
	0 xor 1 = 1
	0 xor 0 = 0
	1 xor 1 = 0

Export Positionlist Data

The **Export Positionlist Data** dialog can be called up via the menu **Filter➡Export.**

This command provides a universal export for all Positionlists. There are three kinds of data export available:

- configurable text file export,
- configurable export to KLA file,
- configurable export to Candela file.

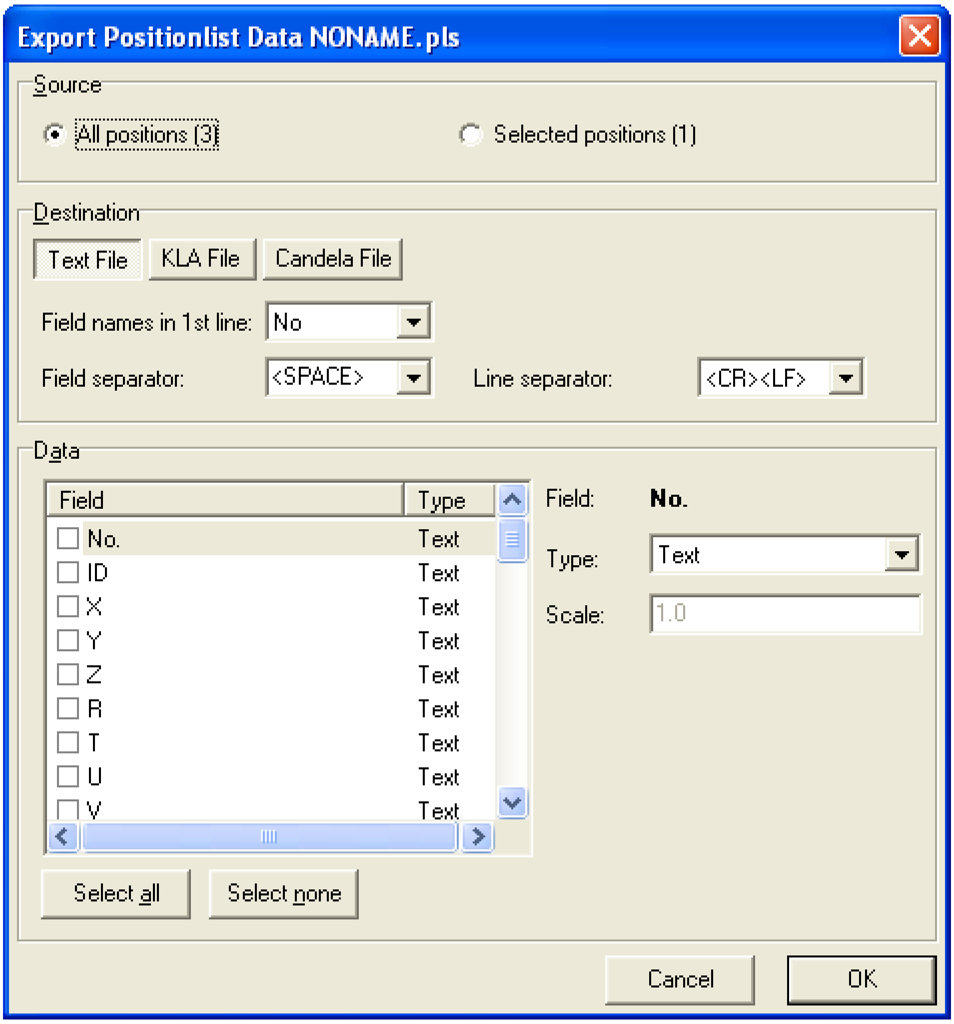


Figure 9-21: **Export Positionlist Data** dialog

Control element	Function
Source	The source can be selected.
All positions (3), Selected positions (1)	Select Source, indicating whether all data lines are to be exported or only the selected ones. The entire number of data lines within the activated Positionlist is indicated in brackets.
Destination	The destination can be selected.
Text file	A text file needs the selection of a field separator. Common field separators are <;>, <,> and <**SPACE**> or if **Field name in 1st line** is selected, the column headers will be added to the target file.
KLA file	Export to a file, which then can be imported into the KLA system or management programs.
Candela file	Export to a file, which then can be imported into the Candela system or management programs.
Data	The data can be selected.
Column listing	Select the columns that you wish to export. To select a column for exporting, check the checkbox in front of its name. Then choose the **Type** depending on the column contents. If selecting the option **Float**, one may enter a scaling factor. If, for example, the external program handles the coordinates as µm instead of mm, the scale factor is 1000. If export to a **KLA File** has been chosen, select from the list KLA field, the column which is used within the KLA software and which corresponds to the active Positionlist column.

To adapt the output to your regional settings, set the variable **Extras➡Settings➡Positionlist Filter : Export➡Decimal Separator** to the used character.

Statistic Results

The **Statistic Results** dialog can be called up via the menu **Filter➡Statistic Results**

This is useful for quality control. The results of the measurements will be graphically displayed and the data values and statistics will be calculated.

The formula is shown in the dialog box, in this case 3x Standard Deviation and the Mean value are calculated.

These calculations are very important for use in quality control for stitching and overlay processes. After the stitching pattern has been completed, measurements can be carried out via linescans.

The statistic result filter can only be called up from the Positionlist, containing evaluated linescans, which means that the **Pos3** field must contain data. The user can easily see from the graphical display of the statistical results if the results are within the specifications.

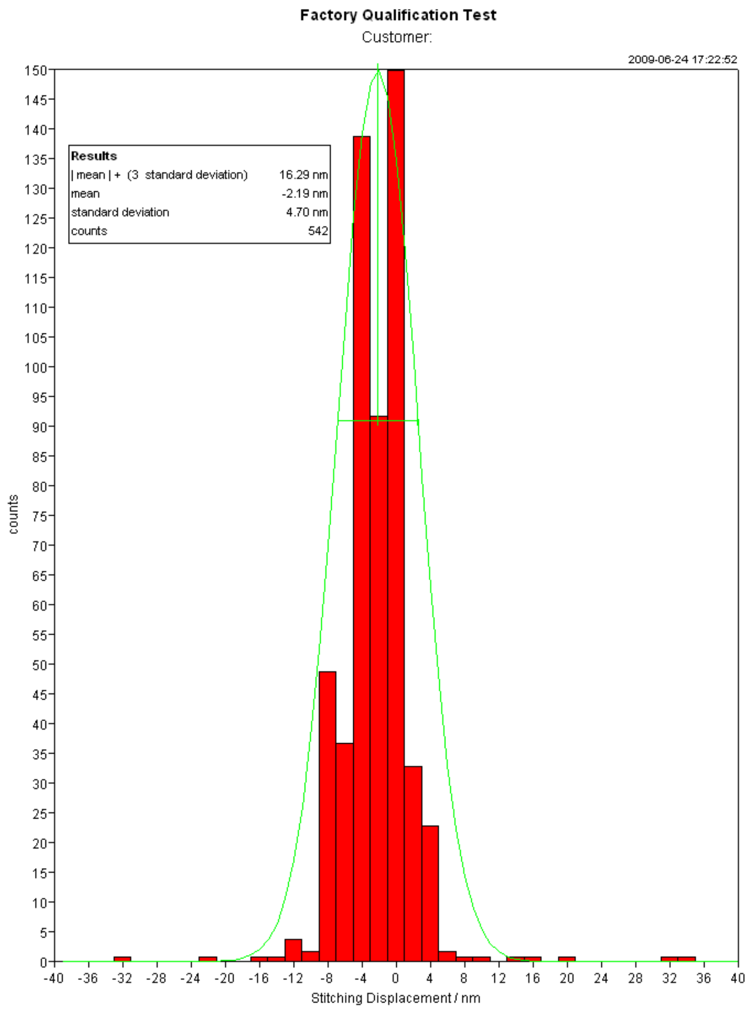


Figure 9-22: Statistic Results

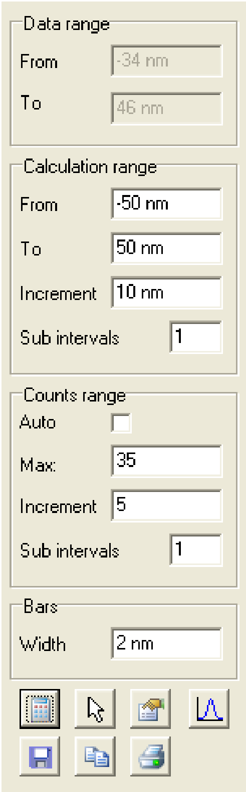
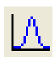


Figure 9-23: Statistic Results Control Elements

Specify the range of the horizontal axis by entering the corresponding numbers within the parameter set **Calculation range**. Specify the range of the vertical axis by entering the numbers under the **Counts range**. Using bars, one can set the width of the count interval.

Control element	Function
Data range	The data range can be selected.
From, To	The range of the data is automatically displayed.
Calculate range	The range can be selected.
From, To	The user can set the range for performance of the calculation.
Increment	Enter a value for the increment.
Sub intervals	Enter the number of sub-intervals for the selected increment.
Counts range	The counts range can be selected.

Control element	Function
Auto	The vertical axis will be scaled automatically corresponding to the total number of data sets.
Max:	The maximum displayed value for the vertical axis can be set manually.
Increment	Enter a value for the increment.
Sub intervals	Enter the number of sub-intervals for the selected increment.
	To update the **Statistical Results** by applying the current parameters. The graph will be updated on the basis of the newly calculated values.
	To move the **Results** box to a different position on the graph. The cursor will change into a hand symbol and the Results box can now be moved.
	To open the **Edit Parameter** dialog, in which the parameter settings can be changed.
	To toggle the **Results** graph **On/Off**. A gaussian distribution will be overlaid on the graph.
	To save the graph as a **<*.bmp>** bitmap file
	To copy the graph into the clipboard, in order to paste it into a different software application, such as an MS Word document.
	To print the graph

The graphic displays the following:

Y axis = number of linescans (counts)

X axis = center position value of the linescan peak stored in the **Pos3** field in the Positionlist.

The distribution of the results will be shown in the graph. The graph can be saved e.g. as a <*.bmp> file or copied to the clipboard.

Statistic Results subdialogs

You may call up the dialog by clicking the  button in the **Statistic Results** dialog.

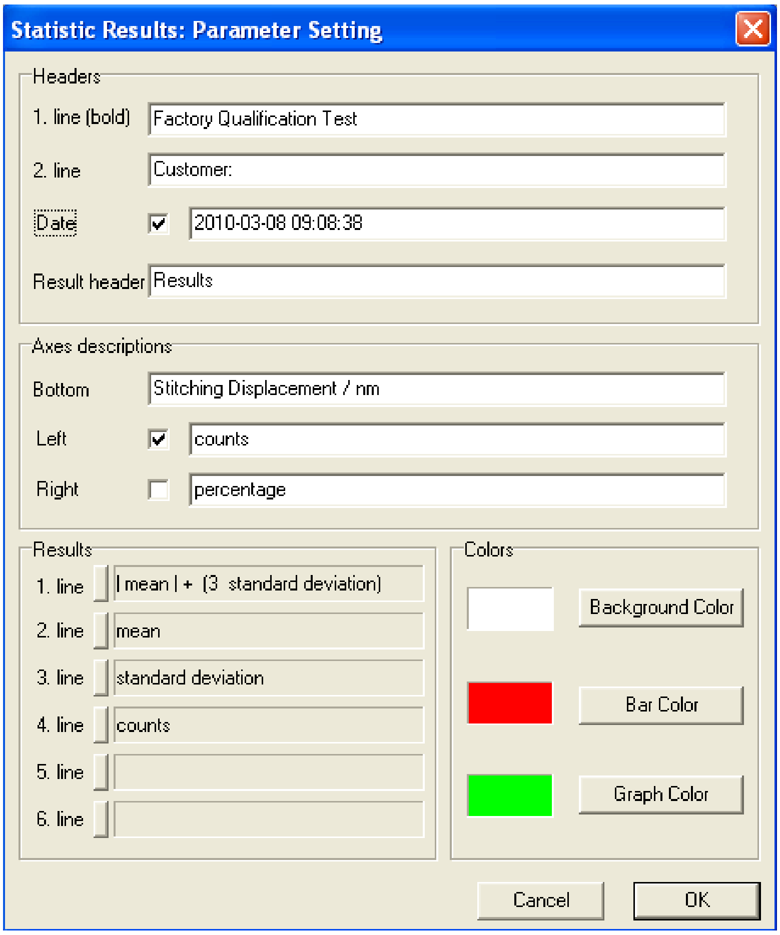


Figure 9-24: **Statistic Results**: **Parameter Setting** dialog

Control element	Function
Headers	The following information can be entered in the **Results** box:
1. line (bold)	To enter the header for the Results box
2. line (bold)	To enter further information
Date	To enter the date. If unchecked, no date will be displayed.
Result header	To enter the header for the statistic results frame
Axes descriptions	The axis descriptions can be selected as follows:
Bottom	Enter the description for the x-axis.

Control element	Function
Left	Select the description for the left y-axis, which can differ from the right y-axis. The left y-axis can only be used to display the absolute number of the datasets.
Right	To enter the description for the right axis, if required. The right Y axis can only be used for the display of the relative number of datasets in %, relative to the overall number of datasets.
Results	The following Results input fields are available:
Input field 1-6	The Results can be described in the input fields 1-6. Clicking on the bar next to each input field will open the **Results description** dialog.
Colors	Colors can be chosen for the following:
Background Color	To select the color for the background
Bar Color	To select the color for the bars
Graph Color	To select the color for the graph

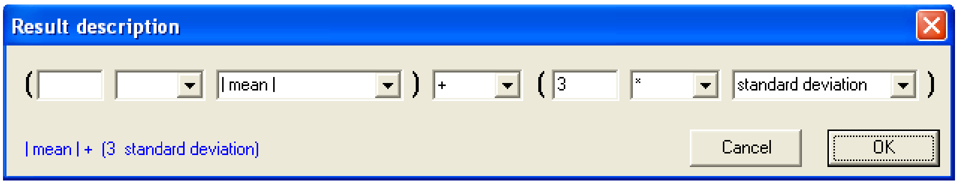


Figure 9-25: **Result description** dialog

Clicking on the bar next to the input field opens the **Result description** dialog, in which a formula can be entered. If one or more input fields are left empty, these fields will not be applied. Each input field has a dropdown list, from which the user may choose the appropriate function or value. The counts, mean, standard deviation etc. can be displayed. The Results formula can be as simple or as complex as the application requires, depending on how many input fields in the formula are selected.

For example, it is possible to simply select **Mean** in one input field and to leave all other input fields empty. In this case, only the mean data will be displayed in the statistical results.

Up to six input fields are available to create the formula. Click on the button next to each input field to display the dropdown list.

Control element	Function
1st input field	In the first input field you may enter a number.
2nd input field	The user may enter a floating point number or integer. Choose from **+**, **-**, **x** and **/**.
3rd input field	The user can enter a simple mathematical application. Choose from **mean**, **| mean |**, **standard deviation**, **counts** and **mean deviation**.
4th input field	The user may enter a floating point number or an integer. Choose from **+**, **-**, **x** and **/**.
5th input field	Enter a pre-coefficient with mathematical application. The term links it to the input field.
6th input field	The results of the formula are displayed. Choose from **+**, **-**, **x** and **/**.
7th input field	Choose from **mean**, **| mean |**, **standard deviation**, **counts** and **mean deviation**.

Estimate Patterning Time

The **Estimate Patterning Time** dialog can be called up via the menu **Filter➡Calculate Patterning Time.**

The **Patterning Properties** dialog will perform the time calculation, but this is only valid for the selected procedure in a Positionlist.

If the user wishes to calculate the patterning time for several tasks within a Positionlist or for a full Positionlist, this can be calculated by the **Calculate Patterning Time** filter.

Either some tasks within a Positionlist or the full Positionlist can be selected for the calculation.

The software will first calculate the patterning time for the individual procedures and then it will add the individual times to obtain the overall time.

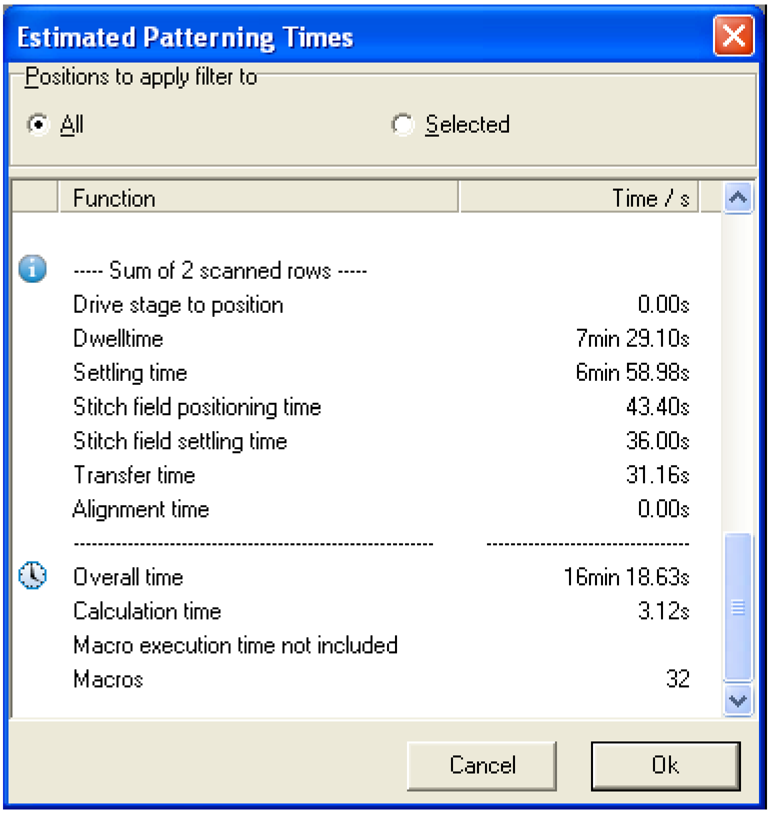


Figure 9-26: **Estimate Patterning Time** dialog

Control element	Function
Positions to apply filter to	The positions can be selected
All, Selected	Select the positions to which you wish to apply the filter, either **All** positions or **Selected** positions.
Function, Time	The results for the function and time will be displayed.

Image to GDSII

When a Positionlist with scanned images is selected, the menu **Filter➡Image to GDSII** will open the following dialog.

This **Image to GDSII** menu item is only displayed if an Image is placed in the Positionlist.

Three different options are available:

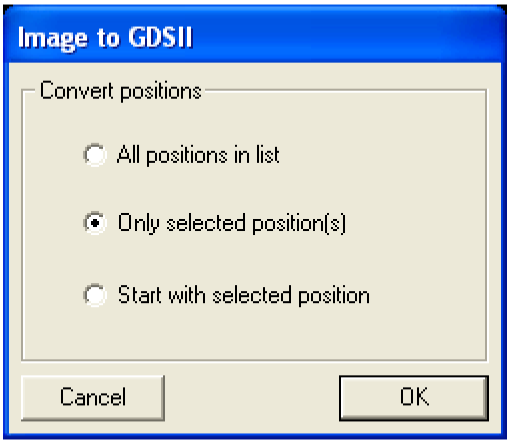


Figure 9-27: **Convert positions** dialog

Control element	Function
All positions in list	All positions will be used (without exclusion).
Only selected position(s)	Only the selected positions will be used.
Start with selected position	All positions will be used (beginning with the selected ones).

After choosing one of the three options, the main menu of the **Image to GDSII** dialog will be displayed, together with the image of the first scan. If an equivalent GDSII database already exists, a pop-up window will ask for replacement confirmation.

Now the conversion parameters can be more closely defined in the following window:

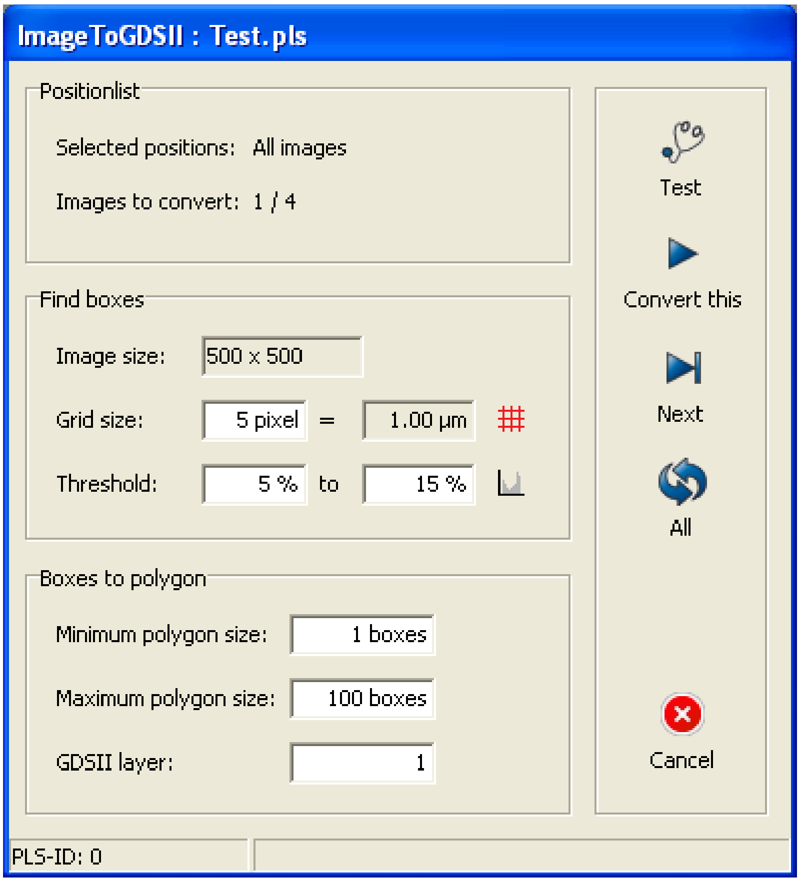


Figure 9-28: **Image to GDSII** dialog

Control element	Function
Positionlist	Shows if either **All positions in list** or **Only selected position(s)** have been selected to be converted. The number of images to convert is also displayed.
Find boxes subset:	
Image size	The size of the selected image is displayed, e.g. 1000 x 1000 pixel.
Grid size	The grid size can be defined in terms of how many microns equals one pixel.
	The grid can be toggled **On/Off**.
Threshold	The value for the minimum and maximum threshold in % can be entered.
	To show a histogram of the distribution of gray scale values in the image

Control element	Function
Find boxes subset:	
Minimum polygon size	**Minimum polygon size** defines the minimum number of connected boxes that will be regarded as a polygon. The polygons will then be stored in the chosen **GDSII-layer**.
Maximum poly-gon size	**Maximum polygon size** defines the maximum number of connected boxes that will be regarded as a polygon. The polygons will then be stored in the chosen **GDSII-layer.**
GDSII layer	The GDSII layer can be defined.
	To initiate a **Image to GDSII** conversion test for checking the parameter settings
	To convert the selected image
	To convert the next image
	To convert all images
	To cancel the conversion

Image to GDSII - Histogram subdialog

When moving the cursor inside the histogram from left to right, the values will be updated to the current cursor position. The histogram displays the gray color values from 0 to 255, where 0 is black and 255 is white. All gray color values will range from black to white between the range of 0 to 255.

When the cursor, for example, is placed on a pixel with the gray value of 14, the frequency value will show the number of pixels with this particular gray level of 14. The percentage gives the percentage value of all pixels having this value in correlation of the overall pixels in %.

The overall number of pixels depends on the size, e.g. an area of 500 x 500 pixels results in 250,000 pixels.

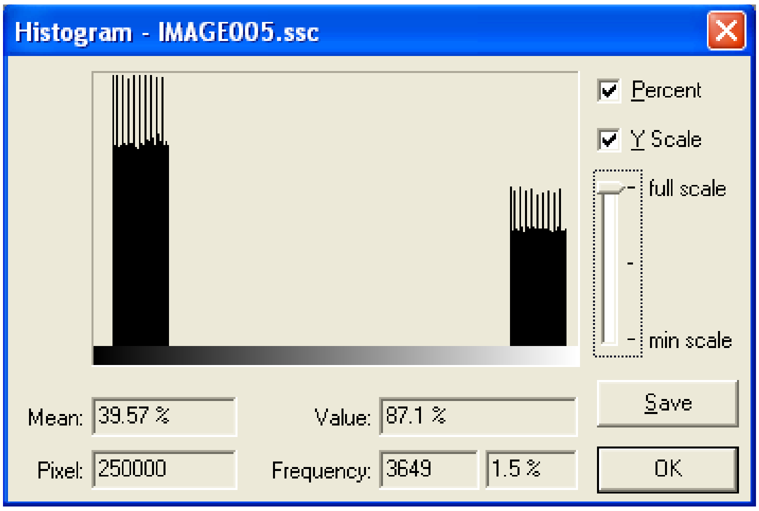


Figure 9-29: Histogram

Control element	Function
Mean	Mean intensity value of all pixels in the image is displayed. The mean value is independent of the current cursor position.
Pixel	The overall pixel value is displayed.
Value	The gray value of the current cursor position is shown.
Frequency	The number of pixels having the selected intensity is displayed in two fields, the absolute number and the number relative to the total number of pixels in the image, expressed as %. Changing the cursor position will update the values for the current position.
Percent	If the checkbox is checked, both **Mean** and **Value** are shown in %.
Y Scale	If the checkbox is checked, the Y Scale will be shown in the graph.
Scale slider bar	The slider bar can be used to adjust the scale between the minimum scale to full scale, between 0 to 100%.
Save	To save the histogram as a <*.**dat**> file
OK	To close the dialog

1.5.5 Scan menu

These commands assist the user in the execution of a list or a task in a Positionlist.

Before scanning, make sure that the required scan options **Drive to position, Scan Manager, Patterning, Execute Macro** are activated, indicated by checkmarks. The Positionlist automatically differentiates the options required to execute a selected task. This differentiation is performed by interpreting the Positionlist column Type.

The menu items have the following functions:

Scan menu item	Function
Cancel scan	This will stop the scan in progress.
Selection	To scan the selected positions. If no data line is selected, this command has no effect. If a group of lines is selected, only these lines are executed.
Next	To scan the next position after the currently selected one. If no data line is selected, this command has no effect.
Previous	To scan the previous position before the currently selected one. If no data line is selected, this command has no effect.
Line number/ID	To select a specific data line for scanning. Within the additional dialog, choose the **Line number** or **ID** and enter the number.
All...	To scan once through all positions within the Positionlist, starting from the first data line and ending with the last data line.
From current...	To scan once through the positions within the Positionlist, starting from the currently selected position and ending with the last data line.
Endless...	To continuously loop through all positions within the Positionlist, starting from the currently selected position. Choose the command **Cancel Scan** or the **ESC** key to cancel the scan, otherwise the Positionlist will be scanned again from top to bottom and so on. This command is usually used for testing purposes.

Scan menu item	Function
Drive to position	To drive the stage to each selected position. If this option is not in effect, the stage will not drive to the positions. Normally this option should be switched on. A Positionlist can be created for positions only. If **Scan endless** is selected, the stage will drive endlessly from position to position.
Scan Manager	To execute adjustment and alignment procedures as well as Imagescans and Linescans. Normally this option should be switched on. When this option is unchecked, none of the Scan Manager procedures, such as alignment procedures, Linescans, image scans etc. can be executed.
Patterning	To enable the execution of Patterning steps of GDSII structures. The name of the GDSII structure is listed in the column **Comment**. The GDSII file is listed in the column **File**, and their destinations are defined as sample coordinates **U**, **V** within the Positionlist. In addition, the position can store information about a dose factor, step sizes, dwell times, the working area and layers. These parameters are described in more detail in the section **Patterning**. When this option is unchecked, the patterning will not be initiated. No GDSII structure can be written, the next procedure in the Positionlist would be executed.
Execute Macro	To enable the execution of macros and scripts The name of the macro to be executed is listed in the column **File** within the Positionlist. When this option is unchecked, the macros will not be executed.
Column Control	Choose this option for Column Control. When this option is unchecked, none of the Column Control commands will be executed.

Unchecking certain types, such as **Drive to Position**, **Scan Manager**, **Patterning**, **Execute Macro** or **Column Control** will exclude those types from execution. This is therefore an additional filter method in order to exclude certain types from a Positionlist to be executed.

While executing a Positionlist, it can be difficult to reach the menu command **Cancel scan**. Press the **ESC** key instead.

1.6 Mouse and keyboard commands

The usage is very similar to other spread sheet control elements. A task can be selected by mouse click. The selected task is marked by highlighting. The selection of groups or multiple lines is possible by applying the standard Microsoft Windows operations using the **Shift** and **Control** keys respectively. All tasks can be selected using **Ctrl A**.

A double click within a task opens an object which is linked to this position. This can be a GDSII design within the GDSII viewer, an image which was taken at this position, or in the case of an alignment or adjustment task, another Positionlist.

To open the corresponding object, the software stores a link to this object in the data field **File**, e.g. filename and path of a GDSII. If the software can not find this object, a simple dialog opens to edit the data fields. This can be used to test if the link to the object is broken.

The **right mouse** button opens a menu, in which additional commands are available.

Whenever a data line is selected and if a corresponding UV window has been opened in parallel, a related flashing marker will be highlighted, flashing three times. This enables fast orientation to the location of an object of interest.

The table below gives a summary of all available mouse commands within a Positionlist window.

Table 9-2: Mouse and Keyboard commands

Key and mouse	Functionality
none+left(single)	Select a line and unselect all other lines.
Control+left(single)	Select or deselect a line without deselecting all other lines.
Shift+left(single)	Select all lines from the first selected one to the latest selected one and unselect all other lines.
none+right(single)	Open a context menu with the commands: **Open**, **Edit**, **Properties**, **Scan**, **New**, **Delete** and **Select all.**
none+left(double)	Open a window related to the type of the task, e.g. an Imagescan, a Linescan, a GDSII viewer.

1.7 Drag and drop

A line can be copied or deleted by the drag and drop technique:

- To copy a line to another Positionlist, select it, move it to the other Positionlist and release the mouse button.
- To copy a line within the same Positionlist, select it, move it to its new location by pressing the **Ctrl** key.
- To move a line within the same Positionlist, select it and move it to its new location.
- To delete a line, drop it anywhere else within the program desktop.
- A task can also be relocated using the drag and drop technique by picking it up in the list and dropping it into any UV window.

A summary of drag and drop operations is listed below:

Table 9-3: Drag and drop

Source	Object	Target	Action
Positionlist	line	Positionlist	Copy lines
Positionlist	line	Image	Redefine the location of a line using Image coordinates of mouse cursor as new location.
Positionlist	line	Wafermap	Redefine the location of a line using Wafermap coordinates of mouse cursor as new location.
Positionlist	line	Program desktop	Delete a line
Scan Manager	procedure or scan	Positionlist	Generate a position with an adjustment or alignment task or scan using current UV location.
GDSII database	structure	Positionlist	Generate a position with a Patterning task using current UV location.
Automation	file	Positionlist	Generate a position with an automation task with no stage movement, i.e. Options=**STAY**.

Table 9-3: Drag and drop

Source	**Object**	**Target**	**Action**
Writefield Manager	parameter set	Positionlist	Generate a position setting with selected writefield with no stage movement, i.e. Options=**STAY.**
Column Control	parameter set	Positionlist	Generate a position setting with selected column parameters with no stage movement, i.e. Options=**STAY.**
Positionlist	line	Adjust UVW 3-Points Tab	Assign the position of a selected line to the chosen adjustment mark.

1.8 Configuration

Using the command **Edit➡List properties**, the Positionlist format can be configured for various applications. The default configurations within the Raith software are designed to allow all standard tasks such as adjustment and alignment procedures, Patterning runs and automation jobs. If the drag and drop technique is used, all relevant parameters are transferred automatically into the Positionlist. A reconfiguration is not necessary, but to customize the functionality of a Positionlist or for debugging purposes, it is essential to know the functionality of each column and of the stored data.

For a general overview, the relevant Positionlist columns are sorted by the tasks that they are reserved for.

1.8.1 Universal columns

Column	Function
No.	Serial numbers are set automatically by the software. They are used for addressing commands and to select a position.
ID	Identification numbers can be set by the operator. They are used for addressing commands and to select a position.
X Y Z	Cartesian stage coordinates, generally **X**, **Y**, in the horizontal direction and **Z** in the vertical direction. Values are given in mm.
U V W	Cartesian sample coordinates, generally **U**, **V**, in the horizontal direction and **W** in the vertical direction. Values are given in mm.
R T	Rotation and tilt stage coordinates. Values are given in degrees.
Attribute	A combination of letters is used to assign and identify the task and status of the position. **X** identifies a Patterning procedure. **W** identifies an adjustment or alignment procedure with images. **I** identifies an Imagescan. **L** identifies a Linescan or alignment procedure with Linescans. **N** identifies a newly created position. **S** identifies a successfully scanned position. **E** identifies an error. The attributes controlling the status (**N**, **S**, **E**) have a color coded line indicator: a blue dot for **N**, a green dot for **S**, a red dot for **E**.
Template	Defines how the addressing of the stage is done. Use the letter of a coordinate to use the corresponding axis. Use a **d** in front to perform a relative drive command. For example, for lithography tasks, **UV** are almost always used.
Comment	This column is automatically filled with additional information which is specific to one task. For example, in the case of a Patterning task, it identifies the GDSII structure.

Column	Function
Options	**STAY** or **DRIVE** to prevent or enforce a stage movement.
Type	The Type is internally used to identify and differentiate between the various tasks. The type is automatically generated. The Raith NANOSUITE software uses the following types: EBEAMSCAN, SLOWSCAN, MACRO, EXPOSURE, ALWF_MANUAL, ALWF_AUTO, ALWF_AUTOLINE, ADUV_MANUAL, ADUV_AUTO, ADUV_AUTOLINE, MARKSCAN, AUTOSCAN, AUTOMARKSCAN, RMARKSCAN, RAUTOSCAN, RAUTOMARKSCAN, CALIBSCAN, VIDEOSCAN, REFMARKSCAN, GDSII_MAN, GDSII_AUTO, PEIZOSCAN, PATH_MAN, PATH_AUTO, PATH_AUTOL.

1.8.2 Linescans and Imagescans

Column	Function
U V	Sample coordinates of the center of the scan.
Attribute	Must be **L** in the case of a Linescan and **I** in the case of an Imagescan.
Comment	Name of the scan which is executed; must be defined in the **Scan Manager**.
Options	**STAY** or **DRIVE** to prevent or enforce a stage movement.
Size-U, Size-V	Defines the scan length and width, e.g. of a mark, in sample coordinates. Data are given in microns.
Points-U, Points-V	Number of points, which can be defined freely up to 4096 points.
Dir	Scanning direction of a Linescan in sample coordinates, i.e. U or V.
Avg	Number used for point-by-point averaging.
Lavg	Number used for line averaging.
Frame	Number used for frame averaging.

Column	Function
Pos1, Pos2, Pos3	This column is used to store the result of a detection process, e.g. of the threshold algorithm. **Pos3** may contain the line width, the center or, if this scan is linked to another one, the distance.
Link	Using this column, two scans can be linked to each other to calculate the distance between them. A link is used to configure long distance measurements between different entry points in the Positionlist. Based on a link between scanned and evaluated entries in the Positionlist, the distance may be either calculated between Pos1, Pos2, or Pos3 or combinations of them. It only makes sense to link Linescans of the same type and orientation. Links can be generated by text editing, graphically on any UV window.
File	Name of the file that contains the scan.
Time	Scan time in seconds since 1/1/1970, i.e. UNIX timestamp.
Timestamp	Readable time when a scan was recorded.
Angle	Angle at which the scan is acquired, relative to the main scanning direction.

1.8.3 Adjustment and alignment procedures

The following types are used in connection with adjustment and alignment procedures: ALWF_MANUAL, ALWF_AUTO, ALWF_AUTOLINE, PATH_MAN, PATH_AUTO, PATH_AUTOL, ADUV_MANUAL, ADUV_MANUAL, ADUV_AUTOLINE.

Control element	Function
U, V	Sample coordinates at which the adjustment and alignment procedure starts.
Attribute	Must be **W** or **L**.
Comment	Name of the scan which is executed; must be defined in the **Scan Manager**.
Options	**STAY** or **DRIVE** to prevent or enforce a stage movement.

Control element	Function
Size-U, Size-V	Defines the scan length and width, e.g. of a mark, in sample coordinates. Data are given in microns.
Dir	Scanning direction, by default set to the U direction.
File	Link to the Positionlist containing the sub-tasks for the procedure.

1.8.4 Patterning runs

The type **EXPOSURE** is used in connection with a Patterning run.

Control element	Function
U, V	Sample coordinates of the center of the first write-field.
Attribute	Must be **X**.
Comment	Name of the structure which is written; has to be defined in the GDSII file.
Options	**STAY** or **DRIVE** to prevent or enforce a stage movement.
Size-U, Size-V	Gives the length and width of the working area in µm, only for information purposes.
Pos1, Pos2	Half of the writefield size in µm.
File	Link to the GDSII file.
Layer	This refers to the layer which will be exposed. The layers are separated by </>, <,> or <;>. A range of layers is indicated by <->.
Area	The working area which will be exposed. The design coordinates are separated by <;>.
Dosefactor	A factor by which the dose is multiplied, i.e. the dwell time, prior to patterning.
Dwelltime	Area dwell time in ms to be used for the Patterning. If this column is empty, the dwell time is taken from the global settings in the **Patterning** module.

Control element	Function
StepSizeU, StepSizeV	Area step size and line spacing in µm to be used for the Patterning
SplDwell	Dwell time in ms for single pixel lines. If this column is empty, the SPL dwell time is taken from the **Patterning** module.
SplStep	Step size, in µm, for single pixel lines. If this column is empty, the SPL step size is taken from the **Patterning** module.
CurveStep	Curve step size in µm to be used for the Patterning
CurveLine	Curve line spacing in µm to be used for the Patterning
CurveDwell	Dwell time in ms for Curved elements
FBMSArea	Stage speed in mm/s used for the patterning of FBMS areas
FBMSLine	Stage speed in mm/s used for the patterning of FBMS lines
MBMSAreaStepSize	Area step size in µm for MBMS base element.
MBMSAreaLineSpacing	Line spacing in µm for MBMS base element.
MBMSLineStepSize	Line step size in µm for MBMS base element.
MBMSDwelltime	Pixel dwell time in ms for MBMS patterning.
MBMSPitchScaling	Scaling factor for the MBMS **Pitch parallel to the path** (please refer to the chapter ⇒ *Adding MBMS paths* on page 5-46 for more details).

1.8.5 Automation task

The type MACRO is used in connection with an automation task.

Control element	Function
U, V	By default set to U=V=0.0 in combination with Template = **dUV** to prevent the stage from moving
Attribute	Must be **M**

Control element	Function
Template	By default set to **dUV** in combination with U=V=0.0, to prevent the stage from moving
Options	**STAY** to prevent stage movement
File	Link to the macro or script file to be executed

Chapter 10
Linescans and Images

This chapter describes the recording and data processing of Linescans and Images. The intensity of, for example, the secondary electron beam, can be recorded and displayed within the Raith NANOSUITE software as a function of position. The user firstly needs to distinguish between two different evaluation types.

- Linescans: The intensity is recorded one-dimensionally. Even if several different cycles covering a particular area are recorded, the results of each scan are summed to yield an intensity curve as a function of position along a straight line.
- Imagescan: The intensity is recorded two-dimensionally. The intensity is given as function of position on the sample over an area. A normal image is acquired.

This chapter explains the following items:

- Background information on Linescans
  (⇒ *Linescans* on page 10-2),
- Detailed description, with formulae, of the Threshold Algorithm
  (⇒ *Calculation information for Threshold algorithm* on page 10-4),
  (⇒ *Setting up a Threshold Algorithm* on page 10-7),
- Explanation of how to perform post processing on a Linescan
  (⇒ *Post processing Linescans* on page 10-7),
- Detailed description of how to drop GDSII structures onto an image
  (⇒ *Drag and drop GDSII structures onto an Image* on page 10-9),
- How to work with MAP and MAW
  (⇒ *Working with MAP and MAW* on page 10-11),
- Detailed description of the Surface Editor
  (⇒ *Surface Editor functionality* on page 10-12),
- Explanation of the software parameters for Linescan and Images
  (⇒ *Linescan window* on page 10-16),
  (⇒ *Image window* on page 10-32),

A Functional Description

1 Linescans

Linescans can be displayed either as Ebeam linescans or Image linescans. The window will display the data acquired and it is possible to apply various **Filters** to the linescans. The linescan window is required when running automatic linescan procedures.

If, for example, a linescan is selected via the **Scan Manager**, the linescan will be acquired and once completed, post processing may be carried out.

Digital **Images** can be acquired and displayed conveniently within the Raith NANOSUITE software by making use of the high resolution scan generator. The primary application of image acquisition for lithography applications is related to mark recognition, which is an essential part of every lithography task.

The image acquisition commands within this mode make use of the high resolution scan generator. The images are acquired with the current scan calibration, i.e. writefield **zoom**, **shift**, and **rotation**. In addition, the images are displayed using UV-coordinates.

The link to the absolute sample coordinate system is derived from the position sensing system of the sample stage. The operator has access to this transformation in the status bar when moving the cursor over the image window. The status bar displays the sample coordinates in UV and the window coordinates. Scaling is indicated by scale bars in the lower left corner of the Image window.

The images are handled as UV windows, allowing addressing, drag and drop and overlay options.

1.1 Linescan window

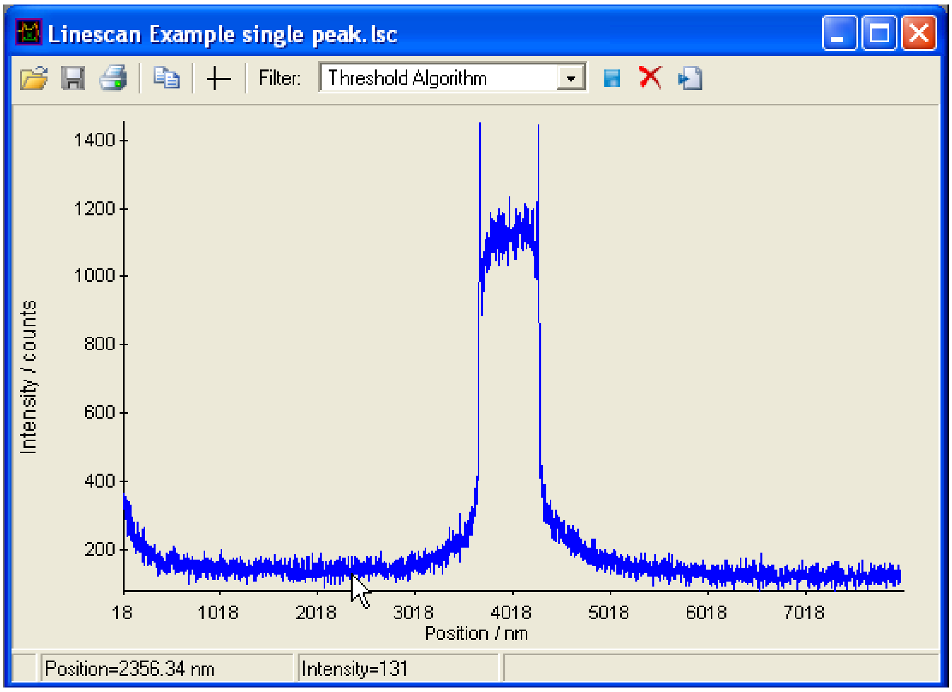


Figure 10-1: **Linescan** window, displaying the intensity as a function of the position

The **Linescan** window opens automatically whenever a **Linescan** has been started, when using the command **File➡Open Linescan**, or by double clicking on a **Scan** procedure within a Positionlist.

The **Image** window opens automatically whenever an **Image** has been started, when using the command **File➡Open Image**, or by double clicking on an **Image** or **Markscan** procedure within a Positionlist.

2 Calculation information for Threshold algorithm

The basis for **Peak Detection**, **Edge Detection**, or **Pitch Detection** is a threshold algorithm. The idea of the threshold algorithm is that in all cases, the intensity has to satisfy two threshold, or magnitude criteria. In the figure below, these two thresholds are indicated by two green lines.

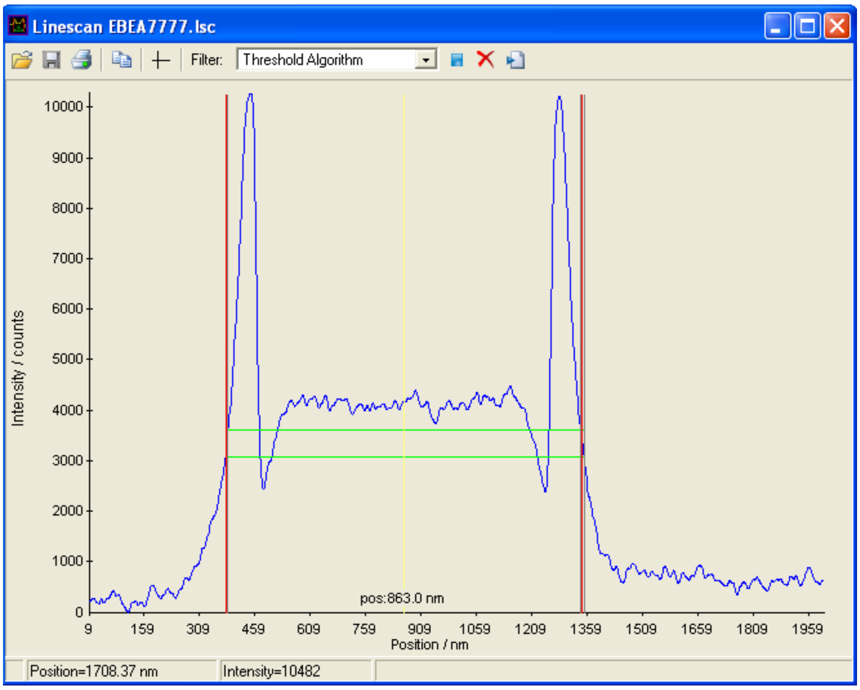


Figure 10-2: Linescan showing threshold levels

The range between the intersections of threshold levels with the intensity slope is called an edge. The boundaries of the two edges are indicated by gray lines. The current position of an edge can be defined between these boundaries. The position is marked by red lines in the figure above.

After detection of two edges, it is then possible to calculate either the distance or the center between them. The distance or the center can be interpreted as the width or as the mid position of a structure on the sample. For example the center of the double-peak structure, in figure above, is marked by a yellow line and its numerical value is also given.

The threshold algorithm can be modified by several parameters. All of these parameters together form a set, which is stored under one name. In all three cases, i.e. for **Peak Detection**, for **Edge Detection**, and for **Pitch Detection**, these parameters are very similar, so they are described together in section B (⇒ *Linescan tool bar* on page 10-17).

2.1 Cross Correlation filter for Linescans

This algorithm compares two linescans.

In the literature, two different algorithms are discussed.

- The approach of a pure cross-correlation has the disadvantage that pre-processing is generally required in order to detect larger shifts, because only small shifts are detected reliably.
- The approach of a pure deconvolution has the disadvantage of being sensitive to noise, such that careful noise-reduction is required beforehand.

In this software, a combined approach is used, having the advantages of both cross-correlation and deconvolution methods. In the following paragraphs, this combined approach is described as far as is required to understand and to use the software.

The starting point is the assumption that the shifted image can be calculated by using the following equation.

"shifted image" = "reference image" $\delta(u\text{-}u_0, v\text{-}v_0)$ + "noise".

The function describes the convolution of two images and $\delta(u\text{-}u_0, v\text{-}v_0)$ is the one-dimensional δ-function, which is shifted by u_0 and v_0. The term "noise" encompasses all sources of noise. This includes the background noise visible in all images acquired by electron or ion optics. The different appearance of the two linescans can also be interpreted as "noise". There is also the noise which comes from periodic boundary values, which occur in calculations. For our purpose a real number representing the magnitude of the noise is sufficient and makes the calculation much easier. The parameter "noise" determines the noise-sensitivity during the search for maxima. Any neighboring peaks where the maximum value between valley and the top of the peak is within the defined maximum noise level, will be ignored.

After calculating the Fourier transformation of the "shifted linescan", as well as the "reference linescan" and by giving the quantity "noise", the algorithm calculates the linescan $\delta(u\text{-}u_0, v\text{-}v_0)$. It is important to understand that the so-calculated δ depends on the value of "noise" and one has to search for the optimum δ by varying the value of "noise". Fortunately, the implemented algorithm does not depend critically on the exact value of "noise" and a value nearby gives a sufficient result. When the optimum δ has been found, u_0 and v_0 are determined by using the coordinates of the maximum intensity in the δ linescan.

When using this detection algorithm automatically, it is very important to distinguish between successful detection of structures and unsuccessful ones. An unsuccessful detection might occur when, for example, a structure is completely missing. In this case, the real noise is much larger than that given by the quantity "noise" and a non-optimum δ is calculated. By using the maximum intensity of δ, in comparison to the variation of the intensity values in δ and the extension of the maximum peak, a quality is calculated. Its value can be between 0.0 and 1.0, with 1.0 corresponding to a perfect δ.

Even if the comparison of two linescans was successful, it might be that a shift is measured which is too large. In this case, it is possible to define, in addition, a maximum permitted shift.

Cross correlation filter results are stored in the Positionlist in the fields Pos1 (shift in µm), Pos 2 (quality) and Pos3 (middle position of corrected scan).

B Tasks

1 Setting up a Threshold Algorithm

The threshold parameters can be accessed from several points within the software. The method described below has the advantage that different parameters can be tried out with a pre-recorded Linescan, which has been opened beforehand.

- Open a pre-recorded Linescan using **File➡Open Linescan.** Some example Linescans come with the software.
- Choose the command **Filter➡Select active Filter** to select one filter type, either **Threshold** filter for detecting a peak or valley, **Edge detection** filter or **Pitch detection** filter.
- Choose the command **Filter➡Apply active Filter** to open the window for setting up the parameters.
- Select the name of an existing parameter set or, if a set of appropriate parameters does not exist, create a new set by choosing the button.

Enter the parameters for threshold levels, the edges and the structure type, to define the measurement.

- After finding a parameter set, press **Apply** to start a measurement.
- Choose **Save** to store the optimum parameter set under the current name.

Sometimes, it may be difficult to find the correct parameter set for the threshold algorithm, especially when dealing with a new Linescan. In this case it is recommended to scan first one exemplary scan. Open this scan by using **File➡Open Linescan** or by double clicking it in a Positionlist. Start the Threshold algorithm by using the commands **Filter➡Select active Filter** and **Filter➡Apply active Filter.** In this case the **Apply** button is accessible, applying the current parameters to the Linescan. This makes it easy to try out various parameters and, if the right set is found, it can be stored and recalled for later use.

---

2 Post processing Linescans

The following paragraphs describe how a Linescan can be processed, for example to reduce the noise, and how a specific structure can be found, for example, using an intensity maximum.

2.1 Post processing filters

Very often, both recording and processing are required. The setup of these parameters is described below. It is important to remember that these filters modify the data itself. Thus, it is not possible to switch back to the unmodified data after applying one of these filters.

Active filter	Function
Noise reduction	Choose this filter to perform numerical averaging after recording the data. Two parameters define the averaging. **Number of average points** is used to define the number of points along the scan direction over which the average is calculated. **Number of iterations** defines how many times the averaging is done.
Offset correction	Choose this filter to subtract a background. The calculation is done in such a way that the minimum value of the intensity is set to zero.
Derivative	Choose this filter to calculate the derivation of the intensity signal.

2.2 Finding structures

After recording and optionally processing, Linescans can be evaluated also for line width, position and distance measurements. It is important to know that these filters do not modify the data itself. These filters store only the result of the evaluation.

Active filter	Function
Check amplitude	Choose this filter to verify that the intensity is above a certain level. A typical application would be to search for a structure and to identify those locations on the sample at which the structure is missing.
Peak detection	Choose this filter to identify a peak structure within the recorded data, for example to determine its width. Select first the **Edit** button to set the parameters for the peak detection algorithm. A detailed description of the parameters is given in the section **Threshold algorithm**. After selecting these parameters, activate the option and define if the position or the width should be stored.

Active filter	Function
Edge detection	Choose this filter to search for an edge within the recorded data, for example to determine its position. Select first the **Edit** button to set the parameters for the edge detection algorithm. A detailed description of the parameters is given in section **Threshold algorithm**. After selecting these parameters, activate the option and define if the position or the width should be stored.
Pitch detection	Choose this filter to search for an edge within the recorded data, for example to determine the distance between two edges. Select first the **Edit** button to set the parameters for the pitch detection algorithm. A detailed description of the parameters is given in section **Threshold algorithm** (⇒ *Threshold Algorithm* on page 10-19). After selecting these parameters, activate the option and define if the position or the width should be stored.
Cross correlation	Choose this filter to compare the measured intensity with a previously recorded intensity. This option is very helpful when the irregular shape of a structure prevents the operator from using another algorithm. Select first the **Edit** button to set the parameters for the cross-correlation algorithm. A detailed description of the parameters is given in the section **Dimensional mark detection.** After selecting these parameters, activate the option. The result of the algorithm can only be a position. Therefore, the **Check detected width** option is disabled when selecting this option.
Check detected width	If a structure with a certain width was scanned, this filter can be used to verify the width. This option is helpful to identify which structures are either too small or too large.

3 Drag and drop GDSII structures onto an Image

GDSII data can be transferred to the image via a drag and drop command.

- Select a GDSII design structure, then hold down the **Alt** key and drag and drop the GDSII structure into the image.
- You will be prompted which layer to select.

The functionality is sufficient for this purpose. It is often used for milling applications, since the GDSII structure can be moved and placed on the image. If some structures already exist, and are shown on the image. The GDSII pattern can now be placed exactly onto the image prior to patterning. As soon as the patterning is completed, another imagescan can be performed to check the progress of the milling.

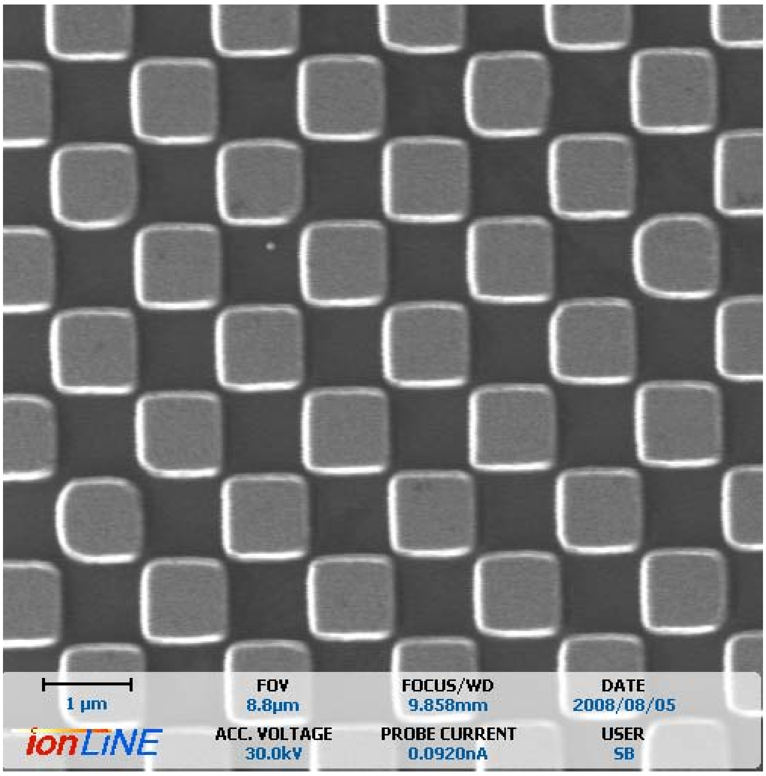


Figure 10-3: Image before placing GDSII structure onto it

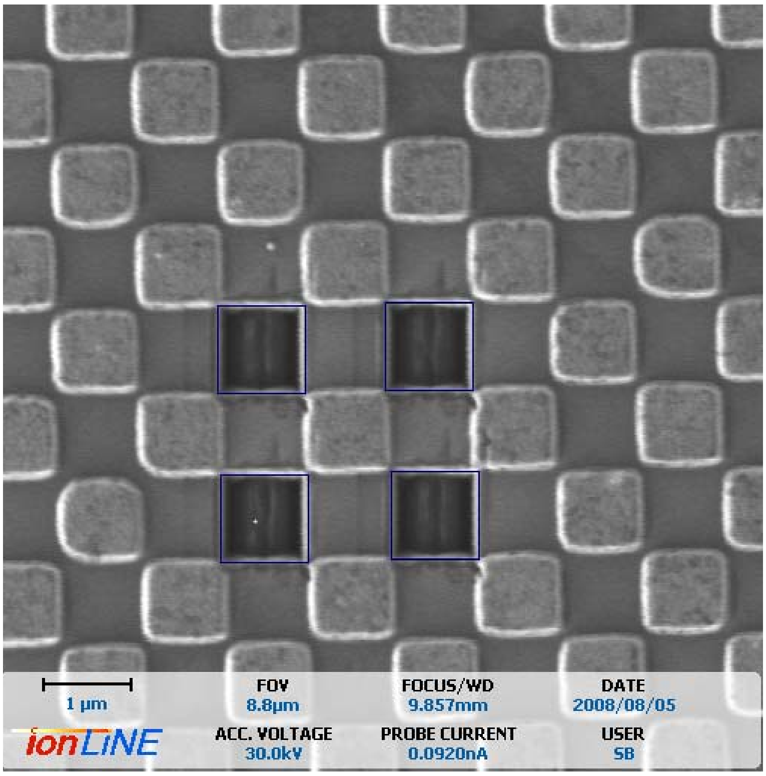


Figure 10-4: GDSII structure placed on the image

- Using the left mouse button, each single element of the GDSII structure can be moved and placed at the required position.
- Press the ▶ button for the next patterning process and then perform another imagescan to view the progress.
- Overlays can be blended out for short periods of time, to check if the patterning has been successful, and afterwards the overlay can be blended back in again.

4 Working with MAP and MAW

To apply a manual position and manual width to a linescan, proceed as follows;

- Open a Positionlist with linescans.
- Select a linescan.
- Double click on the corresponding line in the Positionlist to open it.
- Select MAW or MAP filter via the menu **Filter➡MAW or MAP**. MAP and MAW are abbreviations for manual position and manual width.
- Define two positions in the linescan by two sequential right mouse clicks.

▶ Click on **Apply active filter** to store to the Positionlist the width or center position between defined positions. Depending on the selected filter, the width or position is stored in the Pos3 field in the Positionlist.

5 Surface Editor functionality

The **Surface Editor** module has been implemented as an integrated part of the common software platform and can be used for simple milling applications.

Unlike a classical GDSII/CAD based lithography approach, the surface editor module is completely based on digital images acquired by electron or ion optics. All defined editing actions can be executed directly without any pattern storage or compilation. In addition, there are several end-stop features. Modified areas will be directly located and overlaid onto acquired images, independent of the current calibration.

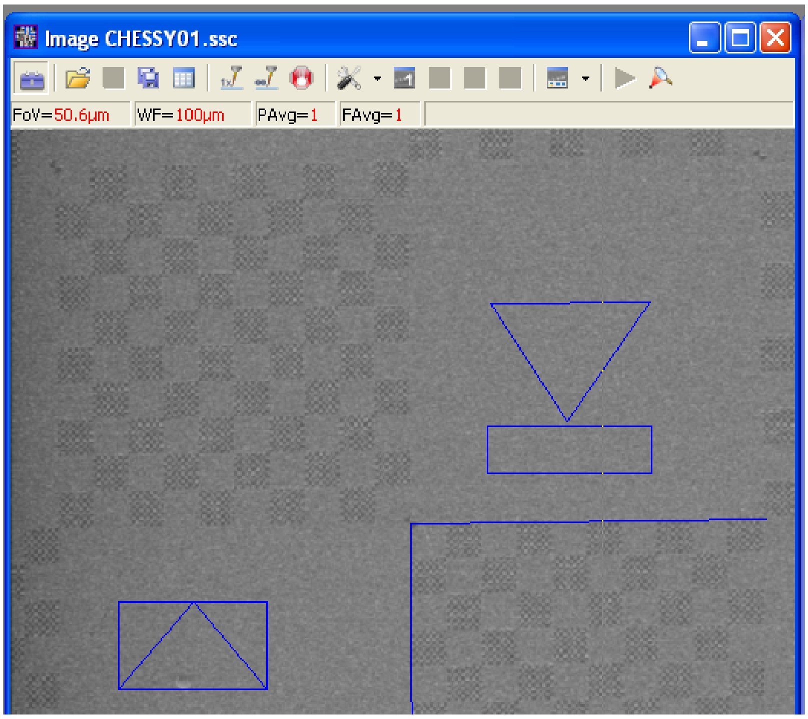


Figure 10-5: **Surface Editor** dialog.

▶ Open the **Surface Editor** control via the menu bar **Edit➡Surface Editor.** This is only possible if the image is active. The surface editor control box will be attached to the image. The dialog allows an interactive definition and execution of simple or more complex milling jobs.

▶ You may scan an image using the **Scan** button.

▶ You may draw **Surface edit features** on top of the acquired image at arbitrary positions and check the resulting process time in the status line at the bottom of the **Surface Editor** window, based on your parameters.

Select an image, either smaller than or as large as the writefield, then select a rectangle, polygon, line or dots or wedges. These can be selected and placed on the image via the mouse.

- For fine placement, click on **Ctrl** and the mouse key to place the structures precisely on the image.
- When **Wedges** are selected, the direction and angle can be defined. The wedge angle is defined as the angle in the cross-section.
- Click on **Escape** to exit the mode.
- When the required designs are complete, you may choose **Loops**, just as for milling applications. The patterning process should be repeated several times until the required depth has been reached.
- The duration time can also be entered.
- You can also choose **Element Repeat** or **Line Repeat**. Each element or line can be looped, the selected number of times.
- Select the **Dwell time** and **Area Step** size. The Area Step size is also applicable to lines.
- Select the **direction**, either horizontal or vertical. This is applicable for Areas as well.
- Enter a **Milling coefficient**, which depends on the material used. The user can enter a number of ms per micron.
- The **Beam current** can be entered. The software will then calculate the corresponding **Area Dose**. When the milling coefficient is changed, the calculated corresponding Area Dose will be updated.
- Clicking on the **Start** button will initiate the **Surface Edit** process. A single process can be initiated. The single process is also executed with a pre-defined number of loops.
- By clicking on the **Loop** button, you can perform a loop process. For example, a normal Surface Edit process may be followed by an additional Scan process and then the Surface Edit process again, if the selected **Duration** time for the loop permits. A loop process continues for a given duration.

All standard image commands such as **Zoom in, Zoom out, Measure** and **Calibrate** can be used and performed in combination with the surface edit tools set. Surface edit tool overlays will be automatically resized and displayed after zooming operations. All in-image navigation commands will also remain valid.

5.1 Wedge feature functionality

The surface edit process of a wedge angle is performed in the following manner:

Firstly the large box (**first surface edit layer** (1), dark blue column in the Figure below) is processed with the selected **Dwell time**.

Then the Dwell times for the **following surface edit layers** (1', 2, 3,., n) are calculated automatically using the following formula:

$$\Delta T_D = L \times K_0 \times \tan\alpha / n$$

This results in an **effective Dwell time on the sample surface** for area **i** (dark blue column in schematic plus the associated column number "i"):

$$T_{D_{eff,\,i}} = T_{D_0} + i(L \times K_0 \times \tan\alpha / n)$$

A final **effective Dwell time** of the **last surface edit area** (n, red column in schematic) is generated

$$T_{D_{eff}} = T_{D_{eff,\,n}} = T_{D_0} + L \times K_0 \times \tan\alpha$$

for the last box **deepest part**.

Whereas:

$$\Delta T_D = \text{dwelltime applied to each following surface edit individually}$$

Looking more closely at the upper wedge feature, we can examine the effect of the parameter selected inside the toolbox above (e.g.: **Dwell time** 0.001 ms, **Step size** 0.103 µm, **Direction** is horizontal).

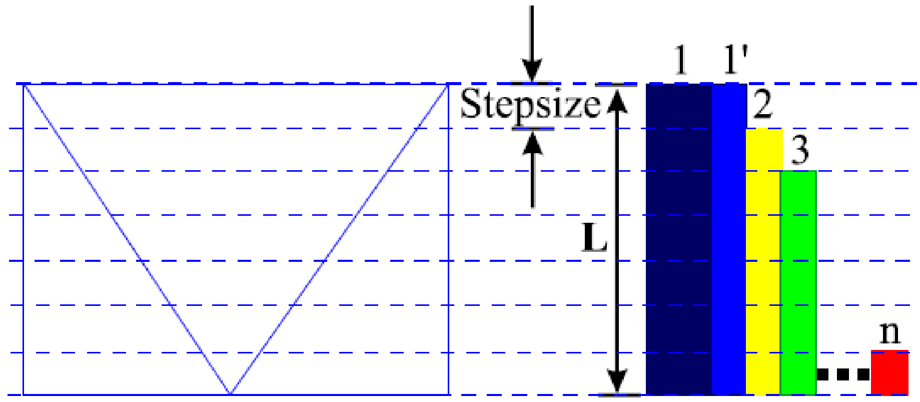


Figure 10-6: Wedge angle process step by step

The colored bars in the figure above show the surface edit process in cross section. Firstly the blue area will be processed completely, corresponding to the complete outlined blue box inside the software image window, with the selected dwell time and step size. The second yellow box will be smaller by one step size in the horizontal direction than the preceding blue one. This rectangle will be milled with the dwell time Del-taT_D.

C Software Reference

1 Linescan window

The title bar shows the file name. In the vertical axis, the measured intensity is displayed, in the range from 0 to the maximum intensity. The horizontal axis shows the scan range in nm.

The status bar at the bottom of the **Linescan** window displays the position in nm, from the start to the end of the linescan, relative to the position at which the linescan has been acquired. The length of the linescan and the intensity which has been measured are displayed.

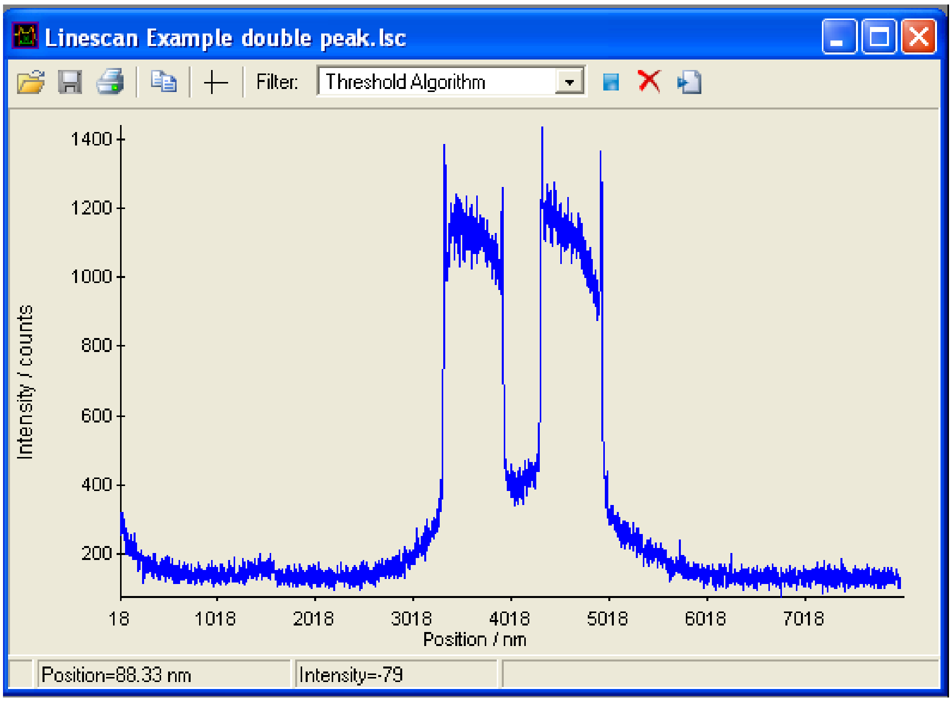


Figure 10-7: **Linescan** window example

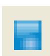
1.1 Linescan tool bar

In **Extras➡Settings➡Virtual Beam Control,** the **ScanWith12Bit** should be set to **On**.
With a 12 bit scan, the linescans acquired with the Ebeam will consist of 12 bit grayscales, equal to 4096.
When this is set to **Off**, the linescan will be acquired with 8 bit grayscales, equal to 256.
For a scan with 12 bits, the scan resolution will be significantly higher, compared to images acquired with 8 bits.
When overlaying an existing 8 bit image with an image linescan, the image linescan will also be limited to 8 bit. Therefore linescans performed with an Ebeam at 12 bits will result in significantly improved resolution and accuracy.

The linescan shows the following tool bar.

Figure 10-8: Linescan tool bar

Control element	Function
	The **Open another linescan** command will open a new linescan file.When a linescan is opened, the corresponding directory is automatically the currently logged in user and timestamp. The linescans have the file extension LSC.
	The **Save linescan** command will save the linescan file. If a filter has been used, it will save the result after the filter algorithm has been applied.
	By clicking on the **Print linescan image** command, a linescan can be printed.
	Clicking on the **Copy linescan image to clipboard** command, the linescan can be copied to the clipboard and then be pasted into a different application.
	Clicking on **Cross cursor On/Off** will toggle between displaying the cross cursor and default cursor.

Control element	Function
Filter subset:	
Threshold Algorithm	Two **Thresholds** are applied, since the intensity has to satisfy two criteria. The use of two levels makes the algorithm robust, since by setting a lower threshold above the bottom noise and a higher threshold below the top noise, outliers can be filtered out.
Edge Detection	**Edge Detection** is used to define the edges. The range between the intersections of threshold levels with the intensity slope is called an edge. The result is either the width or the position of an edge.
Pitch Detection	Once the two edges have been defined, **Pitch detection** can be applied, to calculate the distance between them.
Cross Correlation	**Cross Correlation** is used to compare a measured intensity with a previous recorded intensity.
Noise reduction	**Noise reduction** can be applied, improving the signal to noise ratio by taking the average over adjacent points.
Offset Correction	The background where the horizontal line starts will be subtracted. The calculation is done so that the minimum intensity is set to zero.
Derivative	The **Derivative** filter will calculate the derivation of the intensity signal.
MAP-manual position	The filter positions are set manually.The **MAP** is started from the Positionlist.
MAW-manual width	The **MAW** is set manually.
	Clicking on **Apply active filter** will open the relevant sub-dialog, in which the parameters can be edited and then applied to the currently selected filter to the linescan.
	By clicking on the **Delete filter results** button, the linescan reverts to its original status, before any filters were applied.
	To save filter results, e.g. left edge position to Pos1 field, right edge position to Pos2 field and width or position to Pos3 field.

1.2 Linescan tool bar subdialogs

The following subdialogs are displayed for the linescans. They can be selected via **Apply active filter** in the Linescan tool bar.

1.2.1 Threshold Algorithm

You may call up this dialog by selecting the **Threshold Algorithm** filter and clicking the button.

The **Threshold Algorithm** takes into account that the intensity must always satisfy two criteria, a low and a high threshold. Defining both thresholds, the software can then identify outliers using the Threshold Algorithm.

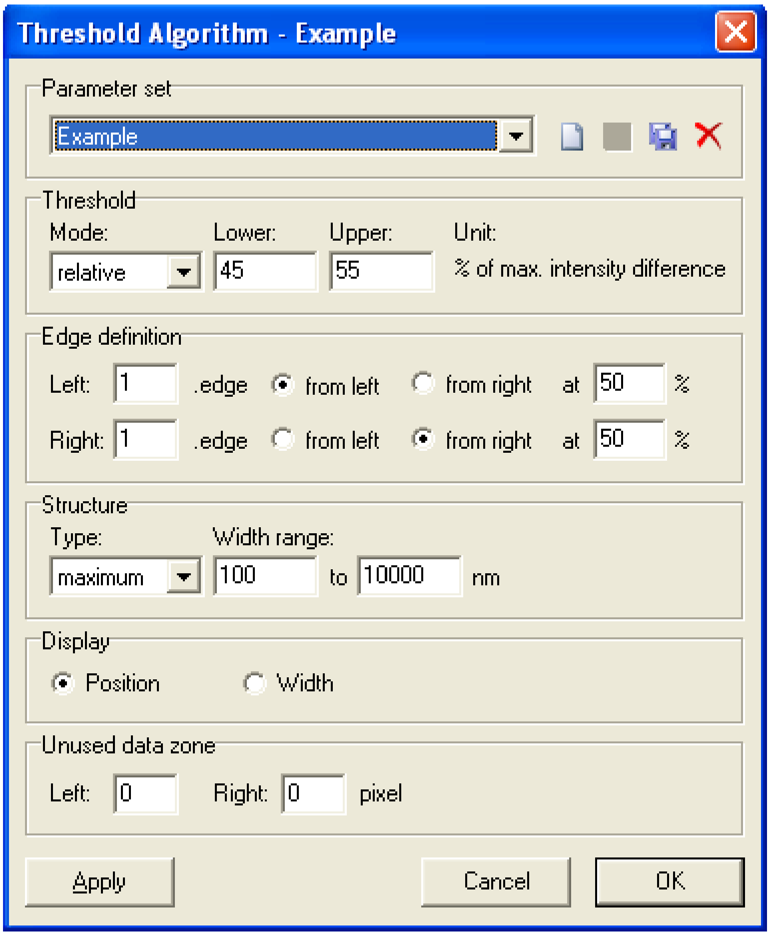


Figure 10-9: **Threshold Algorithm** dialog

Control element	Function
	To open a new parameter set
	To save the parameter set
	To save the parameter set under a different filename
	To delete the parameter set
Threshold	The following threshold parameters can be selected.
Mode	The mode can either be relative or absolute.The software will check the intensity either; - **Relative** - in % between the minimum and maximum intensity values - **Absolute** - the user can enter the intensity value. Select a relative or absolute definition of the threshold values. Normally, relative definition is the more robust detection, because the success does not depend on the signal brightness and contrast. To discriminate out scans that do not achieve a certain signal level, it can make sense to use an absolute definition.
Lower, Upper	The lower and upper threshold limits can be defined relative to (% of) the maximum intensity difference. Type in the threshold values either from 0% to 100% in the relative definition or from 0 to the maximum intensity in the absolute definition mode. Press **Apply** to see the current threshold levels as horizontal, green lines.
Edge Definition	The following edge definition parameters can be selected.

Control element	Function
Left, Right	The left and right edges can be defined. Choose which edges define the structure and from which direction the algorithm is to search for these edges. This setting becomes important if more than one peak is present within the Linescan or when determining the pitch of a grating. The intersections of the intensity curve with the two threshold levels define a range on the position axis. To calculate the position of the edge within this range, use the **at** parameter. A value of 50% means the edge is in the center of this range, whereas a value of 0% or 100% means it is at the left or the right border of the range.
from left, from right	The edges can be defined either from the left or the right.
Input field	The edges can be defined as an intensity level in percent.
Structure	The following structure parameters can be selected
Type Min, Max	The type can be defined as either minimum or maximum. Select either **minimum** or **maximum** from the dropdown list box, if the structure is defined by a minimum or a maximum in the intensity slope.
Width range	The lower and upper width range can be entered in nanometers. Using this setting, it is possible to reject structures which are too small or too large. This function limits the range over which the algorithm searches for the second edge after finding the first edge. If the second edge is not found within this range, the algorithm sets the error flag. Position of the edges as well as the center, the size, and the pitch are not calculated.
Display	The following display options are available:
Position, Width	The user can choose if the position or the width should be displayed. Choose this option to display the width or the center between the two edges, numerically as well as by a yellow line. This option affects only what is displayed, not the information which is stored in a Positionlist.

Control element	Function
Unused data zone Left, Right	The user can enter how many pixels should not be used for the evaluation. For example, at the start (left) and the end (right) of the data acquisition in order to avoid, for example, incorrect evaluation due to dynamic effects.
Apply	The result of the threshold algorithm can be viewed. If the result is satisfactory, it can be saved and written into the Positionlist.

1.2.2 Edge Detection

Call up this dialog by selecting the **Edge Detection** filter and clicking the button.

Edge detection is based on the threshold algorithm, but its functionality has been extended. The width of the edge between the left edge and right edge is calculated.

Threshold Algorithm subdialog (⇒ *Threshold Algorithm* on page 10-19)

In addition, the following parameters can be set:

Control element	Function
Threshold	The following threshold parameters can be selected.
Define upper, lower limit (%)	Check the checkbox if you wish to set an upper and lower limit in percent. Once the checkbox is checked, you may enter the value for the intensity units for the upper and lower limits.
Show limits	Check the checkbox if you wish to display the limits entered.
Structure	The structure can be defined as follows:
Type Raise, Drop	- **Rise** means that the positive flank of a peak will be measured. - **Drop** means the negative flank of a peak will be measured.
Width range	The user can define the acceptable width range, for example in the range 10-1000 nm.

If the width is less than the pre-defined width then it will not be found by the software.

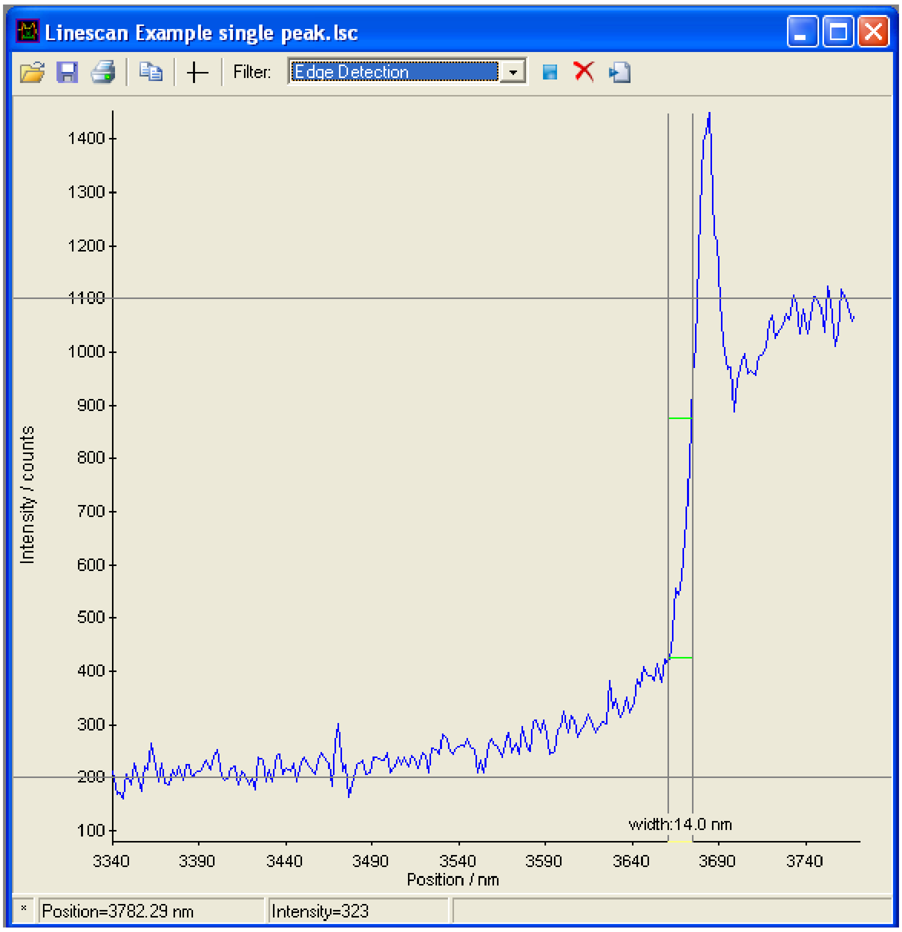


Figure 10-10: **Linescan** dialog - **Edge Definition** applied

To measure the width of the edge manually, define two positions along the linescan by two sequential right mouse clicks. After the second right mouse click, the center position between selected positions and the distance between them is displayed. It is recommended to zoom into the linescan by clicking on the left mouse button and dragging the mouse to the right and releasing the left mouse button. Depending on the distance moved by the mouse, the zoom will vary, the smaller the mouse movement, the further into the linescan you have zoomed.

To zoom out, click on the left mouse button.

Pitch detection

Call up this dialog by selecting the **Pitch detection** filter and clicking the button.

Pitch detection is based on the threshold algorithm, but its functionality has been extended. Pitch detection will measure the distance between two lines. For example, when a double peak should be measured, the pitch in between will be measured. A threshold can be set for the measurement.

Threshold Algorithm subdialog (⇒ *Threshold Algorithm* on page 10-19)

In addition, the following parameters can be set:

Control element	Function
Pitch Definition	The parameters for pitch definition can be selected.
Left, Right	The left and the right pitch can be defined.
from left, from right	The pitches can be defined either from the left or the right.
Input field	The pitches can be defined as an intensity level in percent.

For example, the first edge from the left and the second edge from the left can be defined, as well as type **Raise**. The width can then be measured and confirmed.

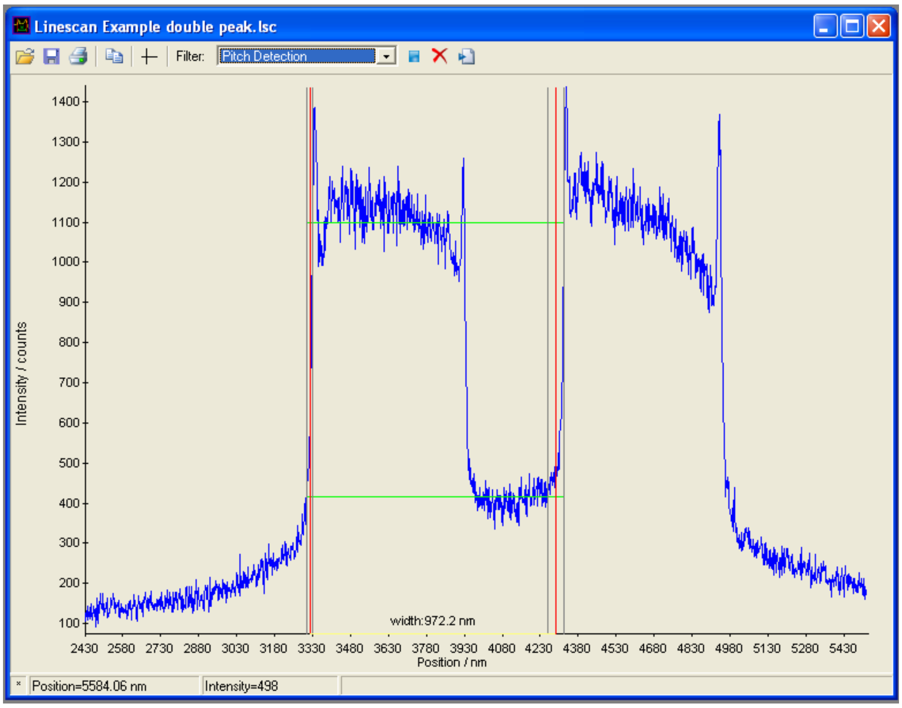


Figure 10-11: **Linescan** subdialog - **Pitch detection** applied

Cross Correlation

Call up this dialog by selecting the **Cross Correlation** filter and clicking the button.

The **Cross Correlation** filter is similar to **Mark Detection** in Images. A reference linescan is selected against which the current linescan is compared. The reference linescan represents the ideal to which the current linescan is compared. The shift is then calculated by the software.

Cross Correlation is a very robust method compared to the **Threshold Algorithm**, in which clearly defined linescans are required to apply the threshold algorithm accurately. It is therefore recommended to use cross correlation if successful measurement is very important to the application.

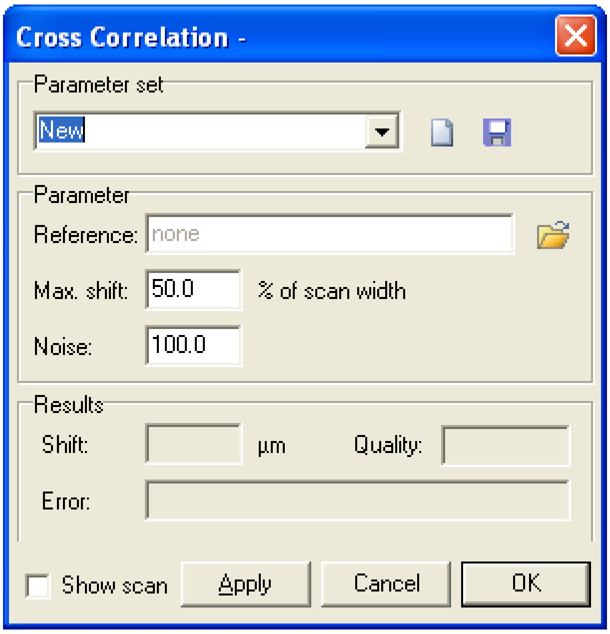


Figure 10-12: **Linescan** subdialog - **Cross Correlation**

Control element	Function
Parameter set	The parameter set can be selected from the dropdown list. - **New** - A new parameter set can be created. - **Save** - A parameter set can be saved.
	A new parameter set can be created.
	A parameter set can be saved.
Parameter	The following parameters can be defined:
Reference	Click on the **Open** button to select a **Reference** linescan from the database.
Max shift	Enter into the input field the maximum shift permitted, as a percentage of the scan width.
Noise	This will reduce the noise observed in the linescan and will improve the signal to noise ratio. The parameter "noise" determines the noise sensitivity during the search for the maxima. Any neighboring peaks for which the maximum difference between the valley and the top of the peak is within the defined maximum noise level, will be ignored.

Control element	Function
Results	The following parameters can be calculated:
Shift	After clicking the **Apply** button, the software will calculate the shift between the reference linescan and the current linescan.
Quality	The software will also calculate the **Quality** of the result. The closer that the Quality value is to 1, the higher the quality of the result.
Error	The error message that indicates the reason that calculation can not be performed.
Slow scan	Check the checkbox if you wish to see the output of **Cross Correlation**: The position with the highest intensity value is marked by a red vertical line. This position represents the shift between the selected linescan and the reference linescan.
Apply	Click on **Apply** to carry out the cross correlation. The **Shift** is now calculated as well as the **Quality** of the measurement.

To interpret the results, the **Quality** of the measurement can be viewed. The optimum value is 1 (maximum), this would indicate a very good signal. The value, calculated by the software, gives a very good indication of the quality of the result. The closer to the value of 1, the better the result.

Cross Correlation is so robust that it even delivers results when the shift value is large or the scope value is poor (not close to the optimum value of 1). It is therefore recommended to set limits for the permitted shift.
It is recommended that the value should be higher than 0.9.

Noise Reduction

For noise reduction, the **Number of average points** and **Number of iterations** are entered.

Call up this dialog by selecting the **Noise Reduction** filter and clicking the button.

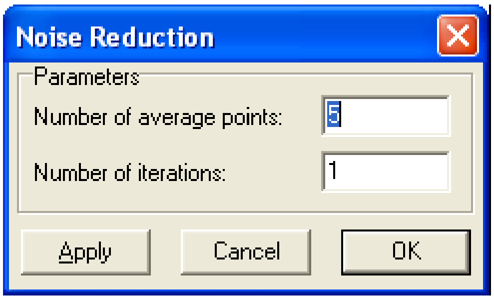


Figure 10-13: **Noise Reduction** dialog

Control element	Function
Parameters	The following parameters can be selected.
Number of average points	Enter the number of adjacent points that you wish to use to calculate the average. The more points selected, the flatter the distribution.
Number of iterations	Enter the number of iterations.

For example, if the number of averaging points entered is 5, the average is taken from five adjacent points. The grayscales will be added and the sum will be divided by 5. This will flatten the distribution, as the noise is reduced by taking the average.

Offset Correction

Call this up by selecting the **Offset Correction** filter.

An **Offset Correction** can be applied to the linescan. When selected, the offset correction is directly applied to the linescan, without opening another dialog. When using offset correction, the background is set to the start of the horizontal line.

Derivative

Call this up by selecting the **Derivative** filter.

The **Derivative** can be applied to the linescan. When selected, Derivative is directly applied to the linescan, without opening another dialog. The **Derivative** filter will calculate the derivation of the intensity signal.

MAP - manual position and MAW - manual width

Call this up by selecting the **MAP** and **MAW** filter.

MAP and **MAW** are abbreviations for manual position and manual width. MAP and MAW are started from the Positionlist (linescan). When selecting MAP or MAW, no further dialog will open.

The distance between two points is defined manually by two sequential mouse clicks. If you then apply the MAP filter, the center position of the two previously defined points will be calculated and stored in the Positionlist, field Pos3. The positions of manually defined points are stored in Pos1 and Pos2 respectively.

If you apply the MAW filter, the distance between the two previously defined points will be calculated and stored in the Positionlist, field Pos3.

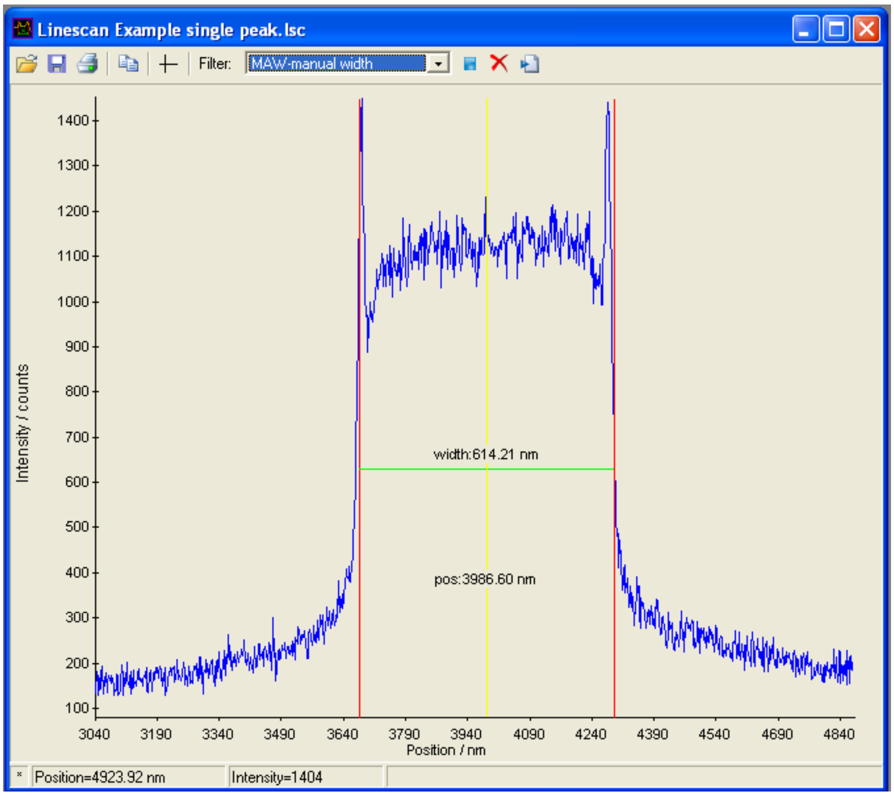


Figure 10-14: Linescan dialog - **MAP** and **MAW** applied

1.3 Main menu

The following menus are available:

1.3.1 File menu

File menu item	Function
Print	To print a Linescan. This command opens the default print dialog.
Open...	To load an existing Linescan. This command replaces the currently loaded Linescan by the newly selected one.
Save	To save the Linescan under its current name
Save as...	To save the Linescan under a freely selectable name
Close	To close the currently selected Linescan
Close all...	To close all currently open Linescans

1.3.2 Edit menu

Edit menu item	Function
Copy	To copy the Linescan into the Windows clipboard as pixmap (device-independent bitmap). From here it can be pasted into another application, for example for documentation purposes.
Cross Cursor	To toggle between the default mouse cursor and a crosshair

1.3.3 Filter menu

Filter menu item	Function
Apply active filter	To apply the selected filter and to automatically display the result. This command is also available via the Linescan window tool bar.
Select active filter	An additional dropdown menu appears. Here you may choose to define the active filter. An overview of the available filters is given in section Linescan Filters. This command is also available through the Linescan window tool bar by selecting a filter from the dropdown list.
Delete filter results	To delete the result of the selected filter. This command is also available through the Linescan window itself.

Filter menu item	**Function**
Store filter results	To store the result of the selected filter to the Positionlist. This command can only be used if the Linescan was opened from a Positionlist. When using one of the threshold algorithms or MAP/MAW filter, the left edge is stored in column Pos1, the right edge is stored in column Pos2, and the width or position is stored in column Pos3. This command is also available through the **Linescan** window itself.

Some of these commands are also available via the Linescan tool bar, Linescan tool bar subdialogs (⇒ *Linescan tool bar* on page 10-17).

1.4 Mouse and keyboard commands

Use **Ctrl+C** to copy an image of the Linescan into the Windows clipboard, to paste it into any other application.

Use **Ctrl+O** to load an existing Linescan into the current window. Use **Ctrl+S** to save the current Linescan.

Use the **right mouse** button to sequentially set two marks within the Linescan. These marks can be, for example, edges or significant features. The positions of these two marks are indicated by two red lines and are stored together with the Linescan.

Use the **left mouse** button and move the mouse to the right to zoom into the Linescan data. It is possible to examine, at higher magnification, a specific substructure. Use the left mouse button again to zoom out.

Key and mouse (click)	**Functionality**
none+right (single)	Sequentially sets two marks to manually define edges, etc.
Drag+left (none)	Zoom into Linescan data. To zoom out, press the left mouse button again.

2 Image window

The image acquisition commands within this mode make use of the high resolution scan generator. The images are acquired with the current scan calibration, i.e. writefield **zoom**, **shift**, and **rotation**. In addition, the images are displayed in UV-coordinates.

The link to the absolute sample coordinate system is derived from the position sensing system of the sample stage. The images are handled as UV windows, allowing addressing, drag and drop and overlay options.

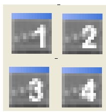
More background information is given in the Functional description section, (⇒ *Functional Description* on page 10-2).

2.1 Image tool bar

Figure 10-15: Image tool bar

Image - tool bar	**Function**
	To open the **Toolbox**.
	The **Open (Ctrl+O)** command will open a new imagescan file.
	The **Save (Ctrl+S)** command will save the imagescan file. If a filter has been used, it will save the result after the filter algorithm has been applied.
	The **Save image to another file (Shift +Ctrl+S)** command will save the imagescan to a different file.
	The **Image information** command opens another dialog to show and change the image information.
	The **Scan image (F5)** command will scan the images once.
	The **Scan image continuously (Shift+F5)** command will scan the image continuously.
	The **Stop imagescan (F6)** command will stop the scan.

 **Imagescan options**

Image - tool bar	**Function**
	Quicksets
Edit Data zone subset:	
Data zone....	A **Data zone editor** dialog will open, in which the user can set the parameters for the data zone display.
Show data zone	If activated, the data zone will be displayed at the bottom of the image. When it is selected, a checkmark is displayed next to it. If it is not selected, the data zone will not be shown and no checkmark will be displayed.
	The command **Start patterning of the image design** will initiate the patterning of the associated GDSII structure opened in the image's own GDSII editor.
	This command will **Burn spot in the middle of the scan area.**

2.2 Image tool bar subdialogs

The following subdialogs are available from the Image tool bar.

2.2.1 Toolbox

The Toolbox consists of several segments, such as **Generic** tools, **Image** tools, and **Column** tools. The toolbox is dynamic, changing according to the windows which are open.

For the description of the **GDSII tools**, please refer to the **GDSII** chapter (⇒ *Toolbox* on page 5-135).

Most of the **Image tools** in the **Toolbox** are also available directly in the **Images** tool bar, (⇒ *Image tool bar* on page 10-32).

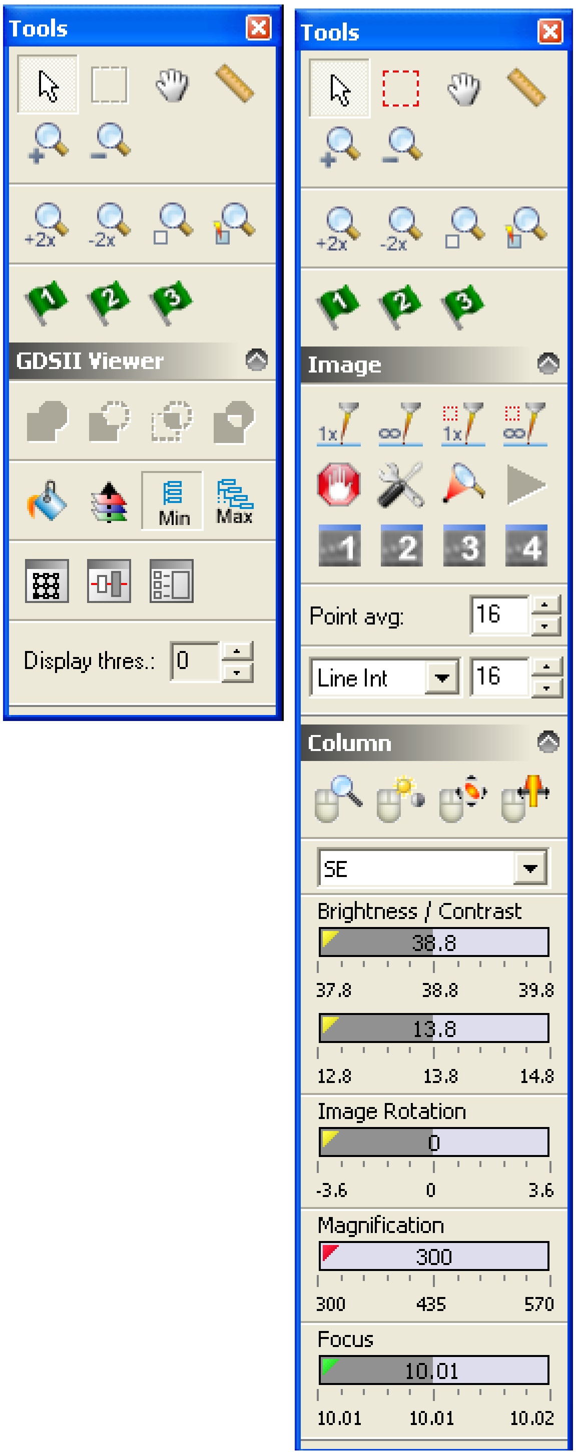
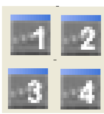


Figure 10-16: Image tool bar - **Toolbox**

2.2.2 Image subset

Image subset:	Function
	The **Scan image (F5)** command will scan the image once. This is a simple single scan.
	The **Scan image continuously (Shift+F5)** command will scan the image continuously
	The **Scan selected area (F7)** command will scan the selected area only once.
	The **Scan selected area continuously (Shift+F7)** command will scan the selected area continuously. This is the recommended setting for using the wobbler.
	The **Stop imagescan (F6)** command will stop the scan. Any scan, either the full image or a reduced image scan will be interrupted.
	The **Imagescan options** are displayed in a dropdown list, consisting of **Imagescan options dialog** and a list of user-defined quick sets.
	This command will **Burn spot in the middle of the scan area.** This is a useful feature when refining the focus, as well as being a check of the beam quality. The spot will be burned into the center of the image. The spot should be round and symmetrical, which is a good indication of correct alignment of the column.
	The command **Start patterning of the image design** will initiate patterning. This is used when a GDSII design structure has been dragged and dropped onto an image when the **Alt** key was pressed.
	Fields 1 - 4 are pre-programmable image scan parameters which can be defined by the user.
Point average	The user can enter a value for the **Point average**.
Frame Avg, Line Int, Frame Int	The user can toggle between **Frame Average, Line Integration** and **Frame Integration**. The value for the average or integration can be entered.

2.2.3 Column subset

The **Column subset** toolbox is available in addition to the **Column Control Fine Tune** window. Both display a similar set of control functions, except for **Image Rotation**, which is an additional feature of this toolbox.

Whenever an image is opened, the toolbox also shows the most important control elements for the column control.

The first four mouse buttons , , and are tools which can be used on the image. Various functions can be applied to the mouse buttons. These mouse functions are only active on the image. Each of the buttons contains a small mouse image.

When a function is applied to a mouse button, e.g. magnification to the left mouse button, the cursor will now display the magnification button when the mouse cursor is placed on the image.

To change, for example, the magnification, click on the left mouse button and keep it pressed down with the cursor on the image, while dragging the mouse to the right, to increase magnification, or to the left to decrease magnification. Release the left mouse button to set the new magnification.

The same principle applies to the other functions, such as **Brightness/ Contrast, Aperture and Stigmator.**

To deselect, click the same button again or press **Escape.**

Column subset:	**Function**
	Assigns the **Magnification/Working distance** to the left/ right mouse buttons.
	Assigns **Brightness/Contrast** to the left/right mouse buttons.
	Assigns the **Stigmator alignment/Working distance** to the left/right mouse buttons.
	Assigns the **Apertures alignment/Working distance** to the left/right mouse buttons.
Detector selection	Select the detector from the dropdown list. The user may toggle between different detectors available for the particular system, e.g. Inlens or SE2 detector. For further information, please refer to the **Column Control➡Fine Tuning** window, (⇒ *Detector* on page 7-36)
Brightness/Contrast	Using the sliderbar, the brightness and contrast can be varied.

Image Rotation	The image can be digitally rotated via the sliderbar. This is an additional control element, only available in this toolbox, since it is only applicable to an image. This command will rotate the image digitally. The rotation angle is controlled from the pattern generator. When a writefield is written, the rotation angle is reset to zero. This digital rotation is purely applied to the image, in order make it easier to view.
Magnification	The magnification can be adjusted via the sliderbar.
Focus	The focus can be adjusted via the sliderbar.

All sliderbars in the **Column Toolbox** are synchronized to the sliderbars in **Column Control.** They will always show exactly the same settings.

The toolbox enables very fast access to the tuning parameters. For example, if the user wishes to change between focus and stigmator several times during the tuning process, this can be performed more quickly using the **Column Toolbox** than by selection via the **Column Control** windows. The main tuning functions are therefore available via the toolbox.

Before the mouse tools are used, the mouse has to be placed on the image. For a detailed description, please read **Mouse Tool Configuration** in the chapter Column Control, (⇒ *Software tools within Fine Tune dialog* on page 7-5). To change the value of the setting, for example to increase the zoom if the magnification has been assigned to the left mouse button; the user can drag the mouse to the right over the image using the left mouse button and the magnification of the image will be increased. Dragging the mouse to the left will decrease the magnification.

The same functional principle can be applied to changing the **Working distance, Brightness/Contrast** etc.

If the user wishes to adjust a setting significantly, the cursor can be dragged using the left mouse button from inside the image to outside the image, in order to change the value over a wider range.

Advanced users can use the following procedure to speed up the tuning process. While an image is active, the keyboard numbers 1 to 4 in the **Image** tool bar will toggle between the four mouse settings. This is suitable for very advanced users, who may wish to keep one hand on the mouse button, while the other hand is placed on the keyboard to toggle between the mouse assignments, using the number keys on the keyboard.

For further information, please refer to the chapter **Column Control**, (⇒ *Fine Tune dialog* on page 7-34).

2.2.4 Image Information

Call up this dialog using the menu **Edit➡Image Information.**

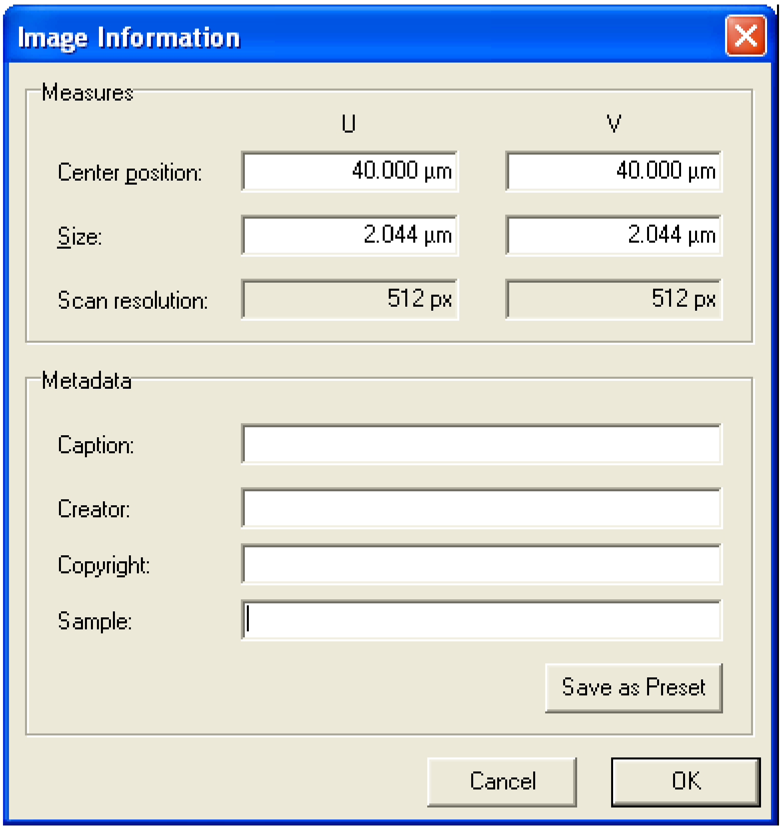


Figure 10-17: **Image Information** dialog

Control element	Function
Measures	The following parameters can be entered.
Center position	The user can enter the position in U and V.
Size	The user can define the image size in U and V directions.
Scan resolution	The scan resolution is given for information only. The unit is pixels which corresponds to one DAC step in the pattern generator.
Metadata	The following information can be entered for the Metadata.
Caption	A **Caption** can be entered in the text field, e.g. a title.
Creator	The user can enter the **Creator**, e.g. the user name and institute or company.
Copyright	A copyright notice may be entered.
Sample	A sample description can be entered.
Save as Preset	Clicking the **Save as Preset** button will save the Metadata as the new preset values.

2.2.5 Imagescan Options

Call up this dialog by selecting the **Imagescan Options** button.

There are four tabs available for **Imagescan Options**, consisting of **Scan**, **File**, **Overlays** and **Quick sets**.

2.2.6 Imagescan Options - Scan tab

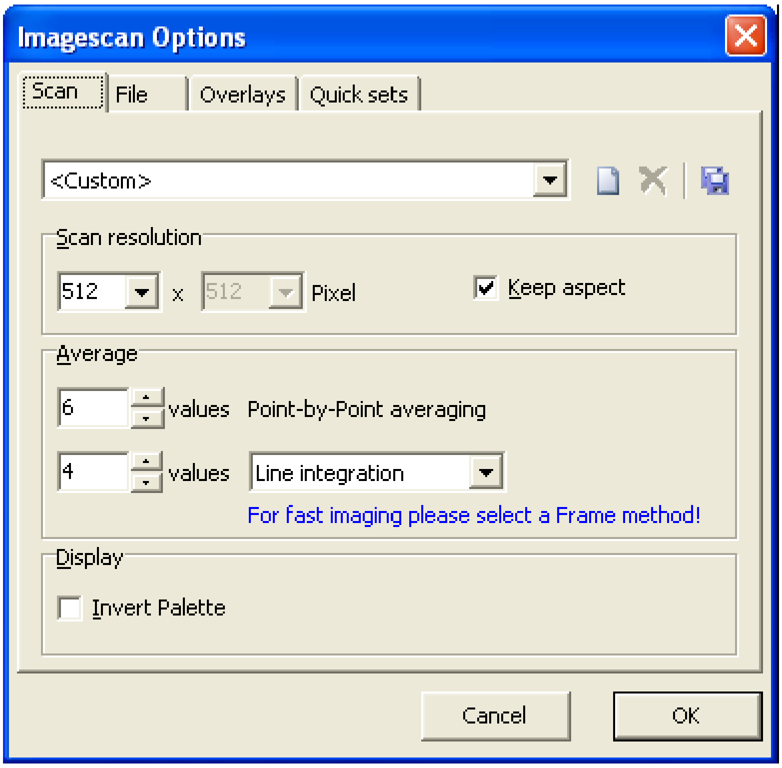


Figure 10-18: **Imagescan Options, Scan** tab

Control element	Function
File selection	A parameter set can be chosen from the listbox.
	To open a new parameter set
	To delete the parameter set
	To save the parameter set under a new name
Scan resolution, Keep aspect	The user can enter the resolution in pixels. If **Keep aspect** is checked, the aspect ratio will be retained and only one axis needs to be defined. Enter the number of pixels along the U and V axes. The number of pixels must be from 64 to 4096.
Average	The following parameters for average calculation can be selected.

Control element	Function
Point-by-Point averaging	The number of values used for point-by-point averaging can be entered into the input field. Enter the number of **Point-By-Point** averaging cycles, which results in a multiple intensity sampling at each individual point, to enhance the signal-to-noise ratio.
Frame avg, Frame Int, Line int	Using the dropdown list, the user can choose between **Frame averaging, Frame integration or Line integration.** The number of values taken into account for either averaging or integration can be entered. Enter the number cycles and choose either **Line integration, Frame integration or Frame averaging** for signal averaging by multiple scans. If **Frame integration** is chosen, the entire image is scanned the number of times defined by the number of averaging cycles. After finishing a cycle and if continuous scanning has been selected, the integration starts with a noisy image again. If **Frame averaging** and continuous scanning are chosen, the entire image is averaged over the last scanned images the number of times defined. If **Line integration** is chosen, each line is scanned the number of times defined by the number of averaging cycles.
Display	The following display parameters are available:
Invert palette	The palette can be inverted for easier viewing, if applicable. If **Invert Palette** is selected, the inverted gray level palette is displayed on the screen. Otherwise the original gray level palette is displayed. This setting will be saved within the image file.

2.2.7 Imagescan Options - File tab

Call up this dialog by selecting the **Imagescan Options** button

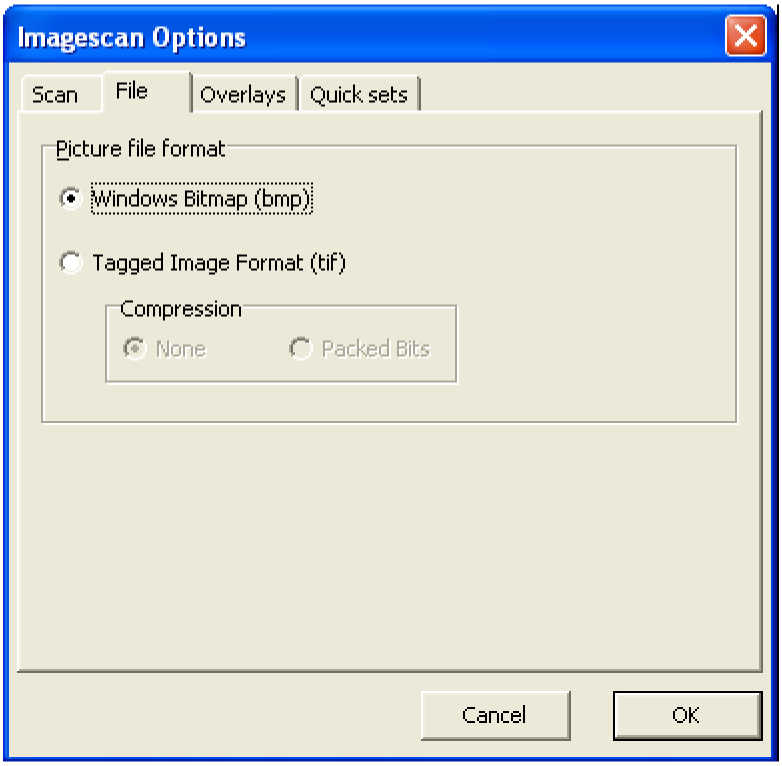


Figure 10-19: **Imagescan Options, File** tab

Control element	Function
Picture file format	The following options are available for the Picture file format:
Windows Bitmap (bmp)	To save an image as an uncompressed bitmap digital image file with a color depth of 8 bits.
Tagged Image Format (tif)	To save an image as Tagged Image Format file with a color depth of 8 bits. In addition, one can choose between uncompressed or compressed via the Macintosh Packbits loss-free data compression scheme.

2.2.8 Imagescan Options - Overlays tab

Call up this dialog by selecting the **Imagescan Options** button.

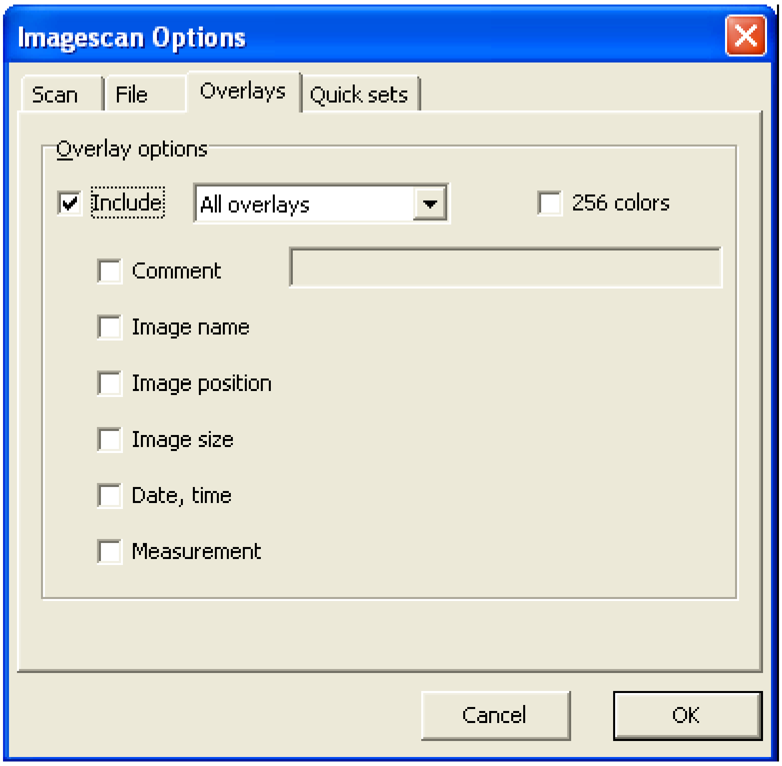


Figure 10-20: **Imagescan Options, Overlays** tab

Control element	Function
Overlay options subset:	
Include	Select **Include** to store overlays together with the image. In this case, choose between **Scaling bars only** and **All overlays,** i.e. overlays of Positionlists, GDSII elements, etc. In addition it is possible to include a comment, the image name, the image position, the date and time, or the image size. If overlays are included, the overlay information is stored within the image itself. Hence, post processing of this image, e.g. an image filter, can not be applied.
Comment	To activate the comment input field. A comment can then be entered and displayed.
Image name	To display the image name
Image position	To display the image position

Control element	Function
Image size	To display the image size
Date, time	To display the date and time
Measurement	To display the image measurement

2.2.9 Imagescan Options - Quick sets tab

Call up this dialog by selecting the **Imagescan Options** button.

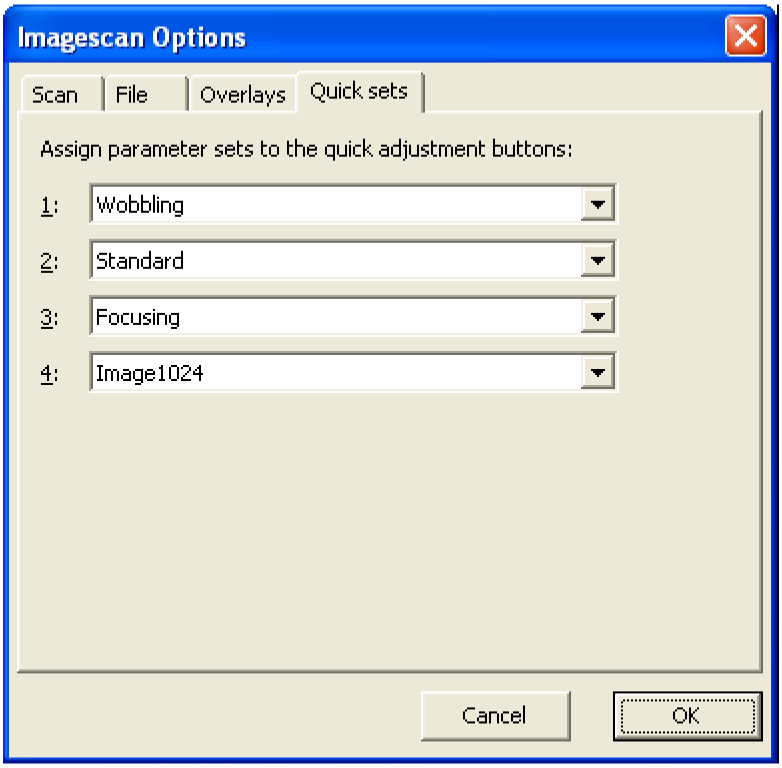


Figure 10-21: **Imagescan Options, Quick sets** tab

The user can create as many **Scan** datasets as required in **Imagescan Options➡Scan** tabs, via the **New** button, followed by the **Save As** button. For each of the **Quick sets**, one of the Scan datasets can be selected and assigned from the dropdown list.

Control element	Function
1, 2, 3, 4	The **Quick set** buttons 1 to 4 can be assigned to the scan parameter sets defined in the **Scan** tab.

2.2.10 Data Zone Editor

Call up this dialog by selecting the **Data Zone Editor** button.

Selecting **Data zone editor** arrow displays a dropdown list with the following functions:

Control element	Function
Data zone	This will display a new dialog, as shown below
Show data zone	The display of the data zone can be toggled **On/Off.**
<Custom>	Built-in data zone configuration set that is selected by default.

Clicking on **Data Zone...** displays the following dialog.

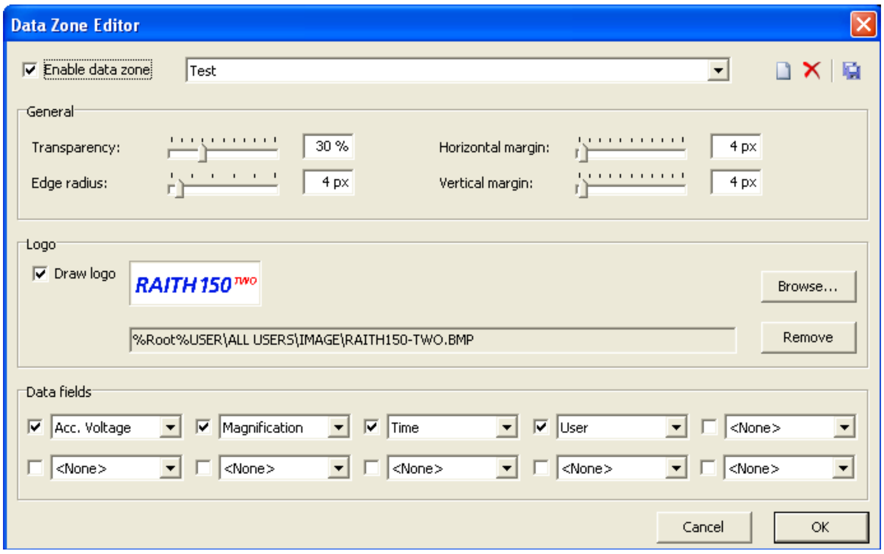


Figure 10-22: **Data Zone Editor** dialog

Control element	Function
Show data zone	Check the checkbox to overlay the data zone in an image window. Select a data zone parameter set from the drop-down list.
	To open a data zone parameter set
	To delete a data zone parameter set
	To save the data zone parameter set under a new name.
General subset:	
Transparency	The transparency of the data zone background can be set in %, using the slider bar.
Edge radius	The edge radius in pixels can be set via the slider bar.
Horizontal, vertical margin	The horizontal and vertical margin in pixels can be set via the slider bar.
Logo	Check the checkbox **Draw logo** if you wish to include a logo in the data zone. Select the filename via the **Browse** button.
Data fields	Check the number of **Data fields** you wish to activate for display. For each Data field, one of the following parameters can be selected from the dropdown list: **None, Acc. Voltage, Active Mode, Average Count. Average Type, Caption, Copyright, Creator etc.**

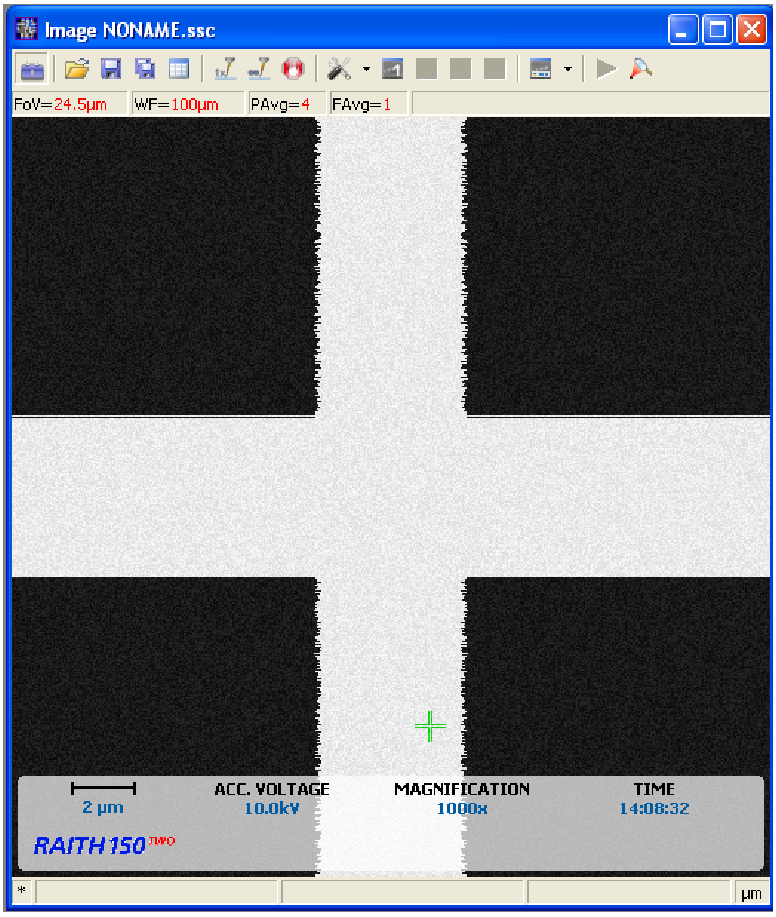


Figure 10-23: Image displayed with data zone

2.3 Context menu

Clicking on the right mouse button inside the image enables the following commands. The commands change dynamically, depending on the windows open.

Control element	Function
Select area	An area can be selected. Click once with the **left mouse** button for the first corner. Leave the mouse button pressed down and drag the cursor to the opposite corner and release the mouse button. The area is now selected. This is the recommended setting for performing fine tuning, e.g. using the wobbler.
Hide selection	The selected area can be hidden.
Calibrate	Choose this command to perform an image calibration. A **Calibrate** dialog will open.

Control element	Function
Zoom in, Zoom all	The **Zoom** command is also available via the toolbox, (⇒ *Image tool bar subdialogs* on page 10-33).
Direct alignment...	Choose this command to perform a direct alignment. A **Direct Align Image** dialog will open.
Shift window UV	The UV origin can be defined. The functionality is described in the chapter **UV Windows,** (⇒ *UV windows* on page 4-4).
Overlays	The user can define the overlay options. When Overlays are enabled, a checkmark will be displayed.
Toolbox	Clicking on **Toolbox** will open the toolbox. Once it is open, a checkmark will be displayed next to it.
Cursor grid	A cursor grid can be applied to the image. The user can select the cursor grid distance from the dropdown list. The options are either **None, 1, 2, 5 or 10** µm. If for example, 2 µm is selected, the cursor will increment by 2 µm.
UV display	Clicking on **UV display ....** opens a dialog, in which the user can define the UV window properties.

2.3.1 Beam Scale Calibration

Clicking on **Calibrate....** in the context menu opens the **Beam Scale Calibration** dialog.

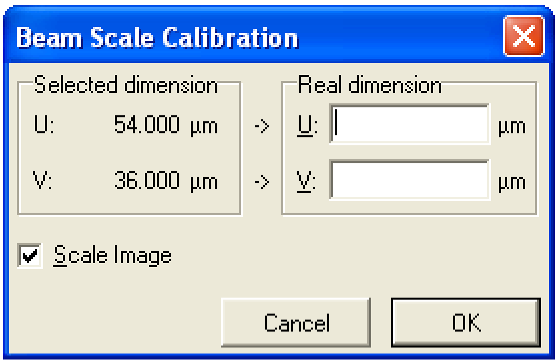


Figure 10-24: **Beam Scale Calibration** dialog

Control element	Function
Selected dimension	The selected dimension is displayed.
Real dimension	The user can enter the real dimension in U and V directions for the calibration.
Scale image	To scale the image

2.3.2 Direct Alignment

Clicking on **Direct Alignment** in the context menu opens the **Direct Alignment Image** dialog. This dialog enables the user to carry out a quick writefield alignment procedure using the reference structure shown in the image. This feature is of particular interest for overlay milling applications.

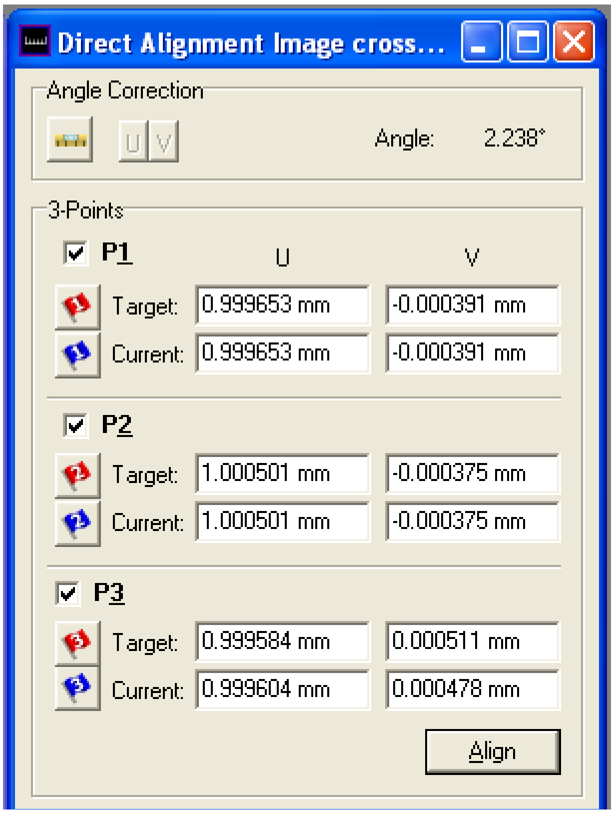


Figure 10-25: **Direct Alignment Image** dialog

Control element	Function
Angle Correction subset:	
	Click on the **Straighten** tool to perform an angle correction.
U, V	Once the **Straighten** tool is selected, the U and V buttons become available. Select either **U** or **V**.
Angle	To perform the **Angle Correction** and display the angle in degrees
3-Points subset:	
Target P1, P2, P3 	Using the red flag, the target positions for **P1, P2, P3** can be defined.
Current P1, P2, P3 	Using the blue flag, the current positions for **P1, P2, P3** can be defined.

2.3.3 UV Display...

Clicking on **UV Display...** in the context menu opens the **UV Display Options** dialog. This is explained in detail in the chapter **UV Windows**, (⇒ *UV windows* on page 4-4).

2.4 Main menu

The main menu has the following commands:

2.4.1 File menu

File menu item	Function
Close	To close the activated image
Close all images	To close all open images
Open...	To open an existing image from the database
Save	To store the activated image under its current name. File format is either BMP or TIF, due to the **Picture file format** settings within the **Imagescan Options** dialog (⇒ *Imagescan Options* on page 10-39).
Save as...	To store the activated image under a freely selectable name. File format is either BMP or TIF due to the **Picture file format** settings within the **Imagescan Options** dialog (⇒ *Imagescan Options* on page 10-39).
Print...	To print out the currently viewed area of the Image
New image	To open a new image window. The image size is automatically set to the writefield size from the **Microscope Control.** The image center position is the current stage position.

2.4.2 Edit menu

Edit menu item	Function
Copy	To copy the active image into the Microsoft Windows clipboard. This allows pasting of the image directly into a different application.
Image information	To show position, size and scan resolution of the activated image. A new dialog will open. Values for image position and size can be edited. The same dialog can be accessed from the **Image** tool bar, (⇒ *Toolbox* on page 10-33).

Edit menu item	Function
Data zones...	To edit the data zone parameters for display. The **Data Zone Editor** dialog will open. The same dialog can be accessed from the **Image** tool bar. (⇒ *Toolbox* on page 10-33).
Surface editor...	To open the **Surface editor** subdialog. (⇒ *Surface Editor functionality* on page 10-12).
Adjustment subset: Commands common to all UV windows, described in the chapter **UV Windows**, (⇒ *UV windows* on page 4-4).	
Set mark 1, 2, 3	The marks for **3-Points Adjustment** can be set.
Measure	Command common to all UV windows, described in section **UV Windows**, (⇒ *UV windows* on page 4-4).

2.4.3 Surface Editor

Selecting **Surface Editor** will display a toolbox, allowing simple and sophisticated direct modification actions:

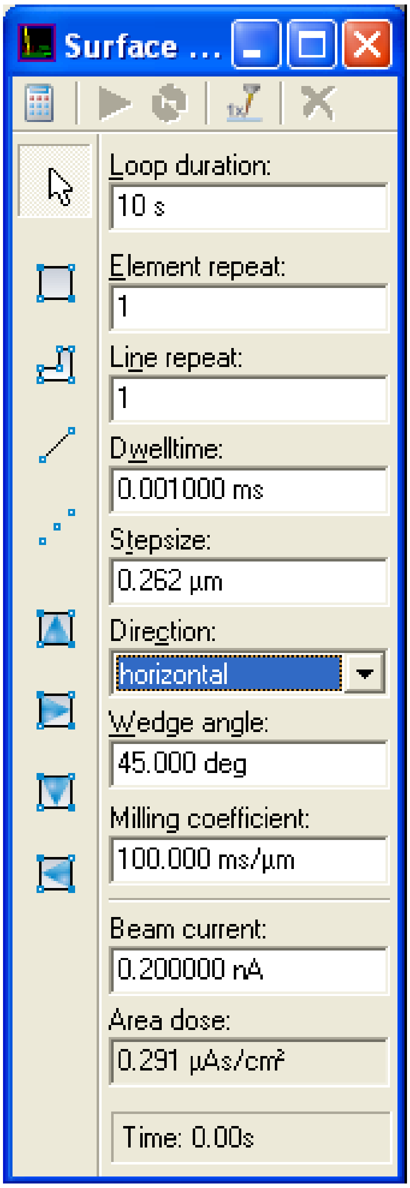
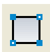
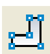
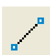
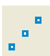
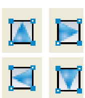


Figure 10-26: Edit menu - **Surface Editor**

Control element	Function
Tool bar subset:	
	Clicking on the **Calculator** button will calculate the process time.
	Clicking on the **Play (F9)** button will start the first single process.
	Clicking on the **Loop (Shift + F9)** button will start the loop process for the given duration.
	Clicking on the **Scan (F5)** button will initiate the scan of the image.
	Clicking on the **Delete** button will delete all elements.
Buttons on vertical bar subset:	
	Click on the **Select** button to select and to move elements. It also stops the element creation process and allows access to normal editing modes in the image. After selecting an element, it can also be moved.
	Click on the **Box** button to create a rectangle on the image. The box can be created and placed directly onto the image. The box coordinates are defined by subsequent left mouse click operations. This command acts with an auto repeat.
	Click on the **Polygon** button to create a polygon on the image. The polygon of any shape can be created and placed directly onto the image. The coordinates are defined by subsequent left mouse click operations until the polygon is closed (filled), by a right mouse click operation. This command acts with an auto repeat. The backspace key during the creation of a polygon deletes the last point.
	Click on the **Line** button to create a line on the image.
	Click on the **Dots** button to create dots on the image.
	Click on one of the **Wedge** buttons to create wedges in the direction of the arrow.

Control element	Function
Text input fields subset:	
Loop duration	Enter the **Loop duration** in seconds.
Element repeat	Repeats every element **n** times. Enter the number of repetitions of the created elements during a **Surface Edit** process.
Line repeat	Repeats every line within elements **n** times. Enter the number of times you wish to repeat the line.
Dwelltime	Enter the dwell time in milliseconds. The dwell time is the length of time that the beam stays on a dot.
Stepsize	Enter the step size in microns.
Direction	Select the scan direction as either horizontal or vertical.
Wedge angle	Define the angle for the wedge in degrees. The defined wedge angle in the text field and the resulting physical wedge angle on the sample can differ.
Milling coefficient	Enter the milling coefficient used for the calculation. The milling coefficient is dependent on the sample and the column parameters used.
Beam current	Enter the beam current in nA to be used for the calculation.
Area dose	The Area dose will be calculated by the software and is given for information only.
Time	The estimated process time is calculated and is given for information only.

2.4.4 View menu

View menu item	Function
Tool bar	Select the tool bar to **view** or to **hide** the tool bar.
Data zone	Select this command to show the Data zone.
Redraw	Command common to all UV windows, described in the chapter **UV Windows**.

View menu item	Function
Zoom subset: A cascading menu opens with additional commands, described in the chapter **UV Windows**.	
All, In, Out, Last, writefield, Zoom in tool, Zoom out tool, Hand tool	Commands common to all UV windows, described in the chapter **UV Windows**, (⇒ *UV windows* on page 4-4).
Overlays	Command common to all UV windows, described in chapter **UV Windows** (⇒ *UV windows* on page 4-4).
Toolbox	Command common to all UV windows, described in chapter **UV Windows** (⇒ *UV windows* on page 4-4).

2.4.5 Options menu

Control element	Function
Histogram...	To show the distribution of gray levels
Imagescan...	To set image scan parameters. This command is also available via the Imagescan tool bar, (⇒ *Toolbox* on page 10-33).
Fix image	To fix the image size
Scale image	Choose this option to display the Imagescan in such a way on the screen that one image pixel corresponds to one screen pixel.
Zoom image	If **ON**, zoom is performed by electron or ion optics during a continuous scan and by digital zoom in the bitmap if the scan was stopped.
Cursor grid subset:	
None, 1, 2, 5,10	Select the cursor grid size.
UV display...	This will open the **UV Display Options** dialog. This dialog is common to all UV windows and is described in the chapter **UV Windows**, (⇒ *UV windows* on page 4-4).

2.4.6 Imagescan menu

Imagescan menu item	Function
Continuous	To start a continuous image scan at the current sample coordinates.
Single	To start a single image scan at the current sample coordinates.
Stop	To stop an image scan immediately.

2.4.7 Filter menu

Filter menu item	Function
Matrix filter	This command can be used for post processing of images. It is comparable with visual effect tools in a graphic program. After selection, a dialog opens to type in a matrix or to select a pre-defined matrix from a list. The filter works in such a way that the given matrix is convolved with the image, which is also interpreted as a matrix. The result is then stored back to the image.

Chapter 11
Scan Manager

This chapter gives an overview of the **Scan Manager** functionality, which is used to define the parameters and positioning of intensity measurements. The Scan Manager is relevant in several areas of the control bar. There are a number of procedure types available within the Scan Manager, such as **Linescans**, **Images**, **Image Linescans** etc.

The chapter consists of the following;

- A table summarizing the various procedure types, which can be inserted from the **Scan Manager** dialog into a **Positionlist,** via drag and drop.
(⇒ *Overview of procedure types within the Scan Manager* on page 11-4),
- Background information on **Images** and **Image Linescans**
(⇒ *Images* on page 11-5),
(⇒ *Image Linescans* on page 11-5),
- Explanation of the principles of **Automatic Writefield Alignment with Images** and **with Linescans**, by inserting the procedure into the Positionlist.
(⇒ *Functional principle of Automatic Writefield Alignment* on page 11-6),
(⇒ *Automatic Writefield Alignment with Linescans* on page 11-7),
- Functional principle of the **Adjustment** procedure.
(⇒ *Functional principle of Adjust UV* on page 11-8),
- Explanation of the **GDSII Writefield Mark Scan** procedure.
(⇒ *Functional principle of GDSII Writefield Mark Scan* on page 11-9),
- The principles of **Beam Tracking Alignment** for **FBMS** Element patterning procedures.
(⇒ *Functional principle of Beam Tracking Alignment procedure* on page 11-10),
- Using a **Linescan** and how to edit Linescan parameters.
(⇒ *Editing a Linescan* on page 11-11),
- Evaluating a **Linescan**, after it has been scanned.
(⇒ *Setting the parameters for Linescan Evaluation* on page 11-12),
- How to use an **Image Filter** procedure and how it can improve results.
(⇒ *Performing an Image filter procedure* on page 11-13),
- Explanation of the **Image Mark Detection Filter** and how to apply it.
(⇒ *Working with the Image Mark Detection filter* on page 11-13),

- Detailed description of the dialogs for the **Scan Manager**.
  (⇒ *Scan Manager dialog* on page 11-15),
- Description of the procedures available in the **Scan Manager**
  (⇒ *Linescans* on page 11-17),
  (⇒ *NanoLinescans* on page 11-24),
  (⇒ *Image Mark Detection Filter* on page 11-31),
  (⇒ *Writefield Alignment procedures* on page 11-33)
  (⇒ *GDSII Writefield Mark Scans* on page 11-41),
  (⇒ *Beam Tracking Alignment procedure* on page 11-44),
  (⇒ *Adjust UV procedures* on page 11-45),
- **Menu** commands within the Scan Manager dialog
  (⇒ *Scan Manager menu* on page 11-50),
- The principle of the **drag and drop** function, used to insert Scan Manager procedures into the Positionlist.
  (⇒ *Drag and drop function* on page 11-55).

The **Scan Manager** dialog cannot be specifically selected via an icon in the control bar. It will be automatically displayed when required, following the selection of one of the other icons in the control bar. Examples include:

- **Writefield Manager**

- **Adjust UVW**

- **Automation**

The dialog for the Scan Manager is displayed to the right, next to the control bar, if any of the above working areas have been selected.

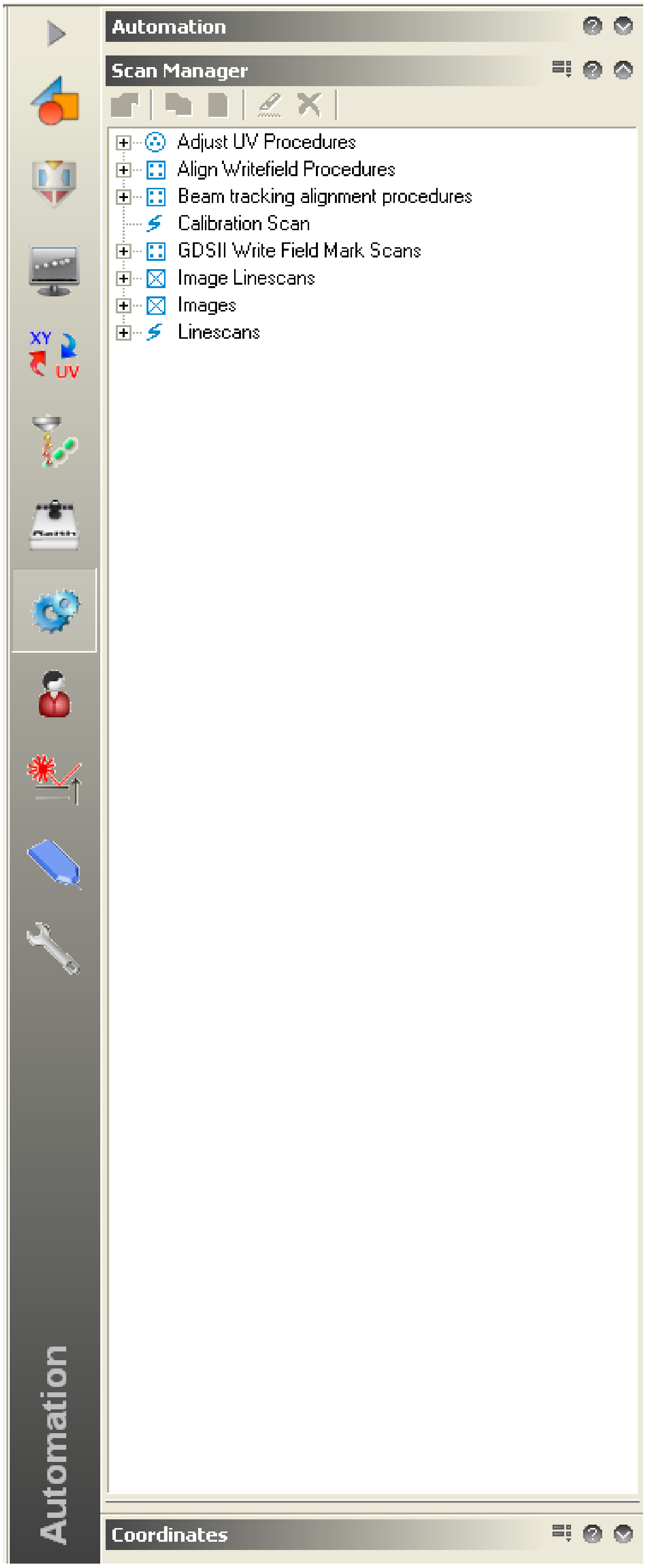


Figure 11-1: Scan Manager

A Functional Description

1 Overview of procedure types within the Scan Manager

Procedures	Function
Adjust UV Procedures	Carries out UV adjustment.
Writefield Alignment Procedures	Writefield alignment procedures are performed to align the writefield in UV prior to patterning.
Beam tracking alignment procedures	When the Beam tracking option is installed, the beam tracking alignment needs to be carried out prior to patterning for placement correction during stage movement.
Calibration Scan	For measurement of widths of structures when the exact line width determination is critical, e.g. in CD measurements.
GDSII Writefield Mark Scans	Writefield alignment procedures that are executed during the patterning process
Image Linescans	Used for measurement of positions or distances in images. Measurements are possible in an off-line manner.
Images	Images are scans acquired in two dimensions, used for recording images, for example for off-line inspection or documentation purposes.
Linescans	Scans made in one direction by an electron beam or ion beam. These scans are used for measurement of positions and distances between structures. Linescans can also be used for line width measurements, if an exact line width determination is not critical.
NanoLinescans	Defines the linescans for the NanoSense option, only available when the NanoSense is installed. All standard functions of the Linescans can also be used for the NanoSense. Using the NanoLinescan, only the Z values will be measured using a defined linescan. This will result in a topographic linescan, where the different heights of the sample surface along a linescan are measured. These values can then be used to set the nanomanipulators to precise Z values onto the sample surface along a linescan.

Scans can be placed within all UV windows or a Positionlist by using the drag and drop technique. When using a UV window as a destination, the scan is stored in all open Positionlists. If no Positionlist is open, one is generated automatically.

After recording the intensity, it is possible to perform post processing, such as noise reduction, as well as a subsequent evaluation, for example peak detection.

Procedures written by Raith specialists are supplied with the instrument for use as soon as the system has been installed, before the user has written any new procedures.

2 Images

The following section gives background information on Images.

2.1 Background information on Images

An image is recorded by making intensity measurements in two scan directions, measuring the intensity at each defined point. Post processing and evaluation are again available. The term **Image** refers to a standard image obtained by either electron or ion optics. It may be obtained by either a manual or automated procedure. The image size may be chosen, but may not exceed the size of the writefield. At each point it is possible to measure the intensity several times, for averaging purposes.

After recording the intensity it is possible to perform post processing, for example noise reduction, as well as a subsequent evaluation, for example peak detection.

3 Image Linescans

The following sections describe the Image Linescans.

3.1 Background information on Linescan

An **Image Linescan** is a **Linescan** which is placed on an image that has already been obtained. The resulting linescan is based on the gray scaling of the image.

This differs from a normal linescan which is performed by the beam according to parameters set specifically for that linescan.

The disadvantage of placing a linescan on an existing image is that the resolution of the linescan will be limited by the image itself. An 8-bit image has 256 gray tones, which is the limitation for any linescan placed on that image. Normal linescans have a gray scale resolution of 12 bits, corresponding to 4096 gray tones.

4 Functional principle of Automatic Writefield Alignment

The following sections describe the different writefield alignment procedures.

4.1 Automatic Writefield Alignment with Images

When the **Create reference image first** checkbox is checked, a two dimensional reference image is acquired at the start. The reference image is compared with a later acquisition of the same feature. This comparison is used to verify that the alignment has not changed throughout the execution of a procedure.

The user has the choice to open a directory and specify the reference image that should be used, or let the software create it automatically.

When the reference image is created automatically by the software, only **Rotation** and **Zoom** values can be corrected. **Shift** values may not be verified, since if a shift has occurred, the software will not be able to recognise the feature when a new image of it is acquired again.

To correct for **Rotation**, **Zoom** and **Shift**, the option **Create reference image first** should not be checked. During the mark detection procedure in the **Post processing** tab, the **Mark detection filter** should be selected. The images acquired will be automatically used by the software for mark recognition. The user then has the choice to select one of the images obtained as the software reference image. The parameter set to be used may be selected via the **Mark detection filter** in the **Post processing** tab, (⇒ *Working with the Image Mark Detection filter* on page 11-13).

If the option **Halt before closure** is active, the automatic procedure stops execution after each mark scan. Do not select this feature, if a fully automatic procedure is required.

4.2 Automatic Writefield Alignment with Linescans

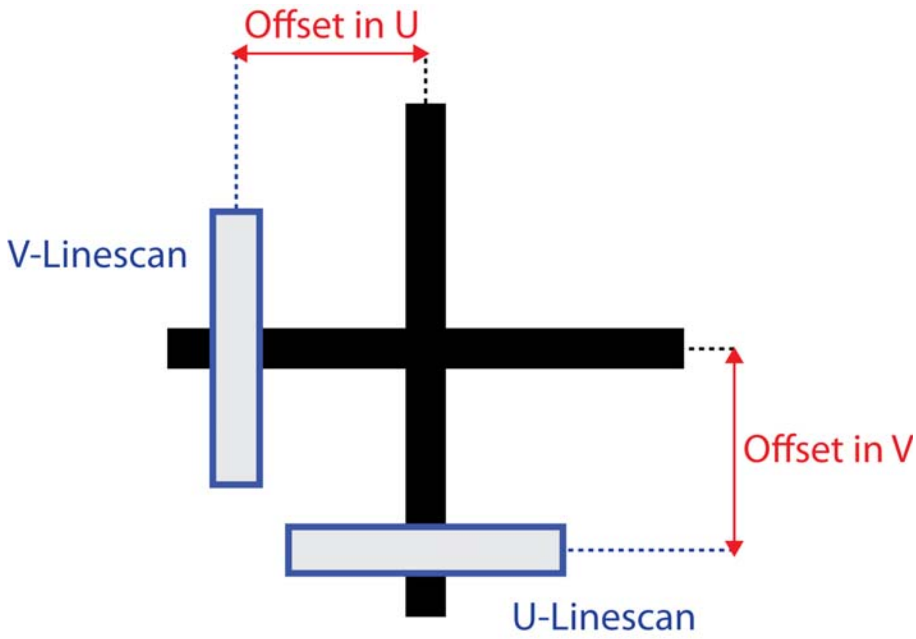


Figure 11-2: Placement of Linescans with respect to a mark using the offset

Automatic Writefield Alignment with Linescans must be applied to pre-fabricated cross shaped structures. Two linescans are scanned across the sides of a structure as shown in the illustration above. **Offset U** and **V** define the distances of corresponding linescans from the center of the cross.

Whenever a clearly defined cross structure is available, An **Automatic with linescans** writefield alignment procedure may be even more precise than using an image. This is because linescans use 12-bit resolution, which is significantly higher than the 8-bit resolution used for images. This higher resolution can lead to greater accuracy in the definition of the position by the software.

5 Functional principle of Adjust UV

Adjust UV can be carried out using a 3-points adjustment, either manually or automatically. It is recommended to begin with a coarse manual 3-points adjustment and then continue to automatic procedures. When, for example, a full wafer is to be written and the stage must drive to individual chips on the wafer, which contain reference marks, it is recommended to perform **Local** 3-points adjustment, since very high accuracy will be required. Here, automatic procedures are advised.

There is a semi-automatic procedure available, for use with images. An image can be selected and up to three marks may be defined.

The adjustment procedure executes the following steps. The stage drives to all specified UV mark positions and records an **Imagescan** or a **Linescan** at each position. From the deviation between actual and reference positions, a new coordinate transformation is calculated and applied. If a manual procedure is executed, the operator must define the deviation using the **Control** key and **left mouse button**.

The position of each mark is specified in **U** and **V**. Thus a coarse UV adjustment must be executed or applied to the sample before using a procedure. This can be a 3-points adjustment by **Adjust UVW** or an unpatterned wafer adjustment. An adjustment procedure can then be used for a fine adjustment.

Within the **Scan Manager**, three types of adjustment procedures are available: **Manual**, **Automatic with Linescans** and **Automatic with Images**. The Manual and Automatic with Images procedures use images to determine the shift at each mark. The Automatic with Linescans procedure uses Linescans in U and V directions and requires a cross-shaped mark.

The details of how to configure **Imagescans** used for the procedures **Manual** and **Automatic with Images** can be found in the section **Defining Images**. Refer to the corresponding section **Defining Linescans** to learn how to configure the linescans used for **Automatic with Linescans** procedures. In the following, only deviations from these settings are described.

5.1 Automatic Adjust UV with Images

The mark scan size will be determined. Up to three marks can be obtained and the option **Adjust after every mark** may be selected, which initiates **Shift** adjustment after the first mark, **Rotation** adjustment after the second mark and finally **Zoom** adjustment after the third mark, if all three marks have been selected. Unlike the writefield alignment procedures, the images may not be created first. The images must be selected by the user and they need to be defined in the software. It is possible to choose a different image for each mark. It is also possible to use the same reference image for all marks.

The precision of an adjustment depends on the accuracy of the position of the reference image. Hence, when recording the reference image, it is essential to first perform a very accurate manual adjustment as well as a writefield alignment.

6 Functional principle of GDSII Writefield Mark Scan

Manual GDSII writefield mark scans are written in the same way as **Manual GDSII Layer based Mark Scans**. The **Automatic GDSII Writefield Mark Scans**, both with images and linescans are written in the same way as the **Automatic GDSII Layer based Mark Scans** with images and lines. The difference is that in the case of GDSII Writefield Mark Scans, more parameter sets can be defined. This flexibility is helpful, since linescans of differing sizes, post processing and evaluation settings are required, depending on each application and evaluation type.

The advantage of the **GDSII Layer based Mark Scans** is that they are defined within the GDSII structure, whereas the **GDSII Mark Scans** are defined in the **Scan Manager** dialog. If a different user on another PC does not have the GDSII Writefield Mark Scan procedure stored in the Scan Manager, the mark scan procedure can not be executed.

GDSII Layer based Mark Scans/Manual with Images and GDSII Writefield Mark Scans/Manual are automatically assigned to layer 63 and **GDSII Layer based Mark Scans/Automatic with Linescans and GDSII Writefield Mark Scans/Automatic with Linescans** to layer 61. **GDSII Writefield Mark Scans/Automatic with Images** are assigned to Layer 62. Layers 63, 62 and 61 are exposed before all other layers of the GDSII design. In this way, every writefield is calibrated first by mark scans and the real patterning is started afterwards.

7 Functional principle of Beam Tracking Alignment procedure

The stage is the principal means of positioning the beam on the sample. The stage does have some very small inaccuracies, arising from the resolution of the stage, as well as hysteresis and acceleration/deceleration effects during the movement. This means that the position actually obtained by the stage for each dot in a pattern is not exactly the target position. Beam tracking addresses these small errors in stage positioning and further enhances the accuracy of beam position on the sample. When applying the **Beam Tracking Alignment**, the beam is deflected by the pattern generator in a way calculated to continuously compensate stage displacement in the closed loop control mode. This helps to decrease the stage settling time.

The **FBMS** and **MBMS** options also require **Beam Tracking Alignment** to achieve high accuracy of patterning FBMS structures. During the Beam Tracking Alignment, **Zoom** and **Rotation** are compensated by the stage controller and **Shift** is compensated by the pattern generator. This is different to the **Writefield Alignment**, in which all aspects of control are performed by the pattern generator.

Beam tracking alignment is very similar to writefield alignment, but the key difference is that beam tracking alignment is made in XY, whereas writefield alignment is made using UV coordinates. There are **Manual** procedures, as well as **Automatic with Linescans** and **Automatic with Images** procedures available. These can all be prepared and applied in a similar way to the automatic Writefield Alignment procedures.

B Tasks

1 Editing a Linescan

When the option **Linescans** is selected in the **Scan Manager** dialog, a list of saved procedures is displayed, from which a procedure can be selected and edited, if required.

In order to insert a new linescan, highlight a procedure or sub-procedure and click on the **Create new scan** button. You may also use a **right mouse** click and select **New** from the dropdown list. The **Scan Properties** subdialog will open, in which the user may enter the name of the parameter set.

- In order to clearly identify the kind of linescan that has been saved, the following nomenclature convention should be used to name the file. The file name should include the writefield size, then the word **linescan**, then the main axis, either **U** or **V**, and the length of the linescan. When the linescan is performed at an angle to the main axis, also include the angle in degrees.

- As an example, a linescan which has been previously created and saved using the name <**100 µm WF - linescan U 10.0 µm 30 deg**> is a 10 µm length linescan in a 100 µm writefield, at 30 degrees to the U axis.

If a linescan is executed in a writefield that is too small, an error message will be displayed and the procedure will not be executed. A linescan designed for a 100 µm writefield clearly can not be written into a writefield with a 50 µm dimension.

- When a linescan is configured, including the number of points and the length, the configuration is only valid for the selected writefield. This is the reason for the nomenclature, so that the size and orientation of the structure are included in the name. This allows the user to easily identify the key features from the parameter set name.

- After the **Name** of the parameter set has been entered, the user can edit the four tabs **Main**, **Advanced, Post Processing** and **Evaluation,** which will be explained sequentially.

The user may define the number of points, the step size and the width of the scanned field.

Scan size and **Step size** are quantities that are normally dictated by the scan task itself, and therefore should be defined first. **No of points** can then be calculated using <No of points> = <Scan size> /< Step size>.

As well as the largest dimension and the main direction of the linescan, the width may also be defined. It is possible, for example, to define a square linescan. In all cases, the scanning direction will be parallel to the specified main scan direction.

As an illustration, the main axis may have been defined as U and the number of points in U might be 2500. If the V direction is defined as 25, then 25 lines of 2500 points will be scanned.

The step size in V defines the distance between lines. If the step size in V is defined as zero, all 25 scanned lines will be in the same position.

The step size for a linescan will influence the resolution of the scan. The smaller the step size, the better the resolution. The **Point average** value affects the dwell time at every single point of the linescan.

2 Setting the parameters for Linescan Evaluation

- Select a Linescan.
- Double click on the procedure to open the **Scan properties** dialog.
- Select the tab **Evaluation**.
- Check the checkboxes **Check Amplitude**, **Peak detection**, **Edge detection**, **Pitch detection**, and **Cross correlation**. Also check the **Check detected width** checkbox to evaluate some of the Linescan properties. The functionality is described in the chapter **Linescans**. When using Peak detection, Edge detection or Pitch detection it is possible to determine which quantity is stored by selecting either **Store position** or **Store width**. The position is the center between the two edges whereas the width is the distance between the two edges.

If, for example, there are two pitches within the linescan, the edge detection will be performed twice and the distance between both pairs of edges will be measured.

- Click the **OK** button to quit the Scan properties definition and to save the new parameter set.

The user may wish to perform an additional evaluation step, e.g. check detected width. If the detected width is not within the given limits, the evaluation will fail.

3 Performing an Image filter procedure

- Select an **Image** procedure in the **Scan Manager**.
- Double click on the procedure to open the **Scan properties** dialog. Select the tab **Post Processing**.
- Choose **Image filter** to modify the image directly after recording it. The functionality is described in the section **Image Linescans**, (⇒ *Background information on Images* on page 11-5).
- After recording and optionally post processing, it is possible to detect a structure within an image and to determine the deviation of a structure with respect to a reference structure.
- Click the **OK** button to quit the **Scan properties** dialog and to save the new parameter set.

An alternative to post processing is to scan the full Positionlist and apply a post processing filter afterwards. Using the **Post Processing** tab is simpler than using a post processing filter after the full Positionlist has been completed. This facility does, however, allow the user to use revised parameters for image processing after all the images have been obtained.

4 Working with the Image Mark Detection filter

To open the **Mark Detection Filter**, go to **Scan Manager** and double click on an Image procedure to open the **Scan Properties** dialog. Select the **Post Processing** tab and click on the button. This will open the **Image Mark Detection Filter** dialog.

The **Image Mark Detection Filter** is a two-dimensional filter, whereas cross correlation for linescans is one-dimensional, (⇒ *Image Mark Detection Filter* on page 11-31).

The mark detection filter compares two images. The first aspect of the comparison is to check if a structure is present and the second aspect is to measure how far the structures in the two images are shifted with respect to each other.
The algorithm must be robust enough to be able to overcome:
- contrast and brightness variations
- image distortions, such as rotations and differences in scaling
- visibility variations, e.g. structures might be only partially visible or even missing
- background noise, which may affect the comparison.

- Go to **Scan Manager** and double click on an **Image** procedure to open the **Scan Properties** dialog.

  Select the **Post Processing** tab and click on the button. This will open the **Image Mark Detection Filter**.

- Select the name of the **Parameter set** and select the **Reference** and if required, enable the noise reduction or background suppression. Confirm changes by pressing the **OK** button.

5 Using Mark Detection in a Writefield Alignment procedure

- The same procedure is used when a **Writefield Alignment** is carried out using Images, (⇒ *Writefield Alignment procedures* on page 11-33).
- If the checkbox **Create reference image first** is checked, the reference image will be created first. If this option is unchecked, then the **Mark Detection Filter** is used, utilizing the reference image. The user can select the reference image already acquired.
- The writefield alignment will then be initiated. It may be that execution is unsuccessful, for example, if the image quality is not high enough, e.g. if the noise is too high or the shift was too large. In such cases, the user can adapt the setting to improve the quality using the **Image Mark Detection Filter** dialog, (⇒ *Image Mark Detection Filter* on page 11-31).

In order to be able to compare the structures in the image with the structures in the reference image, ensure that the structures are definitely comparable. For example, a comparison would be completely incompatible, if a rectangular structure is being compared with a circle.

- Use **Invert image** to create a negative of the selected reference image.
- **Background suppression** can be applied to improve the quality of the comparison. When the quality of the images is poor, or if image qualities are dissimilar, the result of the comparison can be affected, regardless of any other aspects, such as shift. Poor background on one or both of the images can influence the comparison in a negative way and the resulting value (%) yielded for the quality of the comparison will be lower. When background suppression is used, the grayscales below the value on the sliderbar will be deleted. This might improve the comparison between the image and the reference image.

A quality of 95% should be expected. This can be set in **Extras➡Settings➡Image➡Cross correlation quality threshold** = 0.95.
This setting is the lower acceptable limit for quality control purposes. If the quality is lower than 95%, then there is probably some significant mismatch in the mark detection procedure.

C Software Reference

1 Scan Manager dialog

The **Scan Manager** is a supporting dialog, which allows the definition of all procedures. Within the Scan Manager, a range of procedures can be defined. A selection of written procedures is available for the user to choose and edit.

A tree view enables the viewing of all folders and subfolders, containing the written procedures.

Procedures are available to perform the following tasks:

- Adjust UV
- Writefield Alignment
- Beam Tracking Alignment
- GDSII Writefield Mark Scans
- Image Linescans
- Images
- Linescans
- NanoLinescan

Any procedure can be dragged and dropped into a Positionlist and then executed.

Please read the **Positionlist** chapter, to familiarize yourself with the usage of the Positionlist, (⇒ *Positionlist* on page 9-1).

These procedures are then used during more complex tasks, such as dimensional metrology tasks or writefield alignments. These tasks are described in other chapters. Adjusting the sample to the stage is described in the chapter **Stage to Sample Adjustment,** (⇒ *Stage to Sample Adjustment* on page 12-1), aligning a writefield is described in chapter **Beam to Sample Alignment,** (⇒ *Beam to Sample Alignment* on page 13-1) and, performing a patterning procedure, in chapter **Patterning,** (⇒ *Patterning* on page 14-1).

1.1 Tool bar

You may use the tool bar to select the commands to open or create new procedures and to edit them.

You can also access the current executable commands via the context menu.

Figure 11-3: Tool bar

Control element	Function
	To open the **Scan Properties** dialog, in which the user can edit all parameters.
	To duplicate the selected scan
	To create a new scan based on the default parameters stored in the software.
	To rename the selected scan
	To delete the selected scan
	The options button is only available for specific procedures.

When the **Writefield Alignment** procedure is activated, an additional button is available in the tool bar.

Click on the  button. The **Writefield Alignment Options** dialog will open. In the **Options** dialog, the user can determine by how much the writefield alignment values have changed. Delta values show if alignment deviates significantly from the set values. The dialog is explained in **Beam to Sample Alignment**, (⇒ *Writefield Alignment dialog* on page 13-18).

1.2 Scan Manager working area

Parameters can be individually stored for the various scan types. There is a hierarchical organization within the **Scan Manager** section of the control bar. For dimensional metrology tasks, the following general scan types are available.

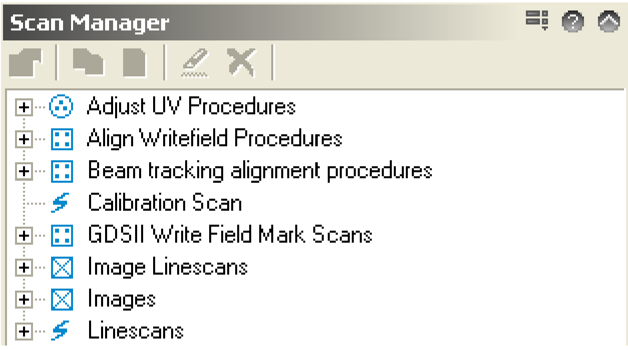


Figure 11-4: **Scan Manager** working area

1.3 Scan properties subdialog

Selecting the button will open a new **Scan properties** subdialog, in which the parameters can be edited. There are four tabs available to define parameters:

- **Main**

- **Advanced**

- **Post Processing**

- **Evaluation**

The **Scan properties** subdialog will be described in the sections **Images**, **Image Linescan** etc, since the content of the **Scan properties** window will depend on which section is being considered.

2 Linescans

A **Linescan** is recorded by moving the beam along a linear path, measuring the grayscale intensity at each defined point along the scanned path. This scan direction can be along, or at an angle to, either the U axis or V axis, one of which is described as the **Main direction**. In addition, it is possible to use the average of the intensity of several beam movements by defining a scan size perpendicular to the scan main direction. The resulting Linescan is a single waveform showing the measured intensity versus a scan direction.

2.1 Scan properties: Main tab - Linescan

The **Main** tab consists of the following parameters:

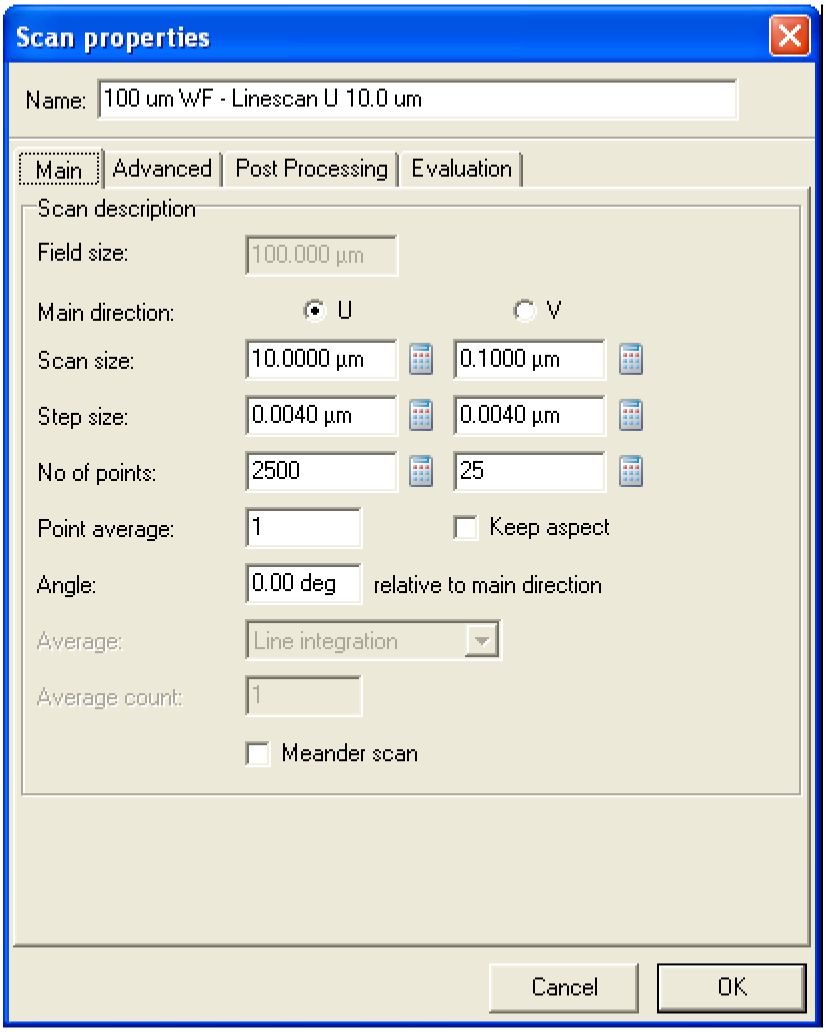


Figure 11-5: **Main** tab for Linescans

Control element	Function
Field size	The field size is automatically displayed as defined in the Writefield Manager.
Main direction	Select the primary scan orientation, either the horizontal U direction, or the vertical V direction.

Control element	Function
Scan size	Enter the length of the scan in µm. In general, the scan size in the main direction is larger than in the averaging direction. Checking the **Keep aspect** checkbox ensures that if one of the scan sizes is changed, the other is automatically scaled, such that the aspect ratio between them is constant. For a finite scan size, the number of Pixels must be larger than 1. The maximum scan size is the current Writefield size.
Step size	It is possible to define the distance between two adjacent points in µm. For the main direction, the step size defines the minimum resolution of the Linescan. For the averaging direction, this defines the distance between two separate beam movements which are then summed for averaging. Step size is a multiple of the **basic step size**.
No of points	It is possible to define the number of points in the main direction, as well as in the averaging direction. The maximum scan size in pixels is 4096 minus the number of pixels used to compensate dynamic effects at the start of the scan. **Scan size, Step size**, and **No of Points** are interdependent, because the Scan size is the product of the other two. Consequently, the third value can be calculated after setting the two other quantities. Use the calculator button behind the value to perform the calculation. A message indicates if these three values are valid and that they conform to this relationship. If so, confirm these settings by pressing **OK**.
Point average	Signal averaging allows measurement of the intensity at the same position several times, thereby enhancing the signal-to-noise ratio. The number of times that a scan should be repeated is entered and when the scan has been performed that number of times, the average values for the gray scale are automatically calculated. Point average defines how many times the scan is repeated at any one point.
Keep aspect	Check this checkbox if you wish to keep the aspect ratio between U and V constant. When, for example, the value for U is doubled, the value for V will also double.
Angle	Input the angle relative to the **Main direction**, at which to record a Linescan. Since the angle is defined relative to the main direction, it may only be varied in the range from -45° to 45°.

Control element	Function
Meander scan	Check this checkbox if you wish to use a Meander scan pattern, in which the beam is not blanked between lines and the whole scan is performed with continuous movement.

2.2 Scan properties: Advanced tab - Linescan

Choose the **Advanced** tab to define additional parameters.

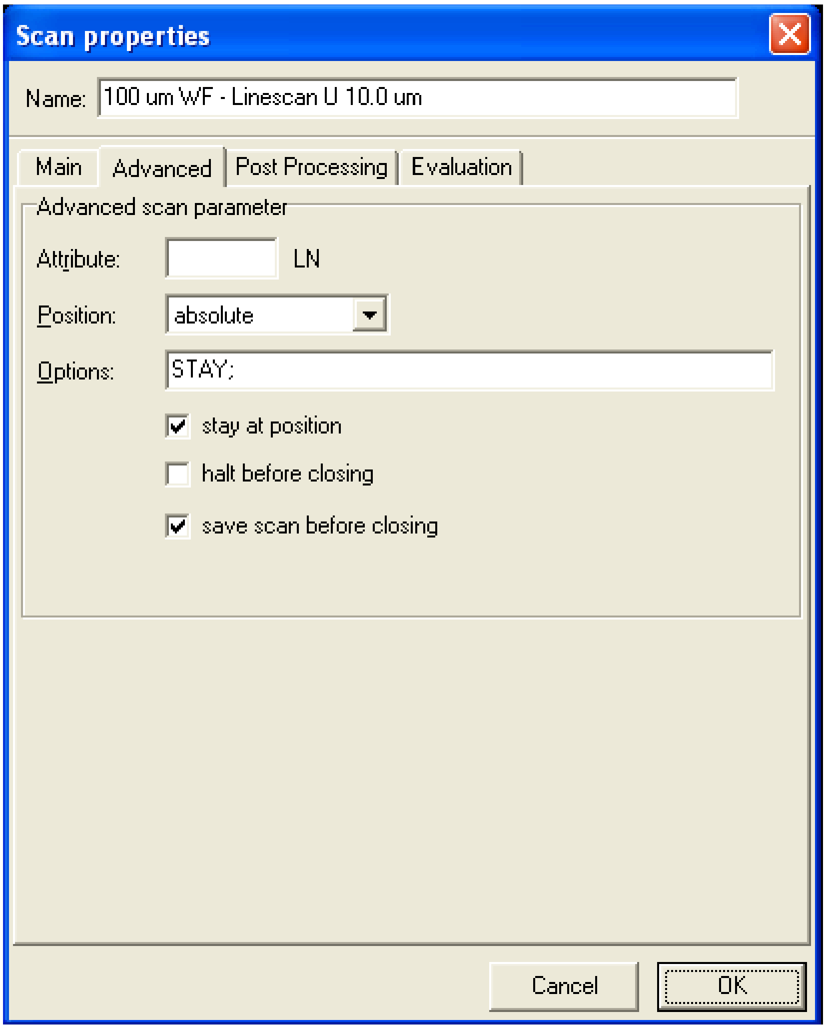


Figure 11-6: **Advanced** tab for Linescan

Control element	Function
Attribute	Enter a character to be added to the **Attribute** column of the Positionlist, when dropping the scan into a target window. Some characters have a special meaning and cannot be used. By default, the character **L** is used to indicate that the scan is defined as a Linescan and the character **N** marks it as a new one. If you use characters to denote some other attribute, ensure that **L** and **N** are not used in connection with this additional meaning, since they must only be used in the way described above.
Position	An **absolute** or **relative** position may be selected. Absolute values refer to the stage movement. Relative positioning refers to the stage movement relative to the current stage position. After the linescan has been inserted from the Scan Manager into the Positionlist via drag and drop, the coordinates of the centers of the Linescans are defined in the Positionlist, not in the Scan Manager.
Options	Within this input field, the user may define some options for the execution of the scan. For a typical Linescan, the **STAY** option is the most important one. This option is selected automatically when the **stay at position** checkbox is checked.
stay at position	Check this checkbox to prevent the stage from driving away from the selected position. The stage will not move before the scan is performed with the stage at the current position. The beam will be deflected to match the target position.
halt before closing	Check this checkbox to halt after completion of the linescan, without closing the window. This is very useful for diagnosing potential errors in parameter selection.
save scan before closing	To save the linescan before closing the window.

Click on **OK** to quit the scan property definition and to save the new parameter set.

2.3 Scan properties: Post Processing tab - Linescan

Very often, both recording and processing are required. The set up sequences of these parameters are described here.

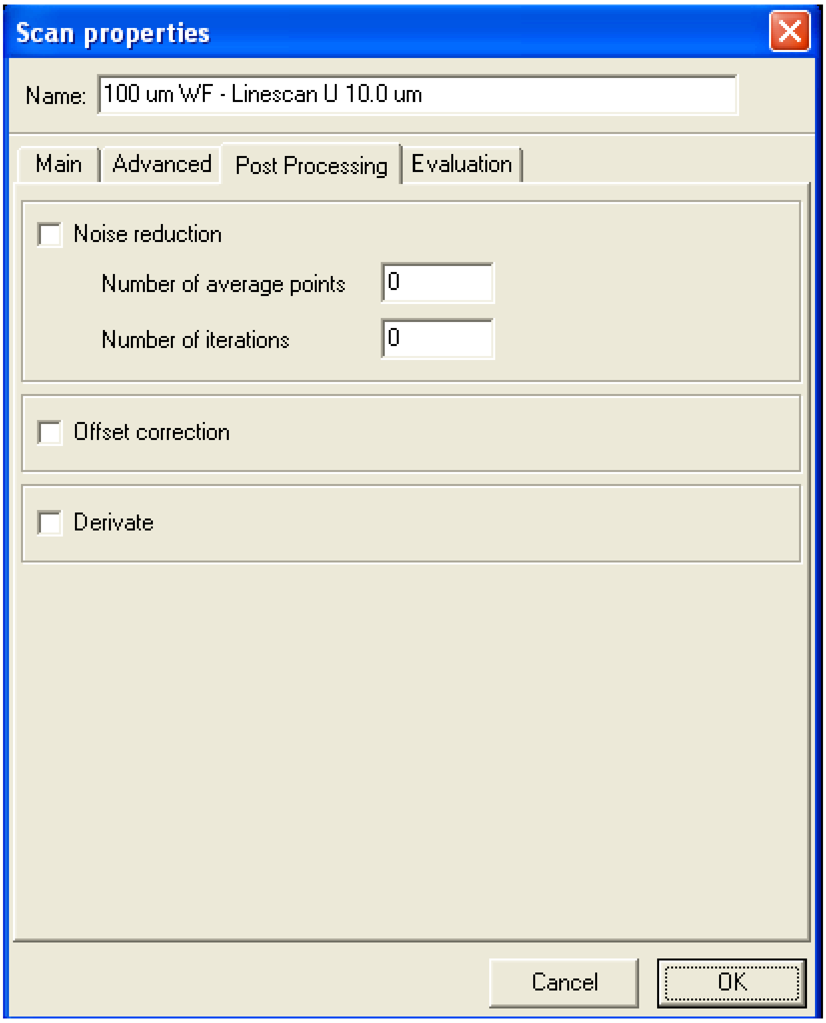


Figure 11-7: **Post Processing** tab for Linescans

Control element	Function
Noise reduction	The user may select the parameters for noise reduction after completion of the linescan. The user may select the number of points for averaging and the number of loops or iterations to be used. Check the checkbox to activate noise reduction.
Number of average points	Enter the number of average points.
Number of iterations	Enter the number of iterations.

Control element	Function
Offset correction	Check the checkbox to perform offset correction.
Derivate	Check the checkbox to calculate the vector derivative of the linescan.

2.4 Scan properties: Evaluation tab - Linescan

After recording and optionally post processing, linescans can also be evaluated for line width, position, and distance measurements. The user may select from a list of stored algorithms, such as **Edge detection**, **Peak detection**, **Pitch detection** and **Cross correlation**. Refer to the **Linescan** chapter for further explanation, (⇒ *Threshold Algorithm* on page 10-19).

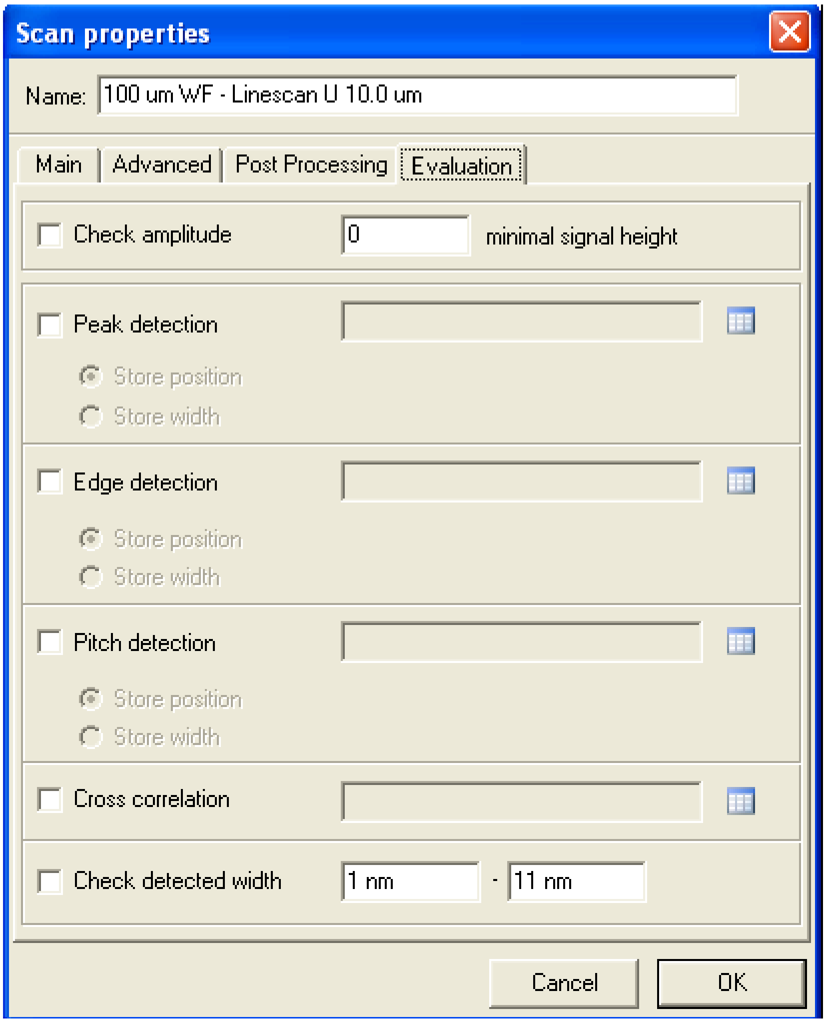


Figure 11-8: **Evaluation** tab for Linescans

Control element	Function
Check amplitude	Check the checkbox to activate the amplitude check. Enter a value for the minimum amplitude. Linescans for which the maximum gray scale value is less than the minimum signal height will not be evaluated. The scan in the Positionlist will be marked red to indicate an error.
Peak detection	Check the checkbox to perform a peak detection. Select a suitable parameter set from the database. - Select **Store position** if you wish to store the filter result as the position. - Select **Store width** if you wish to store the filter result as the width.
Edge detection	Check the checkbox to perform an edge detection. Select a suitable parameter set from the database. - Select **Store position** if you wish to store the filter result as the position. - Select **Store width** if you wish to store the filter result as the width.
Pitch detection	Check the checkbox to perform a pitch detection. Select a suitable parameter set from the database. - Select **Store position** if you wish to store the filter result as the position. - Select **Store width** if you wish to store the filter result as the width.
Cross correlation	Check the checkbox to perform a cross correlation. Select a suitable parameter set from the database.
Check detected width	Check the checkbox to verify the detected width. Enter the minimum and maximum values - the detected width will be checked to ensure that it lies between these entered values.

3 NanoLinescans

In Linescans is the NanoSense Linescan subheader, which defines the linescans for the NanoSense option. This option is only available when the NanoSense option is installed. Clicking on it opens the **Scan Properties**. Only the new options will be described, in comparison to a normal eBeam linescan.

Using the NanoSense Linescan, only the Z values will be measured using a defined linescan. This will result in a topographic linescan, in which the different heights of the sample surface along a linescan are measured. These values can then be used to set the nanomanipulators to precise Z values onto the sample surface along a linescan. Only the additional functions of the NanoSense Linescan will be described here.

3.1 Scan properties: Main tab - NanoLinescan

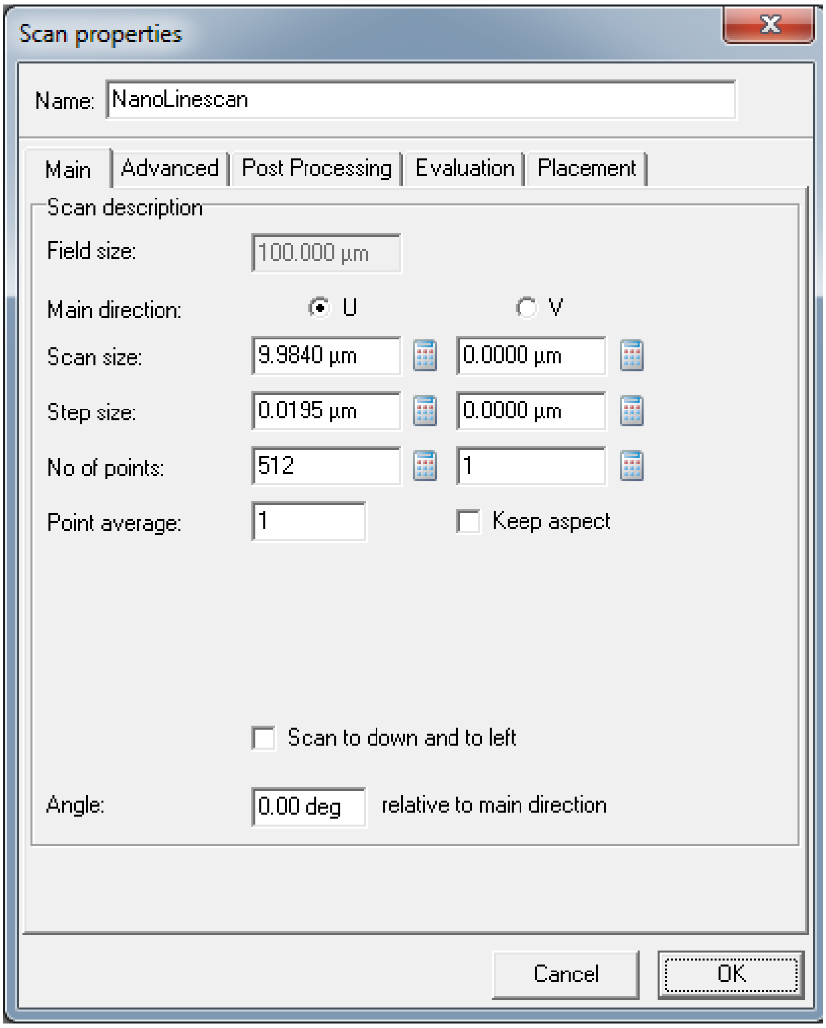


Figure 11-9: Scan properties - main tab

The **Main** tab has a new checkbox **Scan to down and to left**. Checking this will change the scan direction, to the reverse of the standard scan direction; either downwards or to the left. The standard scan direction is from the bottom up or from left to right. Checking this option enables a scan from the top down or from right to left.

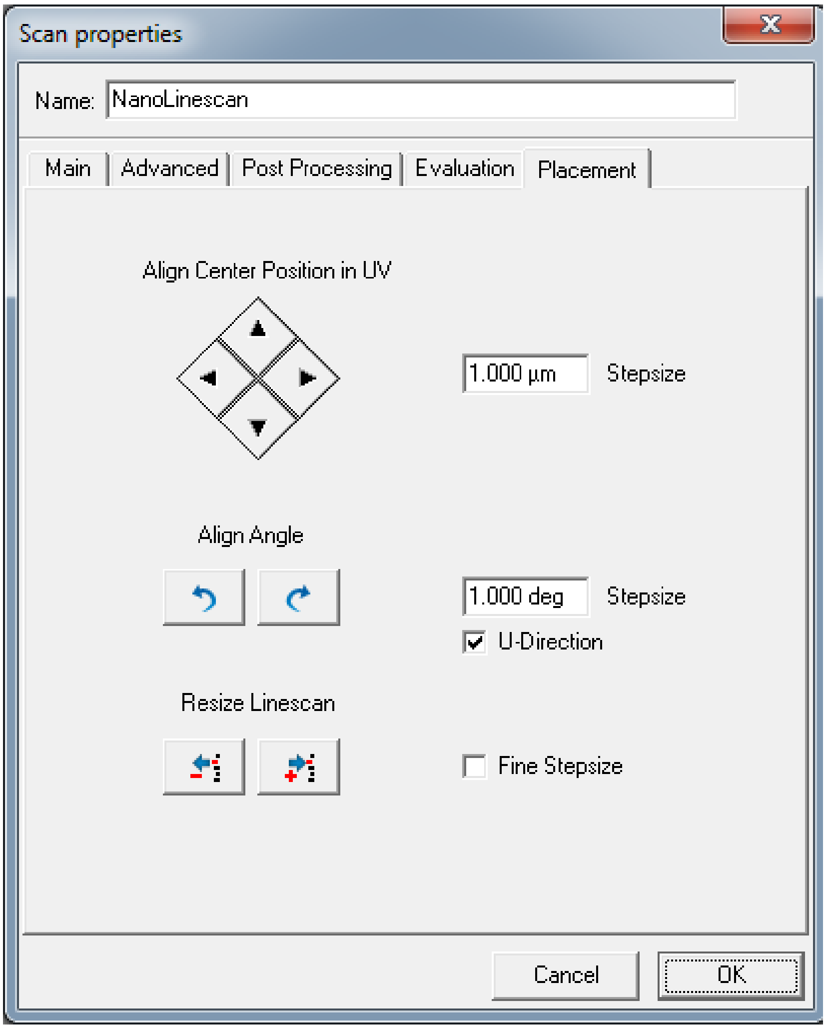


Figure 11-10: Scan properties - Placement

There is and additional tab, **Placement**. Drag and drop the nanomanipulator into an image, then go to Placement in **Scan Properties**. There are now 3 additional options available for Nano Linescans in the dialog:

- **Align in center position in U and V**

  This will use a predefined value to change the position of the linescan in increments. The results will be shown in the image, as well as in the GDSII. The values can also be read in the U and V coordinates in the positionlist and therefore only applicable when procedure has been put into the positionlist before.

- **Align angle**

  This enables rotation of the linescan in increments. The angle is adjusted in increments and will be displayed in **Main > Angle**. The angle can be changed clockwise or anti-clockwise by the increment shown on the right hand side.

U direction can be checked or unchecked. Defines the angle with which the angle definition is started.

- **Resize linescan**
- The length of the linescan can be increased or decreased. **Fine stepsize** can be checked or unchecked. When checked, +/- 2.5% of the current linescan length will be either deducted or added.
- When unchecked, +/-12.5% of the current linescan length will be either deducted or added.
- The stepsize remains the same, only the number of points changes.

4 Images

Images can be scanned, post processed and evaluated using the four tabs in the **Scan Properties** dialog.

4.1 Scan Properties: Main tab - Images

The number of points and the distance between each point may be selected. Please refer to **Scan properties: Main tab - Linescan** for a detailed description, (⇒ *Scan properties: Main tab - Linescan* on page 11-18). Further parameters are available for **Images**.

Fast Image Scan may only be used for Frame routines. Not all average values can be set. The software will automatically check if the input values can be reached. Fast Image Scan in not available for ELPHY *Quantum*™ users.

Control element	Function
Point average	Defines how many times the scan is repeated at each point.
Frame integration, Frame averaging	The selected number of complete images, or frames will be obtained sequentially and the average will then be taken. Frame integration averages each iteration. Frame averaging uses a continuous scan and averages the frames from the beginning.

Control element	Function
Line integration average	Each line will be repeated the selected number of times for averaging, before proceeding to the next line.
Average count	The user can enter the number for the average count. For example, if the number 5 is entered, the frame or line will be scanned 5 times and the average count will be calculated.

4.2 Scan Properties: Advanced tab - Images

Please refer to **Scan Properties: Advanced tab - Linescan** for a detailed description, (⇒ *Scan properties: Main tab - Linescan* on page 11-18).

Control element	Function
Attribute	Enter a character to be added to the **Attribute** column of the Positionlist.
Position absolute/ relative	Image center position is interpreted as the absolute position or position relative to the current stage position.
stay at position	When the checkbox is unchecked, the stage will go to the position and the scanning of the beam will be performed. The stage remains at the current position and the beam is deflected to the target position, when the checkbox is checked. This can lead to an error if the stage is not at the required UV position when the scan starts. An error message **Image not scanned** will be displayed.
halt before closing	To halt before closing the window
save scan before closing	To save the image before closing the window, for example for documentation and evaluation.

4.3 Scan Properties: Post Processing tab - Images

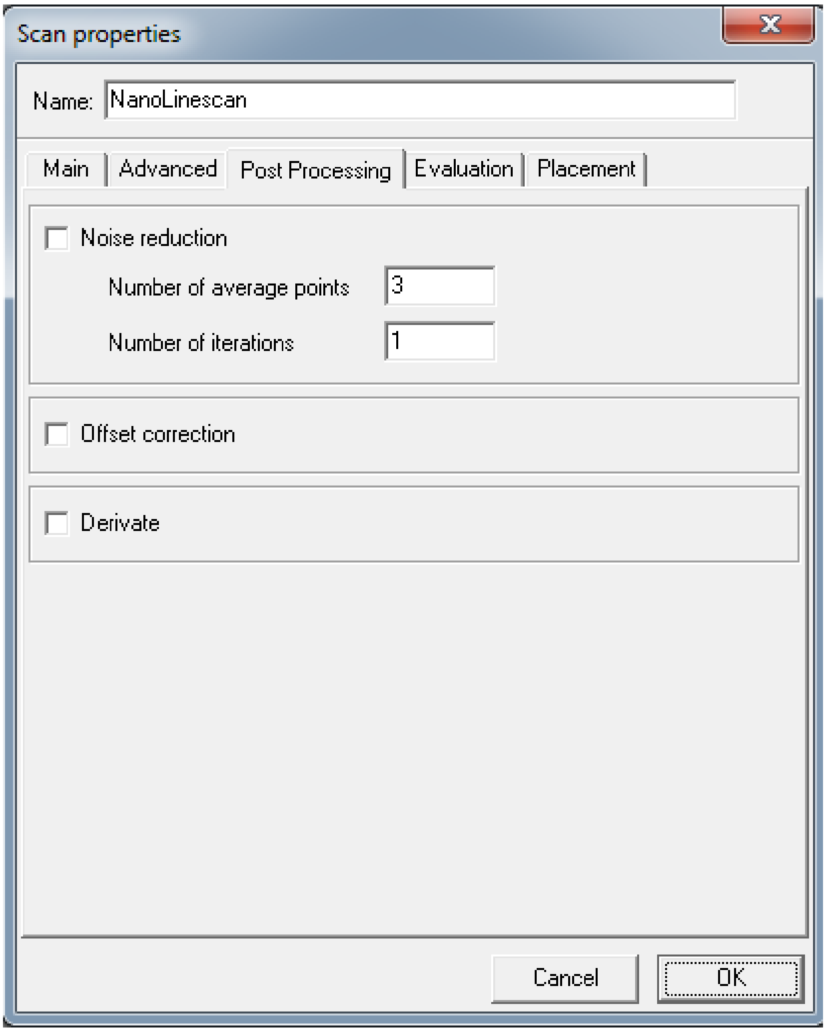


Figure 11-11: Scan properties - Post Processing

Post processing is initiated after an image has been obtained. The image will be adjusted according to the chosen parameters.

Control element	Function
Image filter	Various matrix filters may be chosen. Check the checkbox to activate an image filter process. In the **Image Matrix Filter** dialog, either predefined or user-defined parameter sets can be chosen.
Mark detection filter	Select a parameter set containing the relevant mark detection parameters from the database. See the **Scan Manager** chapter for more information, (⇒ *Working with the Image Mark Detection filter* on page 11-13).

5 Image Mark Detection Filter

The **Image Mark Detection Filter** is a two-dimensional filter. The following parameters can be edited.

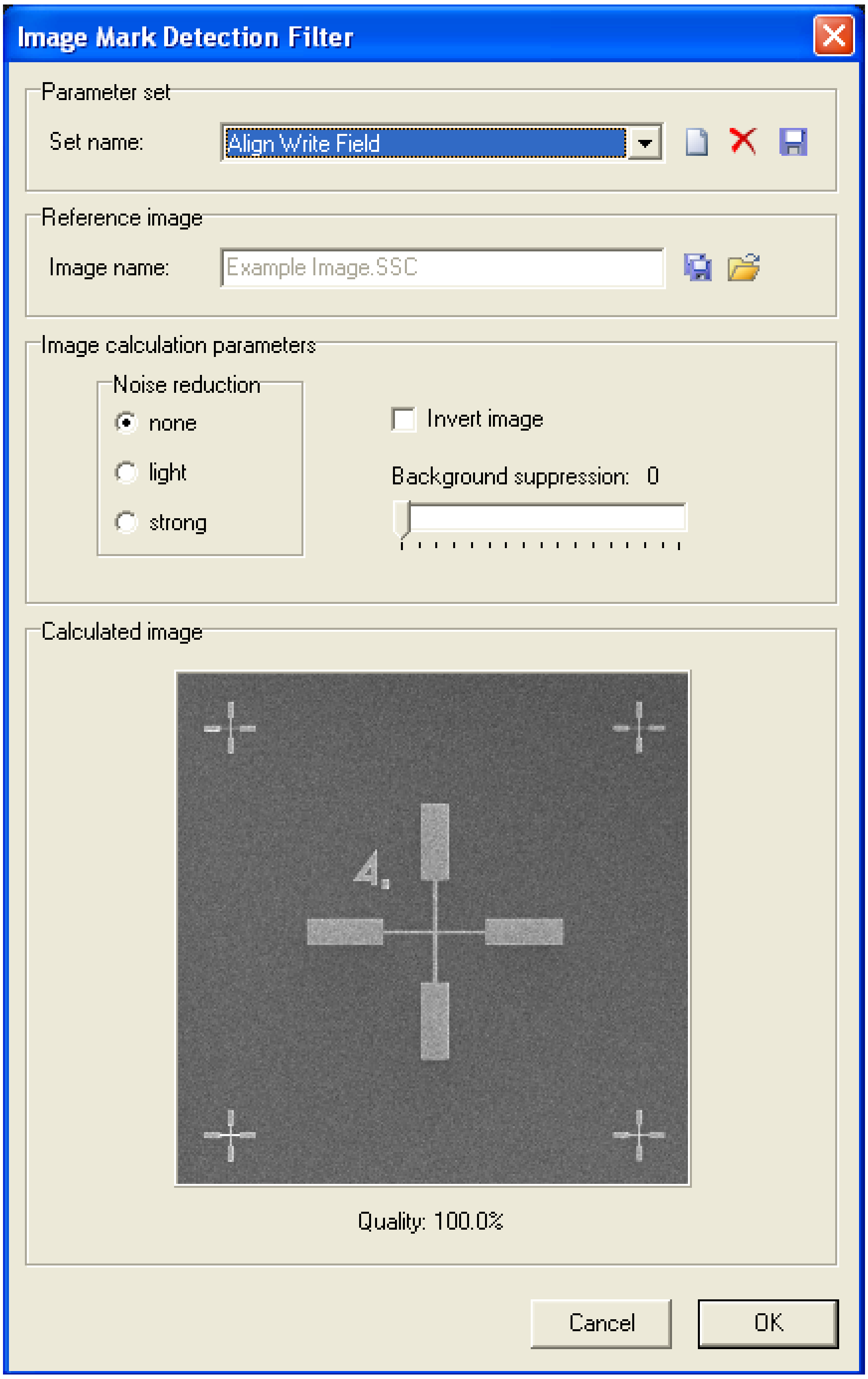


Figure 11-12: Image Mark Detection Filter

Control element	Function
Parameter set subset:	
Set name	Select a **Set name** from the dropdown list.
	To create a **New Parameter set.**
	To delete a **Parameter set.**
	To save a **Parameter set**.
Reference image subset:	
Image name	Select the filename for the image to be used as a reference image.
	To open an existing **Reference image.**
	To save an image as a reference
Image calculation parameters subset:	
Noise reduction	Select between the following noise reduction options: - **None** - **Light** - **Strong**
Invert image	Check the checkbox to convert the image to a reversed gray scale version of the original.
Background suppression	To suppress the background, move the sliderbar. The more you move the sliderbar, the more the background will be suppressed.
Calculated image	The calculated image will be displayed. The **Quality** will be shown underneath. 100% Quality refers to the highest precision of calculation results.

5.1 Scan properties: Main tab - Image Linescan

Please refer to **Scan properties: Main tab - Linescan** for a detailed description, (⇒ *Scan properties: Main tab - Linescan* on page 11-18).

5.2 Scan properties: Advanced tab - Image Linescan

Please refer to **Scan properties: Advanced tab - Linescan** for a detailed description, (⇒ *Scan properties: Advanced tab - Linescan* on page 11-20).

5.3 Scan properties: Post Processing tab - Image Linescan

Please refer to **Scan properties: Post Processing tab - Linescan** for a detailed description, (⇒ *Scan properties: Post Processing tab - Linescan* on page 11-22).

5.4 Scan properties: Evaluation tab - Image Linescan

Please refer to **Scan properties: Evaluation tab - Linescan** for a detailed description, (⇒ *Scan properties: Evaluation tab - Linescan* on page 11-23).

6 Writefield Alignment procedures

There are several subfolders available:

- **Manual**

- **Automatic with Images**

- **Automatic with Linescan**

- **GDSII Layer Based Mark Scans**

Within the **Scan Manager**, several types of writefield alignment procedure are available: **Manual, Automatic with Linescans** and **Automatic with Images**. The **Manual** and **Automatic with Images** procedures use images and perform 2-dimensional mark detections to determine the shift at each mark. The **Automatic with Linescans** procedure uses linescans in U and V directions and requires a cross-shaped mark.

The details of how to configure Image scans used for the procedures **Manual** and **Automatic with Images** can be found in the chapter **Linescans** and **Images**, (⇒ *Scan Manager* on page 11-1).

6.1 Manual Writefield Alignment

Manual writefield alignments are carried out to achieve initial coarse alignment.

6.1.1 Scan properties: Main tab - Manual Writefield Alignment

Please refer to **Scan properties: Main tab - Image** for a detailed description, (⇒ *Scan Properties: Main tab - Images* on page 11-27).

6.1.2 Scan properties: Mark procedure tab - Manual Writefield Alignment

The **Mark procedure** tab enables the user to define up to eight marks.

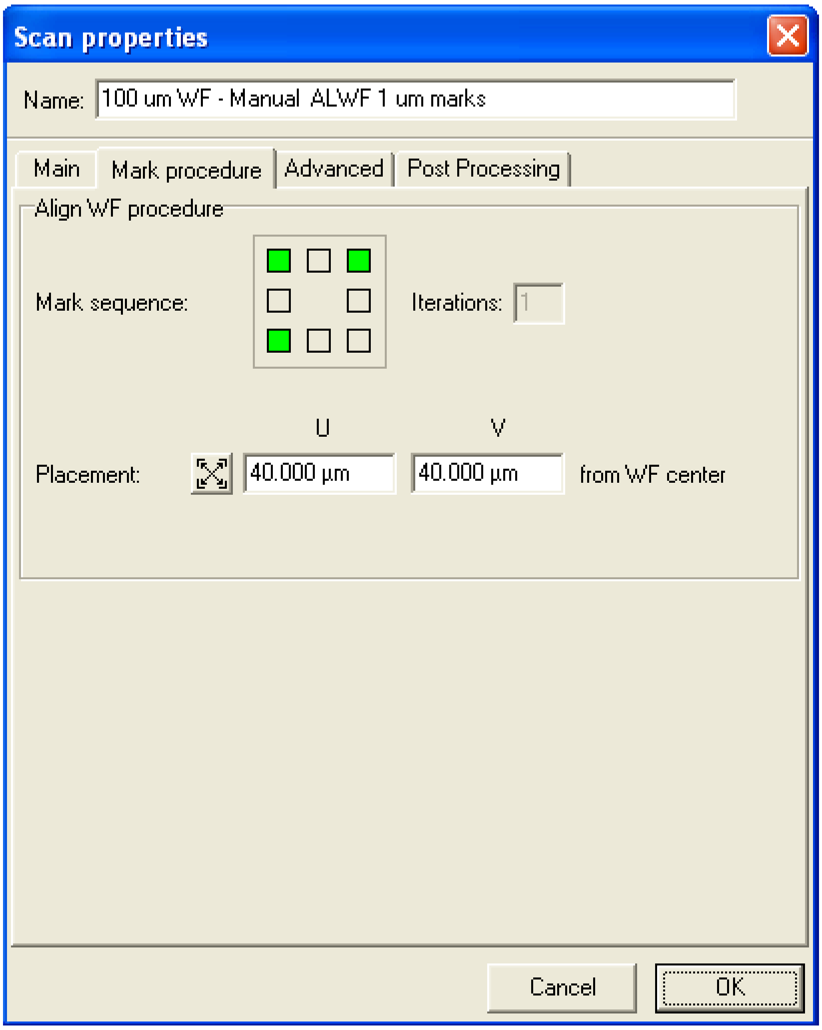


Figure 11-13: **Mark procedure** tab for Writefield Alignment

Control element	Function
WF Alignment procedure subset:	
Mark sequence	Up to eight marks can be defined by the user.
Placement	The U and V distance from the Writefield center can be defined.
	When the Placement button is initiated, the software checks the overall writefield size and permitted distances.

When the writefield has been chosen, up to eight marks can be selected. All marks can be evaluated. See the chapter **Beam to Sample Alignment** for more details, (⇒ *Alignment/Adjustment interrelation description* on page 13-5). The **Placement** of the marks refers to their position in UV, relative to the center of the writefield. When the **Placement** button is clicked, the software checks the overall dimensions of the writefield and calculates the maximum permitted distance from the center of the writefield, which will then be displayed. The user may manually input a placement value, which must be smaller than the maximum value displayed.

6.1.3 Scan properties: Advanced tab - Manual Writefield Alignment

An image filter may be chosen, for image processing, for example to increase sharpness.

Please refer to **Scan properties: Advanced tab - Images** for a detailed description, (⇒ *Scan Properties: Advanced tab - Images* on page 11-28).

6.1.4 Scan properties: Post Processing tab - Manual Writefield Alignment

Please refer to **Scan properties: Post Processing tab - Images** for a detailed description, (⇒ *Scan properties: Post Processing tab - Image Linescan* on page 11-33).

6.1.5 Calculated Writefield Correction

At the end of the manual writefield alignment procedure, if the number of corrected mark positions manually corrected by the user is smaller than the total available marks, the **Calculated Writefield Correction** dialog will be automatically displayed.

This dialog shows the user which alignment values have been calculated for **All marks** as well as those values calculated for **Only corrected marks.**

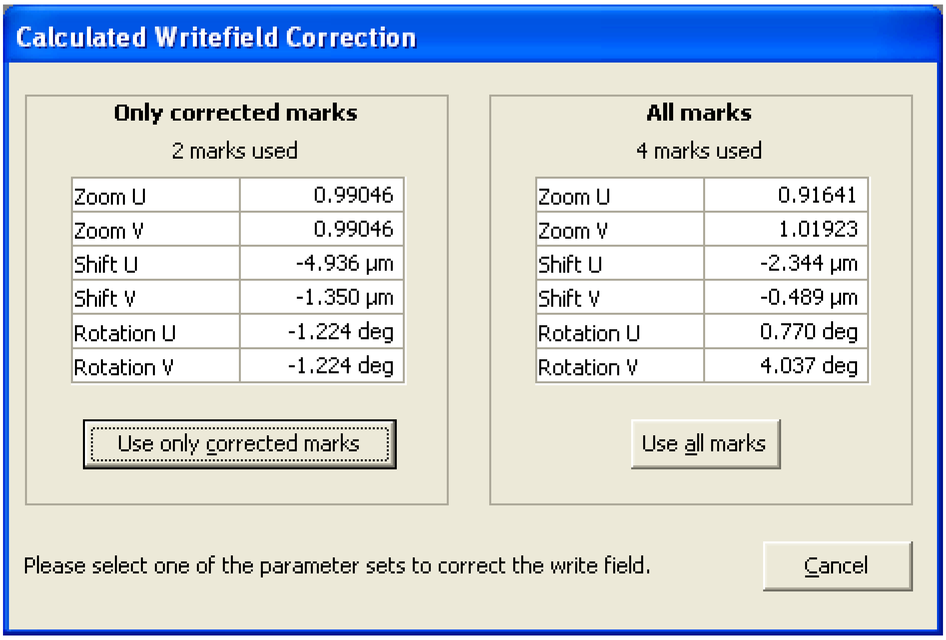


Figure 11-14: Calculated Writefield Correction

Control element	Function
Only corrected marks and **All marks** subset:	
Zoom U	The calculated value for the zoom correction factor in the U direction is calculated and displayed.
Zoom V	The calculated value for the zoom correction factor in the V direction is calculated and displayed.
Shift U	The distance required to correct for the shift in the U direction is calculated in microns and displayed.
Shift V	The distance required to correct for the shift in the V direction is calculated in microns and displayed.
Rotation U	The correction for the Rotation in the U direction is calculated in degrees and displayed.
Rotation V	The correction for the Rotation in the V direction is calculated in degrees and displayed.
Use only corrected marks	Only the corrected marks identified by the user will be used for the writefield correction.
Use all marks	All marks will be used for the writefield correction

There are two options for the user when the number of corrected marks is smaller than the total number of all marks. In the scenario described above, the number of marks that have been recognized is 2 out of a total of 4 marks defined for the procedure.

If the correct position of two marks is achieved, but the two other marks cannot be located or recognized, the user can choose to **Use corrected marks**. In doing so, the correction is performed using only the two marks that have been found and which are at their target positions.

If all marks can be located and they are all in their target positions, the option **Use all marks** can be selected.

6.2 Automatic Writefield Alignment with Images

This type of procedure is similar to a manual procedure, but it is based on the automatic mark detection algorithm. The mark scans achieved during the automatic procedure are compared to a reference scan, which is created manually or automatically before the procedure is started.

It is possible to setup an **Automatic with Images** procedure in two different ways. If no reference image exists, select the option **Create reference image first** at the start. In this case, a reference image is created automatically by recording an image in the center of the writefield.

The precision of an alignment depends on the accuracy of the position of the reference image.

6.2.1 Scan properties: Main tab - Automatic Writefield Alignment with Images

Please refer to **Scan properties: Main tab - Images** for a detailed description, (⇒ *Scan Properties: Main tab - Images* on page 11-27).

6.2.2 Scan properties: Mark Procedure tab - Automatic Writefield Alignment with Images

Please refer to **Scan properties: Mark Procedure tab - Manual Writefield Alignment** for a detailed description, (⇒ *Scan properties: Mark procedure tab - Manual Writefield Alignment* on page 11-34).

In addition, the **Number of iterations** may be selected by the user and the automatic writefield alignment will be carried out that number of times, each time achieving increasing accuracy.

6.2.3 Scan properties: Advanced tab - Automatic Writefield Alignment with Images

Please refer to **Scan properties: Advanced tab - Images** for a detailed description. In addition, the option **Create reference image first** is available.

Control element	Function
Advanced mark settings subset:	
Create reference image first	If this option is selected, a reference image is acquired at the start of the procedure.

6.2.4 Scan properties: Post Processing tab - Automatic Writefield Alignment with Images

Please refer to **Scan properties: Post Processing tab - Images** for a detailed description, (⇒ *Scan Properties: Post Processing tab - Images* on page 11-29).

Automatic detection of the deviation at each mark, i.e. an evaluation algorithm, is mandatory for all automatic procedures. See the corresponding section **Evaluation algorithms** to configure the mark detection.

6.3 Automatic Writefield with Linescan

During automatic writefield alignment, linescans will be used. Two linescans per mark are required to determine the UV position of the mark.

6.3.1 Scan properties: Main tab - Automatic Writefield Alignment with Linescans

Using linescans, the **Main direction** is fixed as the U axis and V as the minor axis. The software will automatically create two linescans in the U and V directions, to determine the deviation in U and V directions at each mark. Thus the selection of a Main direction is not necessary.

Please refer to **Scan properties: Main tab - Linescans** for a detailed description, (⇒ *Scan properties: Main tab - Linescan* on page 11-18).

6.3.2 Scan properties: Mark Procedure tab - Automatic Writefield Alignment with Linescans

Please refer to **Scan properties: Mark Procedure tab - Automatic Writefield Alignment with Images** for a detailed description, (⇒ *Automatic Writefield Alignment with Images* on page 11-37).

For automated writefield alignment with linescans, additional parameters are available.

Control element	Function
Offset	The distance between the center of a UV scan and the center of the Mark is defined in U and V directions.

6.3.3 Scan properties: Advanced tab - Automatic Writefield Alignment with Linescans

Please refer to **Scan properties: Advanced tab - Linescan** for a detailed description, (⇒ *Scan properties: Advanced tab - Linescan* on page 11-20).

6.3.4 Scan properties: Post Processing tab - Automatic Writefield Alignment with Linescans

Please refer to **Scan properties: Post Processing tab - Linescan** for a detailed description, (⇒ *Scan properties: Post Processing tab - Linescan* on page 11-22).

6.3.5 Scan properties: Evaluation tab - Automatic Writefield Alignment with Linescans

Please refer to **Scan properties: Evaluation tab - Linescan** for a detailed description, (⇒ *Scan properties: Evaluation tab - Linescan* on page 11-23).

The type of procedure and the parameters used for evaluation may be chosen by the user. The user must select a filter. **Peak detection** and **Cross correlation** are often used and are recommended. This is similar to a two-dimensional mark detection, but in this case, a linescan will be compared to a reference linescan.

6.4 GDSII Layer Based Mark Scans

These are mark scans which are embedded within the GDSII structures. There are two kinds of procedure available in **Scan Manager**, **Manual with Images** and **Automatic with Linescans**.

Post processing and evaluation settings are handled in the Scan Manager.

The position and size of the mark scan are set in the GDSII editor, (⇒ *Adding Manual Mark Scans – Imagescans* on page 5-65).

6.4.1 Manual GDSII Layer Based Mark Scans with Images

For this procedure, only the **Post Processing** tab is available for the Scan properties.

Please refer to **Scan properties: Post Processing tab - Images** for a detailed description, (⇒ *Scan Properties: Post Processing tab - Images* on page 11-29).

Please refer also to the GDSII chapter, for a description of how to create a manual mark scan, (⇒ *Adding Manual Mark Scans – Imagescans* on page 5-65).

6.4.2 Automatic GDSII Layer Based Mark Scans with Linescans

For this procedure only **Post Processing** and **Evaluation** tabs are available.

Please refer to **Scan properties: Post Processing tab and Evaluation tab - Linescans**, (⇒ *Scan properties: Post Processing tab - Linescan* on page 11-22) and (⇒ *Scan properties: Evaluation tab - Linescan* on page 11-23).

It is not possible to define the number of parameter sets for **GDSII Layer Based Mark Scans.** This property distinguishes this type of procedure from all other types of **Scan Manager** procedures. It means that all GDSII Mark Scans can only be evaluated with the same algorithm and settings.

7 GDSII Writefield Mark Scans

As in standard writefield alignment procedures, **Manual**, **Automatic with Linescans** and **Automatic with Images** procedures are available.

These GDSII Mark Scans can be placed via drag and drop into the GDSII structure. When the patterning process is initiated, the GDSII writefield mark scan will be started as well.

7.1 Manual GDSII Writefield Mark Scans

The **GDSII Writefield Mark Scan** can be carried out manually.

7.1.1 Scan properties: Main tab - Manual GDSII Writefield Mark Scans

Please refer to **Scan properties: Main tab - Images** for a detailed description, (⇒ *Scan Properties: Main tab - Images* on page 11-27).

7.1.2 Scan properties: Advanced tab - Manual GDSII Writefield Mark Scans

Please refer to **Scan properties: Advanced tab - Images** for a detailed description, (⇒ *Scan Properties: Advanced tab - Images* on page 11-28).

7.1.3 Scan properties: Post Processing tab - Manual GDSII Writefield Mark Scans

Please refer to **Scan properties: Post Processing tab - Images** for a detailed description, (⇒ *Scan Properties: Post Processing tab - Images* on page 11-29).

7.2 Automatic GDSII Writefield Mark Scans with Images

The Automatic GDSII Writefield Mark Scan with Images procedure is similar to the Images procedure.

7.2.1 Scan properties: Main tab - Automatic GDSII Writefield Mark Scans with Images

Please refer to **Scan properties: Main tab - Images** for a detailed description.

7.2.2 Scan properties: Advanced tab - Automatic GDSII Writefield Mark Scans with Images

Please refer to **Scan properties: Advanced tab - Images** for a detailed description.

Some additional parameters are available.

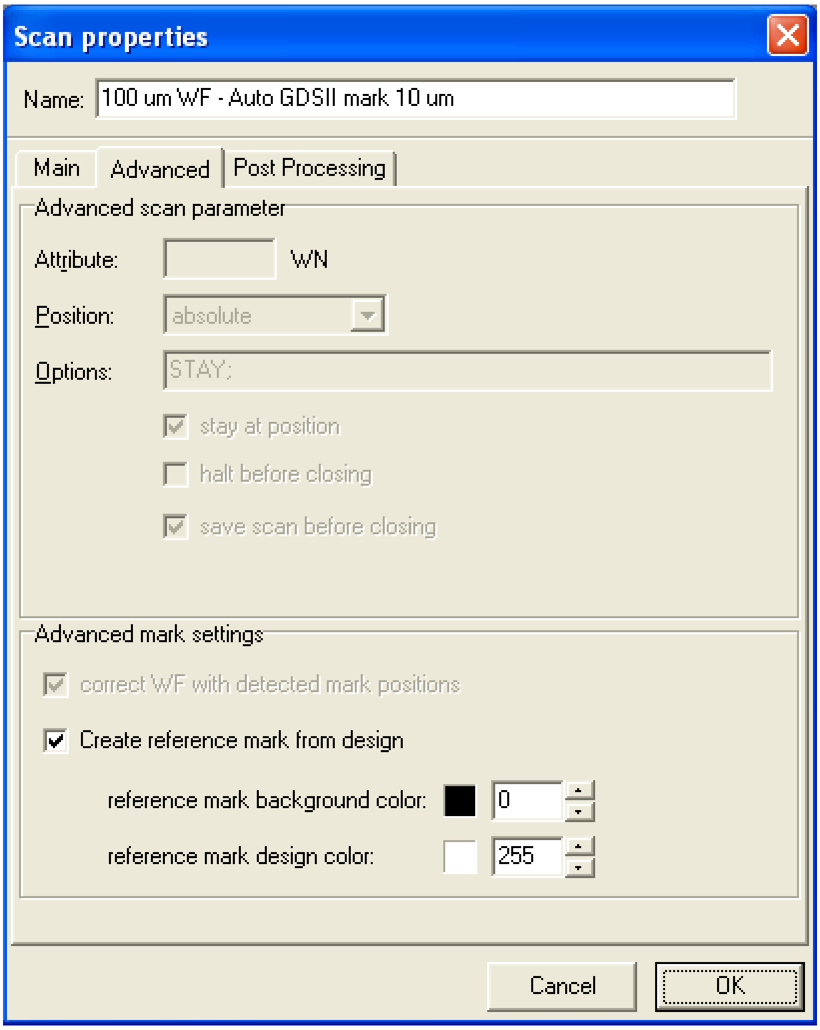


Figure 11-15: **Advanced tab for GDSII Writefield Mark Scans**

Control element	Function
Advanced mark settings subset:	
Create reference mark from design	Check the checkbox to enable the creation of the mark based on the shape of the mark in GDSII.
Reference mark background color	To select the reference mark background color
Reference mark design color	To select the color for the reference mark itself

In addition, the user may choose the reference mark background color and design color, for example black for the background and white for the cross structure. The user may also wish to choose more realistic color schemes, depending on the nature of the sample used.

The reference image can be created automatically from the design. This is the recommended way to create the GDSII writefield mark scan image. It is also possible to define a real image from the image data pool as an alternative.

7.2.3 Scan properties: Post Processing tab - Automatic GDSII Writefield Mark Scans with Images

Please refer to **Scan properties: Post Processing tab - Images** for a detailed description, (⇒ *Scan Properties: Post Processing tab - Images* on page 11-29).

It is absolutely essential to select the **Mark Detection filter**, this is different to the **Post Processing** tab for Images.

7.3 Automatic GDSII Writefield Mark Scans with Linescans

The Automatic GDSII Writefield Mark Scan with Linescans procedure is similar to the Writefield Alignment with Linescans procedure.

7.3.1 Scan properties: Main tab - Automatic GDSII Writefield Mark Scans with Linescans

Please refer to **Scan properties: Main tab - Linescans** for a detailed description, (⇒ *Scan properties: Main tab - Linescan* on page 11-18).

7.3.2 Scan properties: Advanced tab - Automatic GDSII Writefield Mark Scans with Linescans

Please refer to **Scan properties: Advanced tab - Linescans** for a detailed description, (⇒ *Scan properties: Advanced tab - Linescan* on page 11-20).

7.3.3 Scan properties: Post Processing tab - Automatic GDSII Writefield Mark Scans with Linescans

Please refer to **Scan properties: Post Processing tab - Linescans** for a detailed description, (⇒ *Scan properties: Post Processing tab - Linescan* on page 11-22).

7.3.4 Scan properties: Evaluation tab - Automatic GDSII Writefield Mark Scans with Linescans

Please refer to **Scan properties: Evaluation tab - Linescans** for a detailed description, (⇒ *Scan properties: Evaluation tab - Linescan* on page 11-23).

8 Beam Tracking Alignment procedure

Beam tracking alignment corrects for small stage inaccuracies. The beam is deflected by the pattern generator to compensate stage displacement. The beam tracking alignment is similar to writefield alignment, but it is carried out in XY instead of UV coordinates.

8.1 Manual Beam Tracking Alignment

The manual beam tracking alignment procedure is similar to the manual writefield alignment procedure.

Please refer to **Scan properties: Manual Writefield Alignment** for a detailed description, (⇒ *Manual Writefield Alignment* on page 11-34).

8.2 Automatic Beam Tracking Alignment with Images

The automatic beam tracking alignment procedure with images is similar to the automatic writefield alignment procedure with images.

Please refer to **Scan properties - Automatic Writefield Alignment with Images** for a detailed description, (⇒ *Automatic Writefield Alignment with Images* on page 11-37).

8.3 Automatic Beam Tracking Alignment with Linescans

The automatic beam tracking alignment procedure with linescans is similar to the automatic writefield alignment procedure with linescans.

Please refer to **Scan properties - Automatic Writefield Alignment with Linescans, (**⇒ *Automatic Writefield with Linescan* on page 11-38**).**

9 Adjust UV procedures

The main task of a predefined adjustment procedure is to automate the adjustment on wafers (**global** adjustment) and their chips (**local** adjustment). If the user works with the same wafer layout every day, the adjustment procedures will simplify and speed up this task.

9.1 Manual Adjust UV procedures

Some of the manual procedures are similar to the linescans and images procedures.

9.1.1 Scan properties: Main tab - Manual Adjust UV procedure

Please refer to **Scan properties: Main tab - Images** for a detailed description, (⇒ *Scan Properties: Main tab - Images* on page 11-27).

9.1.2 Scan properties: Mark Procedure tab - Manual Adjust UV procedure

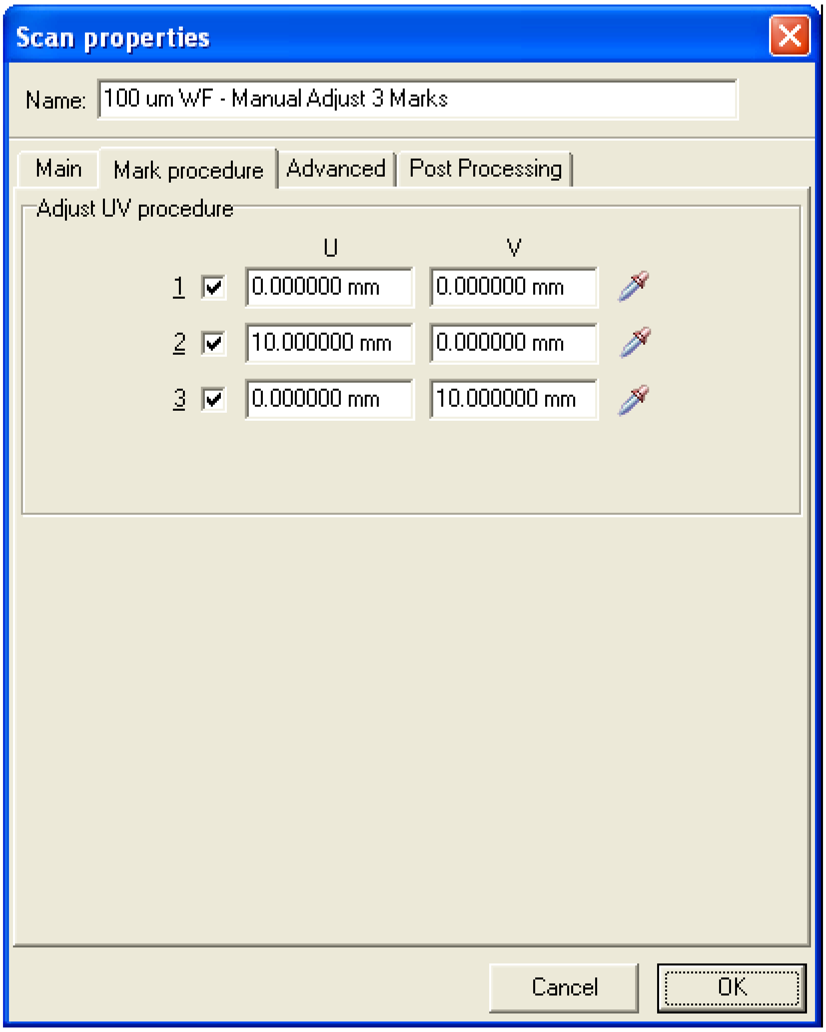


Figure 11-16: **Mark procedure** tab - Adjust UV procedure

Control element	Function
Adjust UV procedure	Up to three marks can be defined for the **Adjust UV procedure**. The UV coordinates can either be entered by the user or read by the software via the **Pipette** (Read) button.

Choose the **Mark procedure** tab to define the number and positions of the mark scans. Use the check boxes to select which marks will be used and type in the **U** and **V** coordinates in the corresponding input fields. The number of marks corresponds to a 1, 2 or 3-point adjustment (⇒ *Transformation with 1, 2, or 3 points* on page 12-7). After selecting the number of marks, type in the U and V coordinates in the corresponding input fields.

If only the **Shift** value needs to be corrected, one mark will be enough. If **Shift** and **Rotation** values should be corrected, two marks are required. If **Shift**, **Zoom** and **Rotation** should be corrected, three marks are required.

9.1.3 Scan properties: Advanced tab - Manual Adjust UV procedure

Please refer to **Scan properties: Advanced tab - Images** for a detailed description (⇒ *Scan Properties: Advanced tab - Images* on page 11-28).

In addition, there are further parameters available.

Control element	Function
Adjust after every mark	Select this option to calculate and apply **Shift** correction immediately after the first mark, followed by **Rotation** correction immediately after the second mark. After the third mark, the **Zoom** will be corrected. This increases the accuracy from step to step and simplifies finding the next mark scan in the sequence. If this option is not selected, all three marks will be obtained first and the three corrections will be applied afterwards.

9.1.4 Scan properties: Post Processing tab - Manual Adjust UV procedure

Please refer to **Scan properties: Post Processing tab - Images** for a detailed description, (⇒ *Scan Properties: Post Processing tab - Images* on page 11-29).

9.2 Automatic Adjust UV procedures with Images

The following sections describe the different tabs for the Automatic Adjust UV procedure with Images.

9.2.1 Scan properties: Main tab - Automatic Adjust UV procedure with Images

Please refer to **Scan properties: Main tab - Automatic Writefield Alignment with Images** for a detailed description, (⇒ *Scan properties: Main tab - Automatic Writefield Alignment with Images* on page 11-37).

9.2.2 Scan properties: Mark Procedure tab - Automatic Adjust UV procedure with Images

Please refer to **Scan properties: Mark Procedures tab - Manual Adjust UV Procedure** for a detailed description, (⇒ *Scan properties: Mark Procedure tab - Manual Adjust UV procedure* on page 11-46).

9.2.3 Scan properties: Advanced tab - Automatic Adjust UV procedure with Images

Please refer to **Scan properties: Advanced tab - Manual Adjust UV Procedure** for a detailed description, (⇒ *Scan properties: Advanced tab - Manual Adjust UV procedure* on page 11-47).

9.2.4 Scan properties: Post Processing tab - Automatic Adjust UV procedure with Images

Automatic detection of the deviation at each mark, i.e. an evaluation algorithm, is mandatory for all automatic procedures. See the corresponding section **Evaluation algorithms** to configure mark detection.

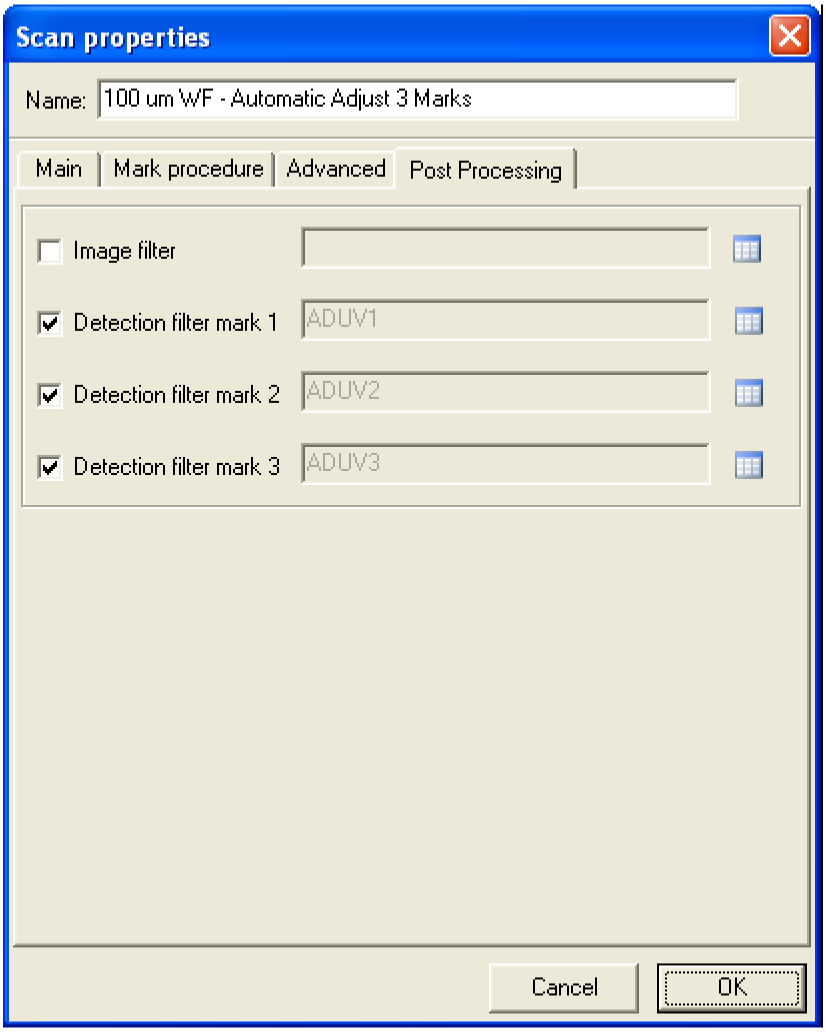


Figure 11-17: **Post Processing** tab for **Adjust UV - Automatic with Images**

Control element	Function
Image filter	Select a parameter set from the database. Check the image filter option to activate it.
Detection filter Mark 1, 2, 3	Select the mark detection parameter set for the desired mark.

9.3 Automatic Adjust UV Procedures with Linescans

In the case of an **Automatic with Linescans** procedure, two Linescans are defined to determine the deviation in U and V directions at each mark. Thus the selection of a **Main Direction** is not necessary

This is a similar procedure to writefield alignment with linescans, but the maximum number of marks here is three, compared with eight marks when performing writefield alignment with linescans. The offset from the centre of the cross will also be displayed.

9.3.1 Scan properties: Main tab - Automatic Adjust UV Procedure with Linescans

Please refer to **Scan properties: Main tab - Automatic Writefield Alignment with Linescans** for a detailed description, (⇒ *Scan properties: Main tab - Automatic Writefield Alignment with Linescans* on page 11-38).

9.3.2 Scan properties: Advanced tab - Automatic Adjust UV Procedure with Linescans

Please refer to **Scan properties: Advanced tab - Manual Adjust UV Procedure** for a detailed description, (⇒ *Scan properties: Advanced tab - Manual Adjust UV procedure* on page 11-47).

There is an additional parameter available.

Control element	Function
Offset	The offsets for U and V can be defined (⇒ *Automatic Writefield Alignment with Linescans* on page 11-7).

9.3.3 Scan properties: Post Processing tab - Automatic Adjust UV Procedure with Linescans

Please refer to **Scan properties: Post Processing tab - Linescans** for a detailed description, (⇒ *Scan properties: Post Processing tab - Linescan* on page 11-22).

9.3.4 Scan properties: Evaluation tab - Automatic Adjust UV Procedure with Linescans

Please refer to **Scan properties: Evaluation tab - Linescans** for a detailed description.

9.4 Adjust UV Procedure Options via tool bar

When the **Adjust UV** procedure is activated, an additional button is available in the tool bar.

(⇒ *Options* on page 12-19)

10 Scan Manager menu

In the **Scan Manager** title bar there is a button available, which brings up four menu items:

- **Linescan Options**

- **Imagescan Options**

- **Videoscan Options**

- **Markscan Options**

10.1 Linescan Options

The parameters for the **Linescan** can be set in this dialog.

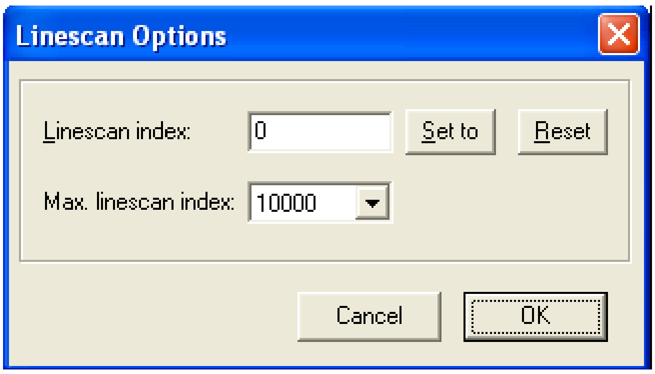


Figure 11-18: **Linescan Options** dialog

Control element	Function
Linescan index	The user can enter a value for the current Linescan index. Clicking the **Set to** button will confirm the setting. All newly scanned linescans will be counted beginning with this index.
Reset	Clicking on the **Reset** button will reset the counter to zero.
Max. linescan index	The maximum linescan index can be selected from the dropdown list.

Using this dialog, the user can define the maximum number of linescans that may be automatically generated and saved. Each new linescan name is automatically generated using the format <EBEA + index + extension .lsc>.

When each linescan is acquired, the index is incremented by 1 and the scan is saved under the name that includes that incremented index. When the maximum index number is reached, the index is re-started at the beginning and the old linescans will be overwritten. In the example described, the maximum number of stored linescans is 10,000.

10.2 Imagescan Options

The parameters for the **Imagescan** can be set in this dialog.

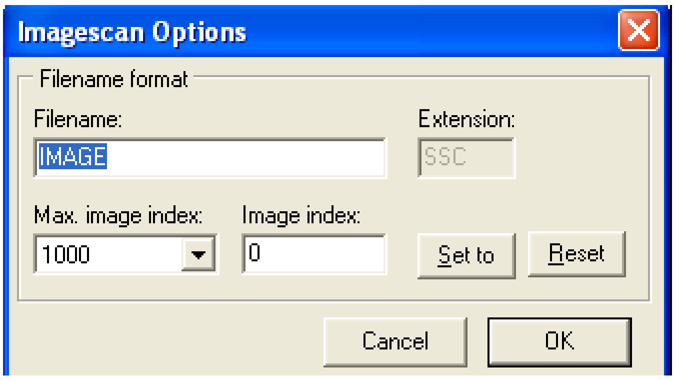


Figure 11-19: **Imagescan Options** dialog

Control element	Function
Filename	The user can enter a string which will be a part of all automatically generated images.
Extension	The software will create the appropriate extension automatically.
Max. image index	The maximum image index can be selected from the dropdown list.

Control element	Function
Image index	The current image index can be entered by the user. Clicking the **Set to** button will confirm the setting. All new scanned images will be counted, beginning with this index.
Reset	Clicking on the **Reset** button will reset the value to zero.

In this dialog, the user may choose the maximum number of individual images that can be automatically generated and saved. When each image is acquired, the software will generate the filename automatically. Whatever the user enters in the **Filename** field will become the prefix to the name of each individual image. Each image will be named, including the index number, using the general format <prefix + image index + file extension .ssc>. The file extension is automatically selected by the software. In the example described, the maximum number of stored images will be 1,000 When this number of stored images is reached, the index is restarted from the beginning and old images are overwritten.

10.3 Videoscan Options

The parameters for the **Videoscan** can be set in this dialog.

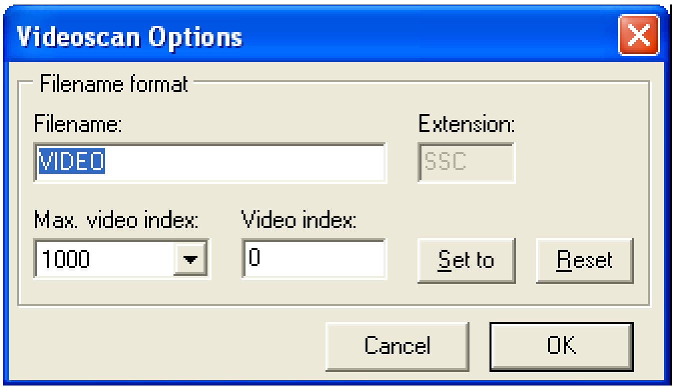


Figure 11-20: **Videoscan Options** dialog

Control element	Function
Filename	The user can enter a filename.
Extension	The software will create the appropriate extension automatically.
Max. video index	The maximum video scan index can be selected from the dropdown list. Clicking the **Set to** button will confirm the setting.

Control element	Function
Video index	The video scan index can be entered by the user. Clicking the **Set to** button will confirm the setting.
Reset	Clicking on the **Reset** button will reset the value to the last set video scan index value.

In this dialog, the user may choose the maximum number of individual video images that can be automatically generated and saved. When each image is acquired, the software will generate the filename automatically. The user entry in the **Filename** field will become the prefix to the name of each individual saved video scan. Each video scan will be named, including the index number, using the general format <prefix + video scan index + file extension .ssc>. The file extension is automatically selected by the software. In the example described, the maximum number of stored video scans will be 1,000. When this number of stored video scans is reached, the index is restarted from the beginning and old video scans are overwritten.

10.4 Markscan Options

The parameters for the **Markscan** can be set in this dialog.

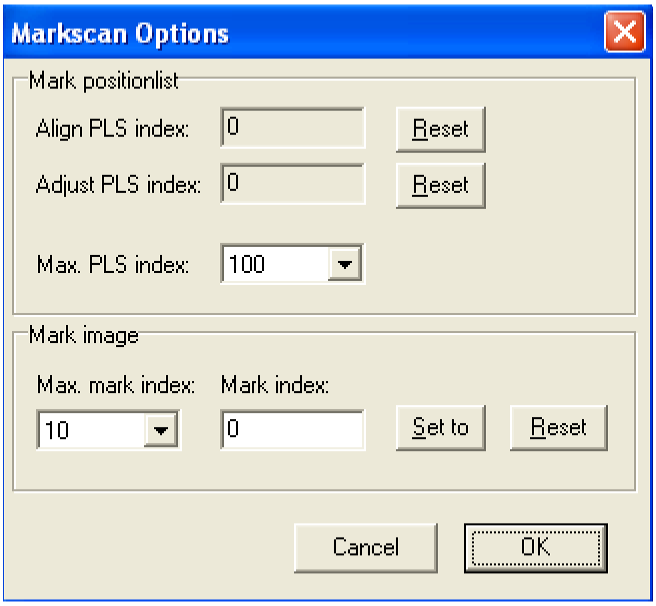


Figure 11-21: Markscan Options

Control element	Function
Mark Positionlist subset:	
Align PLS index	Counter of Positionlists containing writefield alignment procedures. These Positionlists are stored in the folder **<User Root>➡Data➡ScanPLS**. Each Positionlist will be named, including the index number, using the general format: **<prefix ALIGN + Positionlist index + file extension. pls>** Clicking on the **Reset** button will reset the counter to zero.
Adjust PLS index	Counter of Positionlists containing automated UV Adjustment procedures. These Positionlists are stored in the folder **<User Root>➡Data➡ScanPLS**. Each Positionlist will be named, including the index number, using the general format: **<prefix ADJUST + Positionlist index + file extension. pls>**
Max. PLS index	The user can select the maximum Positionlist index from the dropdown list.
Mark image subset:	
Max. mark index	The user can select the maximum mark index from the dropdown list. Clicking the **Set to** button will confirm the setting.
Reset	Clicking on the **Reset** button will reset the value to zero.

Mark scans are automatically created and saved during every alignment or adjustment procedure. **Max. mark index** determines how many individual mark scans can be saved. For each alignment or adjustment procedure, a new Positionlist is automatically created.

Using the **Max PLS index**, the maximum number of alignment or adjustment Positionlists is defined. The index for the alignment and adjustment Positionlist may either be manually set or continuous, in which case the index is incremented by 1 with each successive Positionlist

11 Drag and drop function

Any **Scan Manager** dialog procedure can be dragged and dropped into the Positionlist or any UV window. If a procedure is dropped into the Positionlist, its target coordinate will be the current stage position. If it is dropped into a UV window, the target coordinates of the scan will be the position inside the UV window. The procedure will then be executed. The execution of the procedure may also be initiated within the **Scan Manager** by pressing **F9**.

Please read the **Positionlist** chapter to familiarise yourself with the functioning of the Positionlist, (⇒ *Positionlist* on page 9-1).

Chapter 12
Stage to Sample Adjustment

Some background information on the stage to sample adjustment will be given here, before describing the **Adjust UVW** dialog in more detail.

The following topics and tasks are included:

- Background information on coordinate systems and transformations, explaining the relationship between XY and UV coordinates, with diagrams, (⇒ *Coordinate systems and transformations* on page 12-3),
- Explanation of the difference between **Local** and **Global** coordinates and the transformations between them,
(⇒ *Global and Local transformations* on page 12-6),
- Background information on transformations using 1 point, 2 point or 3 points adjustments,
(⇒ *Transformation with 1, 2, or 3 points* on page 12-7),
- Step by step guide to performing **Origin Correction**,
(⇒ *Defining an Origin Correction* on page 12-9), as well as **Angle Correction**, (⇒ *Defining an Angle Correction* on page 12-9),
- Explanation of **3-Points** adjustment on unpatterned and patterned samples, (⇒ *3-Points Adjustment* on page 12-10),
- Step by step guide to performing multi-field patterning, using working distance correction via **Focus** or via **Stage,**
(⇒ *Performing Automatic Working Distance Correction* on page 12-11),
- Detailed description of the **Adjust UVW** dialog, consisting of the four tabs Origin correction, Angle correction, 3-Points Adjustment and W Adjust, (⇒ *Adjust UVW dialog* on page 12-13).

You can call up **Adjust UVW** by clicking the (Adjustment) icon in the control bar.

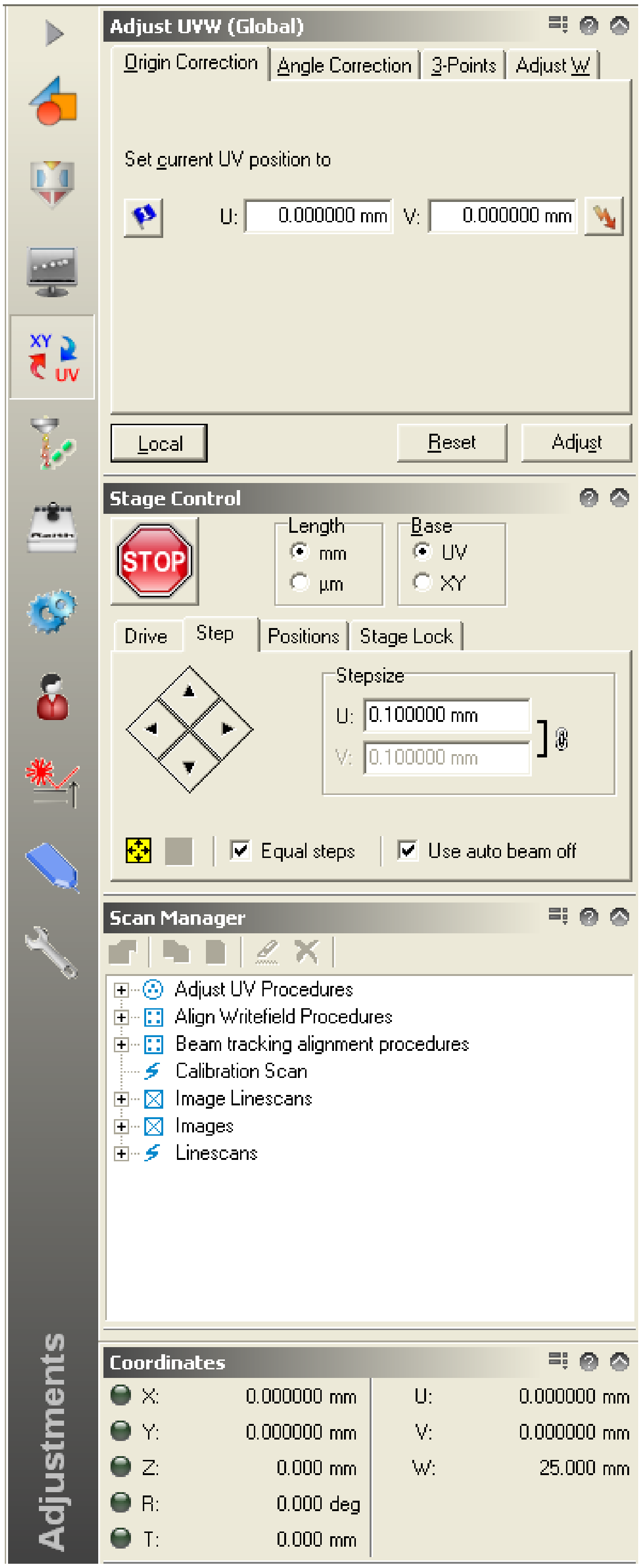


Figure 12-1: **Adjustments**

A Functional Description

1 Background information

It is typical, for nearly all application tasks, to find a specific location on a sample and then, for example, to write at this location a GDSII structure. To define the location, it is more convenient to apply a coordinate system to the sample. Finding this location is then simply a drive command within this coordinate system. The success of the principle depends on the accuracy of the applied transformation, as well as the accuracy and precision of the stage, compared to the required accuracy of positioning.

The various possible coordinate systems of the sample stage, the sample and the beam and their relationship to each other need to be understood in order for the overall process to be successfully implemented by the user. To give an overview, the first part of this chapter introduces the coordinate systems and transformations between them.

Throughout the software and manuals, the terms **adjust** or **adjustment** are used only with regard to the transformation between the stage and the sample. The term **alignment** is used with regard to the transformation between the sample and the beam.

1.1 Coordinate systems and transformations

To define and then find a location, it is very convenient to apply a coordinate system to the sample. For example, the center of a wafer is conventionally (0, 0) and an axis of this coordinate system is perpendicular to the direction of the flat. Within the Raith NANOSUITE software, this sample coordinate system is labeled using **U**, **V**, and **W.**

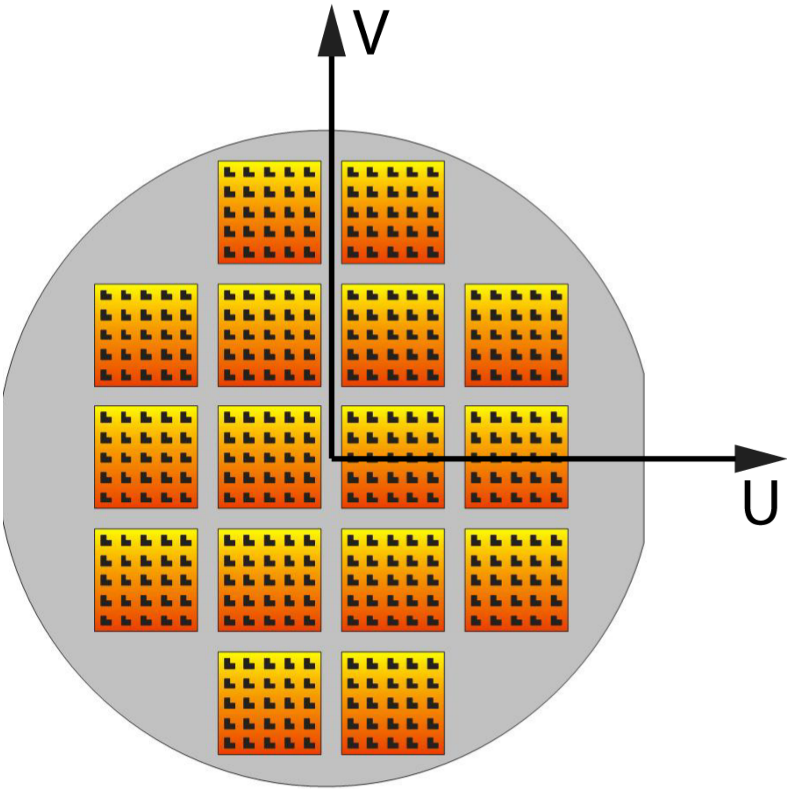


Figure 12-2: Sample **UV** coordinates defined on a wafer

On the other hand, the stage of the system is moving in another coordinate system. The directions of the stage movements define this coordinate system. The directions of this coordinate system are identified throughout this manual and the software using **X**, **Y**, and **Z**. These coordinates are perpendicular to each other. Normally (0, 0) of this system is the center of the travel range and the X axis is then arbitrarily selected as one of the motor axes.

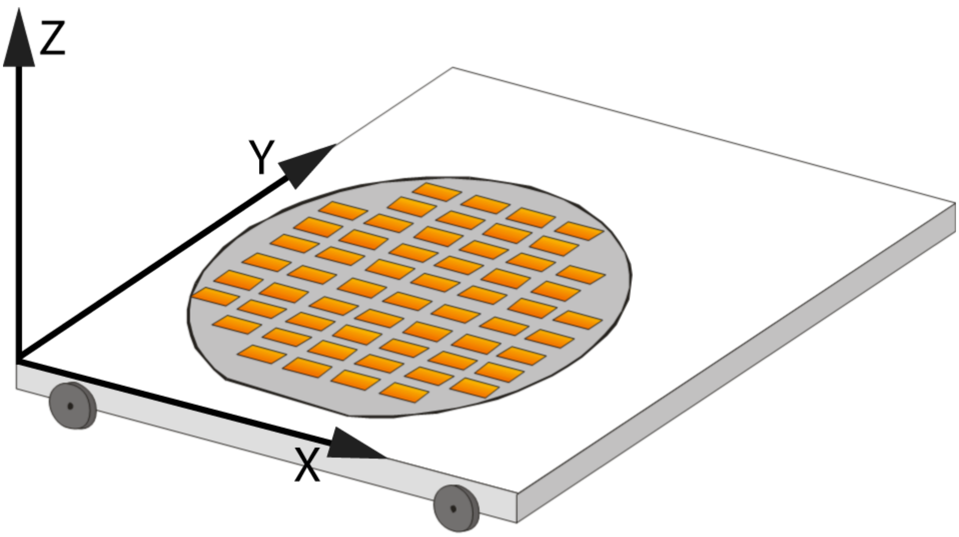


Figure 12-3: Stage coordinates **X**, **Y** and **Z**

These two coordinate systems are almost always not identical. The two coordinate systems will usually have different origins, related by an offset vector, or **Shift**. Additionally, the UVW system may be rotated, compared to the XYZ system and the systems may have different scales.

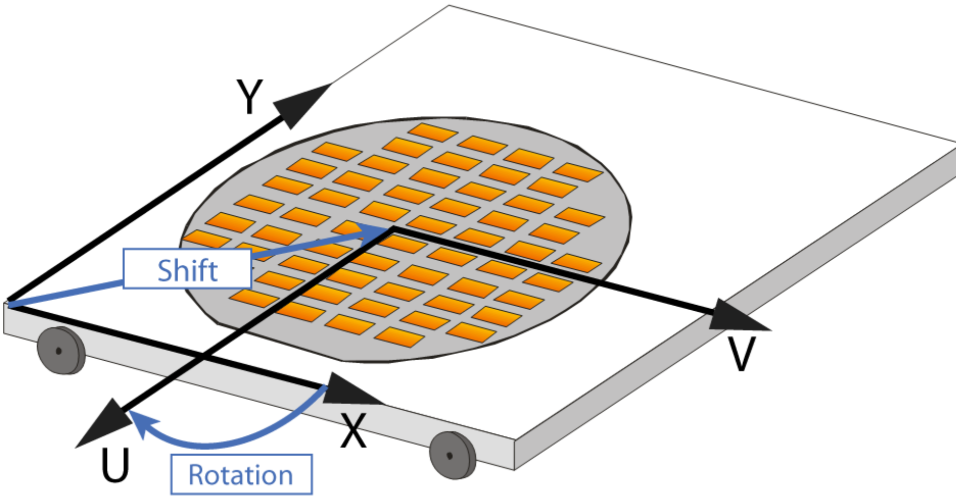


Figure 12-4: Stage and sample coordinates transformation

The user will usually consider the UV axes, which relate to the pattern on the sample, whilst the software needs to drive the stage using XY positioning. Consequently, a coordinate transformation has to be calculated, so that the software drives in XY to locations on a sample, which the user sees and defines in UV terms. How this transformation fits into the scheme of the coordinate systems is shown in the figure below.

The angle of rotation between XY and UV axes can be any angle.

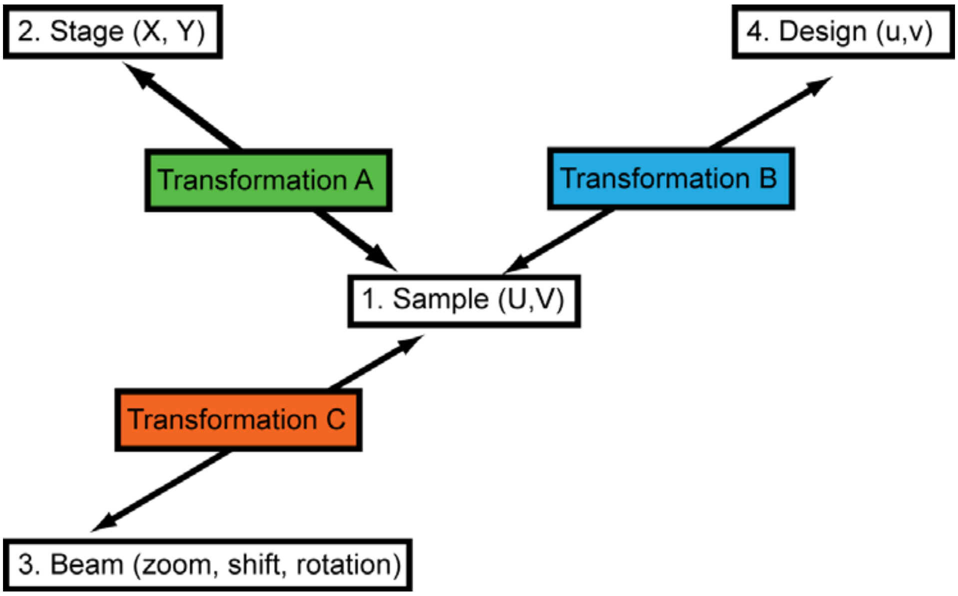


Figure 12-5: Stage to sample coordinates transformation A.

Transformations in some applications are simpler than in others. This chapter will describe applications using simpler transformation, through to more complex situations, so that the user acquires a full understanding of which transformations are needed in which applications. The **Adjust UVW** dialog contains all of the commands required to set up and establish the various transformations between the XYZ system and the UVW system.

1.2 Global and Local transformations

Within the Raith NANOSUITE software, two different transformations exist, called **Global** transformation and **Local** transformation. To introduce two different transformations, it is very appropriate to look at examples of wafer applications. When working on wafers, the use of two independent coordinate transformations is very convenient. The first coordinate transformation can be allocated to the wafer itself, whereas the second transformation serves for navigation within a chip.

Control element	Function
Local/Global 	Choose this command in **Adjust UVW** to switch between **Global** transformation and **Local** transformation at any time. In addition, it is possible to switch between them by choosing the related buttons in the **Tool bar**. The buttons, as well as the window title, display the current status. For both **Global** and **Local** transformations, a complete set of transformation parameters is stored. In addition, each set also includes the locations of the marks which were used to calculate the transformations. Switching between Global and Local activates one of these sets.

The Global and Local transformations are not independent of each other. The following relationships exist between them.

- Any action performed while the **Global** transformation is active affects also the **Local** transformation with the same action.
- Any action performed while the **Local** transformation is active has no impact on the **Global** transformation.

Switching to the **Global** transformation, followed by clicking on **Adjust** also recalculates the **Local** transformation using same transformation parameters. Switching afterwards between Local and Global makes no difference, since each transformation is the same.

Changing Local coordinates and clicking **Adjust**, calculates a new **Local** transformation which is completely independent of the Global setting. After switching back to the **Global** transformation it is possible to use the Global setting again.

1.3 Transformation with 1, 2, or 3 points

Again, the following information applies to the transformation between XY and UV systems only.

In general, a coordinate transformation can be established using 3 points, 2 points, or 1 point only. The number of points required depends on the application. To select the number of points, use the checkboxes next to each blue flag within **Adjust UV**.

- Using 1 point only. By defining **X, Y**, **U** and **V** at one location on the sample it is possible to calculate the **Shift** between the two systems. Other transformation variables are not affected. An alternative is to use the **Origin Correction** tab. In this tab, specify **U** and **V** only and when clicking **Adjust,** the current **X** and **Y** coordinates are used.
- Using 2 points. By defining **X**, **Y**, **U** and **V** at two locations on the U axis of the pattern on the sample, it is possible to calculate the **Shift**,

one rotation angle and one scaling factor between the two systems. The rotation angle and the scaling factor are applied to both axes equally, i.e. both axes are rotated and scaled by the same amount. An alternative is to use the **Angle Correction** tab. The functionality is the same. At each point, X and Y are defined assuming that both locations lie on the U axis, so one rotation angle can be calculated, which is applied to both axes. The other transformation parameters are not affected.

- Using 3 points. By defining **X**, **Y**, **U** and **V** at three locations on the sample, it is possible to calculate the **Shift**, two rotation angles and two scaling factors between the two systems. The rotation angles and the scaling factors are applied to each axis individually, i.e. the axes are rotated and scaled by different amounts.

The following table summarizes the possible coordinate transformations and their outcomes.

Control element	Function
1- Point	New **Shift**. Scaling factors for both axes are unchanged. Rotation angle for both axes are unchanged.
Origin Correction	New **Shift**. Scaling factors for both axes are unchanged. Rotation angle for both axes are unchanged.
2 - Points	New **Shift**. One new scaling factor for both axes. One new rotation angle for both axes.
Angle Correction	**Shift** unchanged. Scaling factors are unchanged. One new rotation angle for both axes.
3 - Points	New **Shift**. Two new scaling factors, one for each axis. Two new rotation angles, one for each axis.

B Tasks

1 Defining an Origin Correction

- Locate a reference structure on the sample, such as the lower left corner.
- Use the appropriate control, e.g. a joystick, to place this reference structure in the center of the field of view.
- Switch to **Global** or **Local** transformation before clicking **Adjust** to calculate the shift for that transformation.
- Press **Adjust** to apply the **Origin Correction**.

2 Defining an Angle Correction

- Locate an extended reference structure, which can serve as the U axis, e.g. the lower edge of the sample.
- Use the appropriate control, e.g. a joystick, to place one point of this reference structure in the center of the field of view.
- Click the **Pipette (Read)** button in the **P1** panel to use the corresponding **XY** coordinates for the angle calculation.
- Use the control to place another point of the reference structure in the center of the field of view. The separation between this point and the first one should be a few mm, e.g. 10 mm.
- Click the **Pipette (Read)** button in the **P2** panel to use the XY coordinates for the angle calculation. After clicking the **Pipette (Read)** button, the **Calculated angle** is displayed on the same tab, in red. Please verify that the calculated angle is of the order that you would expect, especially with regard to the associated positive or negative sign, ensuring that the calculated angle is in the right direction.

  Choose **Adjust** to use the newly defined U direction. The angle between U and X is now displayed in green.

3 3-Points Adjustment

The following sections explain the 3-Points Adjustment procedure.

3.1 An unpatterned, small sample

In this case, no pattern exists on the sample to serve as a reference and the sample itself is not a complete wafer. It may be only a fraction of it with one or more straight edges.

This kind of adjustment requires the completion of two tasks. Switch to **Global** or **Local** transformation. Normally, for simple tasks the Global mode is the better choice. Then click on **Reset** to make sure that the scaling factors, as well as all other parameters are set to the identity transformation. One task is to assign specific coordinates to a particular point on the sample. The other task is to define the U-direction on the sample.

These two tasks may be performed in either order. It is also possible to perform them simultaneously, when one point is used to define the position as well as the angle.

3.2 A patterned sample

If a pattern already exists on a sample, for example from some other previous process step, and one has to navigate on this sample to this structure, it is absolutely necessary to adjust the UV coordinates to this pattern. This ensures that any errors outside the control of the user are eliminated. It may be, for example, that the pattern was defined in a previous step by a mask plate with a scaling error, or perhaps the pattern is incorrectly rotated against a reference direction, e.g. a wafer flat, due to poor alignment.

3.3 Performing 3-Points Adjustment

- Type the **U** and **V** coordinates of the three reference structures into the corresponding **U** and **V** fields. The easiest way is to get them from the GDSII design. Here, it is possible to use the green flags.
- Locate one of the three reference structures, e.g. the first one. Select the **Pipette (Read)** button to automatically update the corresponding **XY** coordinates. Alternatively, use the blue flags to read the XY coordinates from a UV window.

This task is sometimes the most demanding. It may be very helpful to firstly carry out a coarse adjustment, using only an origin and an angle correction, on the **Global** coordinate system. This coordinate system can then be used to find the first reference structure. The following fine adjustment, using three points can then be done in the **Local** coordinate system. Thereby, it is possible to switch at each point back to **Global** to find the next structure or to repeat previous steps.

- ▶ Check the checkboxes next to affected alignment points P1, P2 or P3. The **X**, **Y**, **U** and **V** values will be used to calculate the corresponding transformation.

  Locate the second structure of the three reference structures. Select the **Pipette (Read)** button and check the corresponding checkbox.

- ▶ Locate the last structure of the three reference structures. Select the **Pipette (Read)** button and check the corresponding checkbox.

- ▶ To calculate and to activate the transformation, click **Adjust**.

If the coordinate transformation differs only slightly from the identity or if a **Global** transformation is used (see previous hint), the task to find the second and third structures can be simplified, by pressing **Adjust** after every single corrected adjustment point.

4 Performing Automatic Working Distance Correction

When performing multi-field patterning, it is recommended to correct for any change in the working distance (WD) due to a slightly tilted sample. The automatic **Working Distance Correction** via **Adjust UV** assumes that the surface of the sample is completely level.

By measuring the WD values on three adjustment points, the true orientation of the tilted sample, with respect to the horizontal plane, can be determined and corrected for.

There are two different methods which can be applied:

- **Adjustment via Focus** will compensated for changes in the working distance by changing the focus setting.
- **Adjustment via Stage** will compensate for changes in the working distance by moving the stage in the Z direction.

4.1 Adjustment via Focus

- ▶ Define marks 1, 2 and 3 by reading the U and V coordinates from the GDSII design.

- Drive the stage to the first mark.
- Adjust X and Y of the mark via the joystick.
- Adjust the focus. The focus will be re-adjusted at the marks.
- Click the **Pipette (Read)** button to read in the current X, Y and WD.
- Repeat the same procedure for marks 2 and 3.
- Click on the **Adjust** button to perform the adjustment.

4.2 Adjustment via Stage

- Define marks 1, 2 and 3 by reading the U and V coordinates from the GDSII design.
- Drive the stage to the first mark.
- Adjust X and Y of the mark via the joystick.
- Adjust the focus by changing the Z position via the joystick. The focus remains unchanged. The Z value will vary for a short period, until the sample is in focus.
- Click the **Pipette (Read)** button to read in the current X, Y and WD.
- Repeat the same procedure for marks 2 and 3.
- Click on the **Adjust** button to perform the adjustment.

  Now the UV positions, when driving in the U or V directions, will be adjusted either using the stage or focus, depending on which option was chosen. This is valid for all UV positions, regardless of initiation from the Positionlist or via the **Stage Control** window or via a script.

C Software Reference

You can call up **Adjust UVW** by clicking the (Adjustment) icon in the control bar.

The **Adjust UVW** dialog consists of the following four tabs:

- **Origin correction**: This defines the origin of the coordinates system.
- **Angle correction**: This defines the angle between the XY and UV coordinate systems, which is an important correction to be performed before patterning can be initiated.
- **3-Points Adjustment**: Using 3 points, the adjustment between the stage coordinates (XYZ) and the sample coordinates (UVW) can be carried out.
- **W Adjust**: This will carry out the adjustment for the working distance.

1 Adjust UVW dialog

This dialog consists of four tabs. There are three generic buttons, applicable to all tabs, **Local**/**Global**, **Reset** and **Adjust**. as well as three buttons, the **Pipette (Read)**, **Flash** and **Flag** buttons.

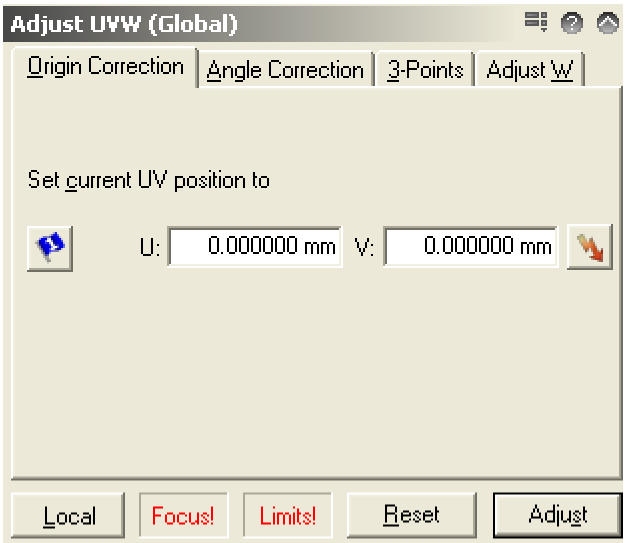


Figure 12-6: Adjust UVW

Control element	Function
	To drive the stage to the selected mark position, once it is defined.
Adjust	To carry out Adjustment
Reset	The Adjustment Reset function for UV and W are separated in order to keep the sample height information for tilting when resetting the UV- adjustment. The Reset button will change between **Reset UV** to **Reset W** depending on the selected tab. Pressing Reset will return the current adjustment to the previously saved settings. In the **Local** coordinates system, local coordinates will be set equal to Global coordinates. In the **Global** coordinates system, the UV coordinates system will be set equal to XY coordinates system. During the **Reset W** process, the new W value and sample height will be calculated based on the **Default W** or **Default W for RT module** settings defined in **Module Status > Adjust UV**. Depending on the stage configuration, Reset procedure will use the first or second
Focus!	To be displayed when **Automatic Working Distance** correction has been activated.
Limits!	To be displayed when **Adjustment limits** check has been activated.

A position within a Positionlist consists of specific coordinate values. It is now possible to drag and drop the coordinate set from the Positionlist onto a mark. The coordinate values will then be used for the mark.

Normally, a user will wish to enter the UV position of a sample and afterwards the XY coordinates can be read. The blue flag from **Adjust UVW** can be moved via a drag and drop into the image of the sample to read the positions at the blue flag position.

Default W or **Default W for RT module** value is set in such a way that between Z=0 to the pole piece, the maximum travel range can be achieved. When Z=0, W is at its maximum value. When the stage is at the pole piece, W = 0 and Z is at its maximum value. This recommended setting is set upon installation.

The **Pipette (Read)** button is used to read the current XY positions.

1.1 Origin Correction tab

The **Origin Correction** tab serves to determine the **Shift** vector between the origin of the XY coordinate system and the origin of the UV coordinate system.

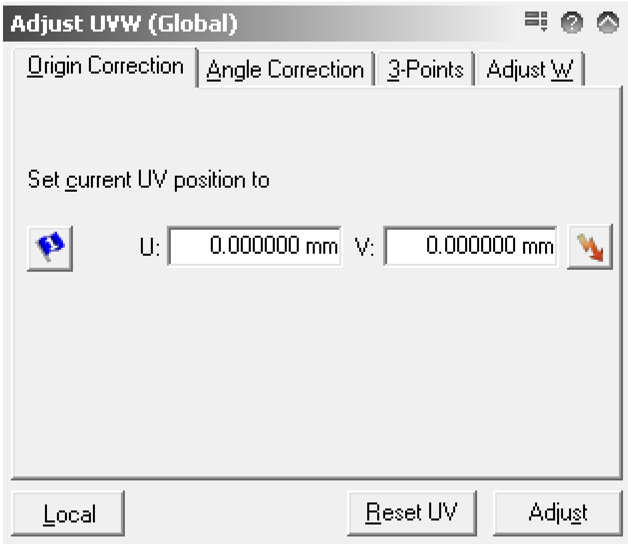


Figure 12-7: **Origin Correction** tab

Control element	Function
Set current UV position to...	To set the current UV position, the user can either: - use the blue flag to obtain the mark position from the image - type in the UV coordinates.
	To obtain the mark position from the image
	To drive the stage to the first mark position, once it is defined

1.2 Angle Correction tab

The **Angle Correction** tab offers the ability to adjust the angle between the X axis and the U axis. To calculate this transformation it is necessary to read in the **X** and **Y** of two different points along the U-axis of the sample.The accuracy of determination of the angle is related to the accuracy of the positioning as well as the distance between these two points. If the distance is larger, the angle is more accurately determined.

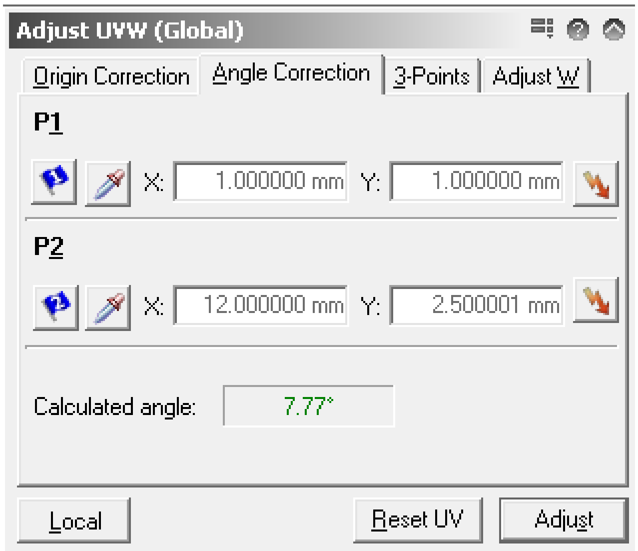


Figure 12-8: **Angle Correction** tab

Control element	Function
P1, P2	To obtain the mark position, the user can either: - use the blue flag to a obtain mark position from the image - use the **Pipette (Read)** button to read in the current XY coordinates.
	To drive the stage to the mark position, once it is defined.
Calculated angle	Once both mark 1 and mark 2 are defined as P1 and P2, the angle for the **Angle Correction** is automatically calcu-lated.

1.3 3-Points Adjustment tab

The 3-Points Adjustment tab enables the calculation of full transformation.

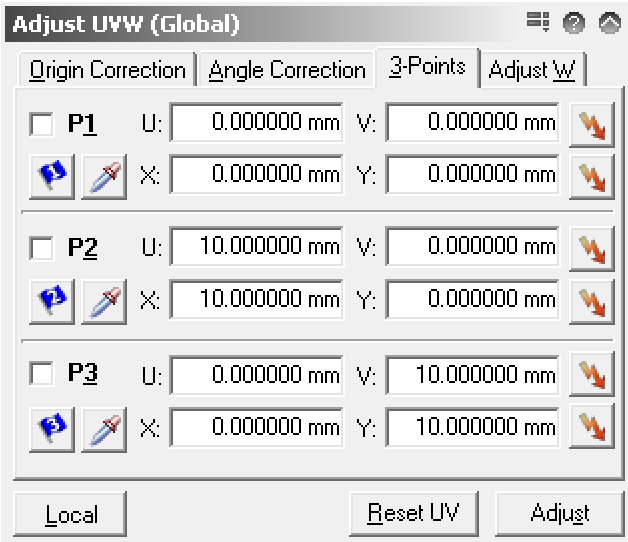


Figure 12-9: **3- Points Adjustment** tab

Control element	Function
P1, P2, P3	To define adjustment marks **P1**, **P2** and **P3**, check the checkbox, then the user may either: - use the blue flag to obtain a mark position from the image - use the **Pipette (Read)** button to read in the current XY coordinates - type in the **UV** or **XY** coordinates.
	To drive the stage to the selected mark position, once it is defined.
Adjust	Once marks 1, 2 and 3 are defined, the **3-Points Adjustment** can be carried out by clicking on the **Adjust** button.

1.4 Adjust W tab

The **Adjust W** tab enables the adjustment of W via the working distance.

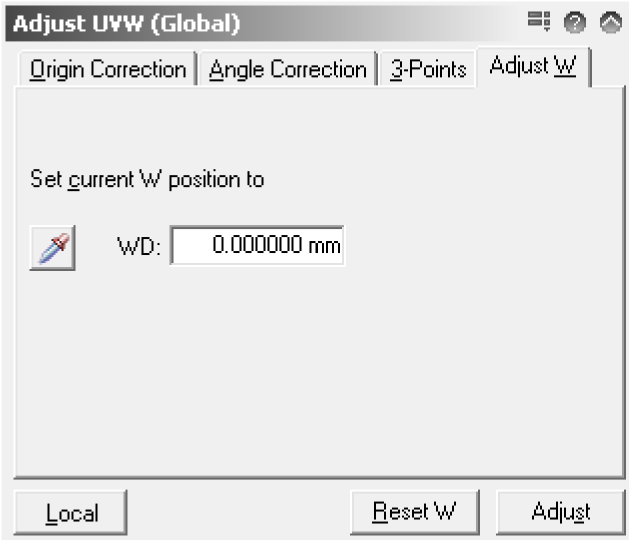


Figure 12-10: **Adjust W** tab

Control element	Function
Set current W position to...	To set the current W position, the user can either: - use the **Pipette (Read)** button to obtain the current working distance. - type in the **WD** (working distance) value.

W is the coordinate value along the Z axis, but in the opposite direction to Z. W=0 at the pole piece, when Z will be at its maximum value. The W value, therefore, is equal to the working distance. In this dialog, the focus may be set. If the beam is switched on and focused onto the sample, the value that is then read is equal to the working distance. The user should then activate **Adjust W**.

NOTICE

Please note that incorrect W adjustment causes the wrong calculation of sample height. This makes also impossible the proper operation of **W-limits** (⇒ *W-limits*) and **Eucentric drive mode** (⇒ *Eucentric drive mode*).

It is not necessary to set W to the working distance while working without the RT module. Any value may be selected, but it is highly recommended to set it to the working distance, since at this setting, W becomes a simple orientation tool.

2 Context menu

By clicking on the **Menu** button in the **UVW Adjust** dialog or by **right mouse click**, there are two further settings available.

- **Get marks**
- **Options**

2.1 Get marks

This option is used to manually read in the mark positions from the Positionlist. In order to carry out the **Get marks** function successfully, it is essential that the correct format and sequence of marks is used in the Positionlist.

When the **Adjust UV** procedure is carried out via the **Scan Manager**, the marks will be read in automatically by the software and this is the recommended procedure.

Get marks should only be used for testing and when the user is very familiar with the system. Using this command, the user may verify that the **Adjust UV** is working correctly.

2.2 Options

When selecting **Options**, a new dialog, **Adjust UV Options**, will open.

In the **Adjust UV Options** dialog, the user may determine the conditions under which the **Adjust UV** should not be implemented, for example, if not enough of the marks could be read in by the software. Setting adjustment limits is particularly important for automatic adjustment procedures that have been created in the **Scan Manager**.

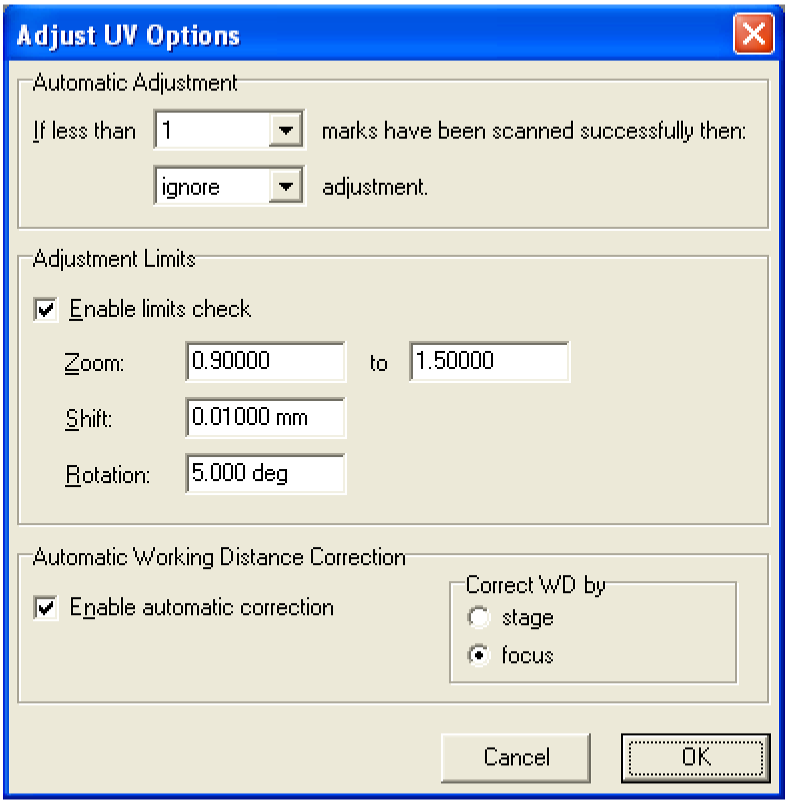


Figure 12-11: **Adjust UV Options** dialog

Control element	**Function**
Automatic Adjustment subset:	
If less than...	To define the minimum number of marks that must be recognized by the software.
Ignore/Pause	If the number of recognized marks is less then the number of minimum required marks, the user can choose to either **ignore** or **pause** the adjustment.
Adjustment Limits subset:	
Enable limits check	To enable the limit check The user may enter the maximum allowed value for **delta**, which is the relative deviation from the set value.
Zoom	To set the minimum and maximum permitted **Zoom** deviations
Shift	Permitted **Shift** can be defined in mm. The user may enter the maximum allowed value for **delta**, which is the relative deviation from the set value.

Control element	Function
Rotation	Permitted **Rotation** can be defined in degrees. The user may enter the maximum allowed value for **delta**, which is the relative deviation from the set value.
Automatic Working Distance Correction subset:	
Enable automatic correction	To **Enable/disable automatic correction** of working distance.
Correct WD by Stage/Focus	The user can then decide to carry out the working distance correction via either of: - **stage** - **focus**

Chapter 13
Beam to Sample Alignment

The Beam to sample alignment topic is key to understanding the working principle of the system. Calling up **Writefield Control**, using the icon in the control bar, opens several dialogs, which will be explained in detail.

The following topics and tasks are included:

- The functional principles of stitching, with and without alignment (⇒ *Functional Description* on page 13-3),
- An overview of the coordinate systems and transformations (⇒ *Coordinate systems and transformations* on page 13-4),
- The relationship between alignment and adjustment, explained using diagrams (⇒ *Alignment/Adjustment interrelation description* on page 13-5),
- The functional principle of **Beam Tracking Alignment** (⇒ *Beam Tracking Alignment description* on page 13-6),
- The **Writefield Alignment** procedure, explained step by step (⇒ *How to carry out a Writefield Alignment* on page 13-8),
- The differences between Writefield alignments with and without a laser stage, outlined with the aid of diagrams (⇒ *Writefield Alignment with and without a laser stage* on page 13-6),
- Description of the **Writefield Manager**, in which writefield settings and parameter sets can be edited (⇒ *Writefield Manager dialog* on page 13-15),
- A description of the **Writefield Alignment** dialog to perform writefield alignment for **Zoom**, **Shift** and **Rotation** (⇒ *Writefield Alignment dialog* on page 13-18),
- Explanation of the optional **Beam Tracking Alignment** (⇒ *Beam Tracking Alignment dialog* on page 13-21),
- A description of the **Beam Tracking Control** dialog, used to check the beam tracking functions (⇒ *Beam Tracking Control dialog* on page 13-21),

---

It is recommended to read the chapter **Stage to Sample adjustment** to enhance understanding of the functional principles of the system, (⇒ *Functional Description* on page 12-3).

The user may call up **Writefield Control** by clicking the (Writefield Control) icon in the control bar.

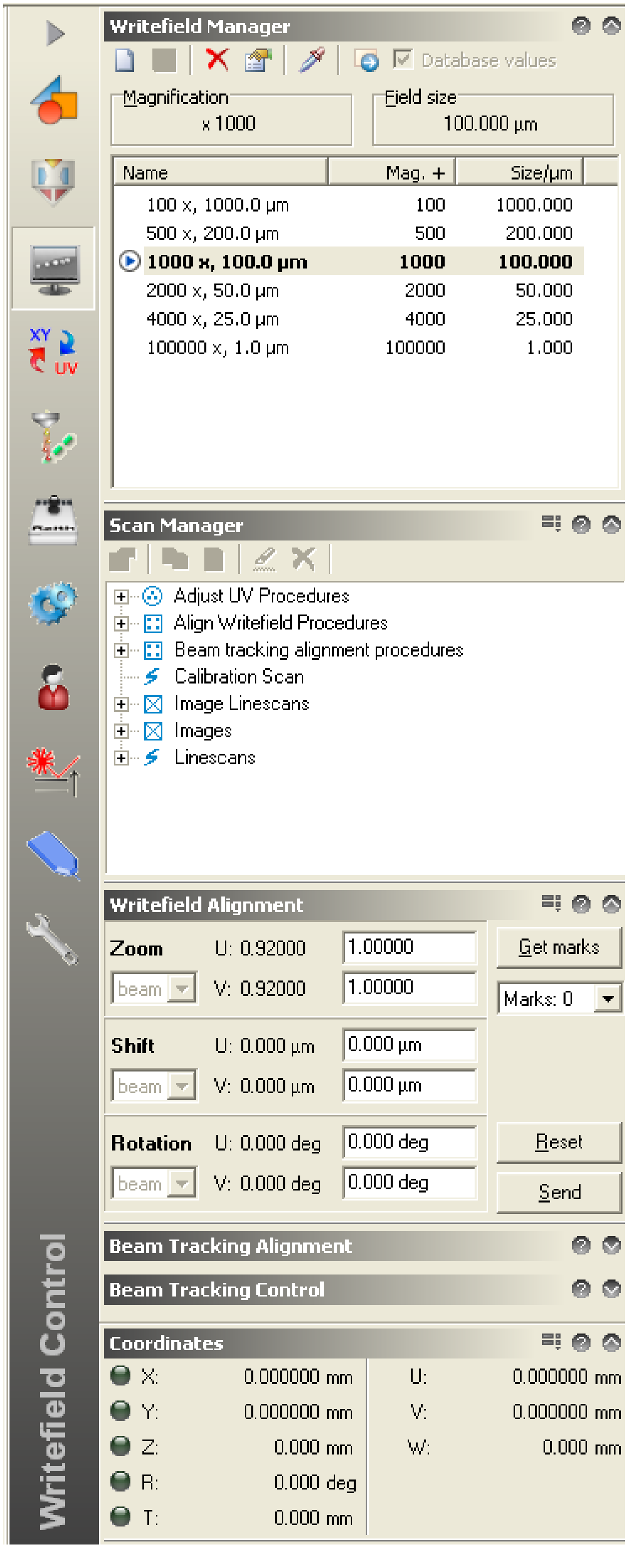


Figure 13-1: Writefield Control

A Functional Description

1 Stitching with and without Alignment

Writefield alignment is central to the process of accurately writing a new pattern into an existing pattern, known as **Mix and Match**, as well as writing large structures consisting of several smaller writefields, known as **stitching**. The effects of good and bad writefield alignment on stitching are displayed in the figure below.

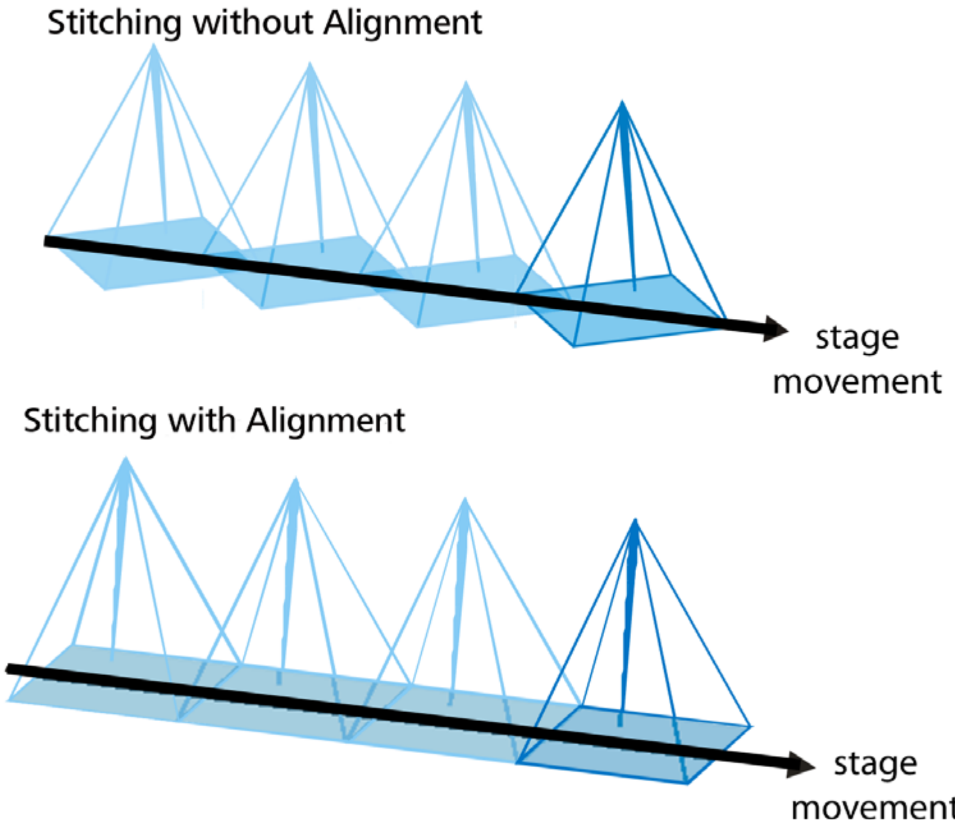


Figure 13-2: Effect of writefield alignment on stitching

In certain situations, the size of the writefield is even more important than the orientation of the writefield. Examples are metrology applications, in which images or linescans are recorded to determine a specific dimension and in CD control, in which elements of specific widths are written.

In all of these cases, it is clear that the beam must be positioned very precisely on the sample, so the beam positioning has to be correctly aligned to the sample coordinate system.

To give an overview of the beam-to-sample coordinate transformation, the first part of this chapter introduces the coordinate system of the beam and its transformation in relation to the sample.

Note that throughout the software and manuals, the terms **align** or **alignment** are used only in connection with the transformation between beam and sample. The terms **adjust** and **adjustment** are used in connection with the transformation between sample and stage.

2 Coordinate systems and transformations

In the chapter **Stage to Sample Adjustment**, the UV-coordinate system of the sample was introduced, whereby a point on the sample can be placed beneath the beam by specifying its U and V coordinates, (⇒ *Coordinate systems and transformations* on page 12-3).

The beam, of course, is usually not static and so the parameters which define the movements of the beam must be given a relationship to the UV coordinate system used for defining the pattern. The key parameters are:

- **Zoom U** and **Zoom V** are the scaling of the beam major axes with respect to the U and V axes. Zoom values are required for the correct scaling of the beam movement. For example, if an image is scanned, the dimensions of the displayed structure should fit exactly to the real size of the structure.
- **Shift U** and **Shift V** are the offset of the beam center with respect to a point in U and V. If the shift is determined correctly, a perfect overlay pattern can be achieved.
- **Rot U** and **Rot V** are the rotation of the beam major axes with respect to U and V axes. The rotation values are responsible for the correct orientation of the beam movement relative to the sample coordinate system. If the rotation is correctly aligned, accurate stitching of a series of writefields is achieved.

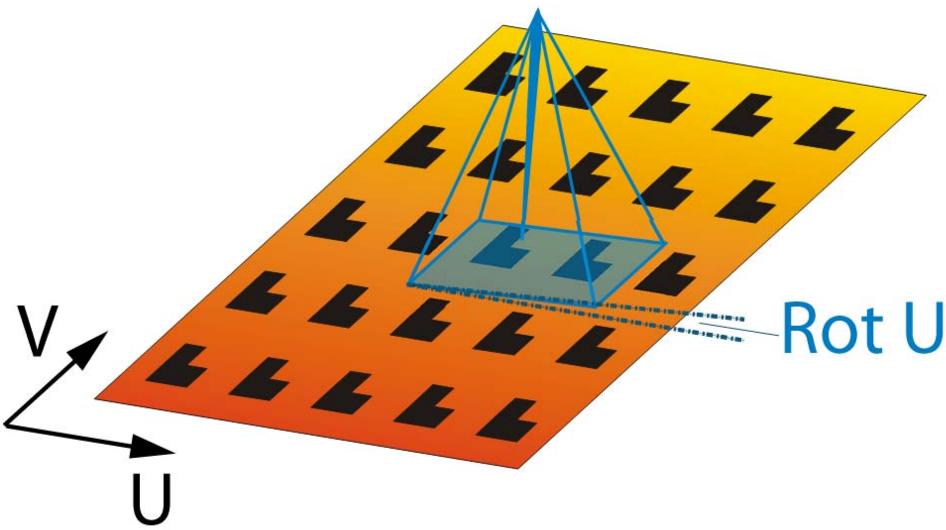


Figure 13-3: The beam system with respect to the sample

The exact values for the above parameters are sent to the **Digital Pattern Generator** or to the **Scanboard**, where these values are used to perform the coordinate transformation electronically.

The way in which transformation between the sample and the beam fits into the general scheme of coordinate systems is displayed in the figure below.

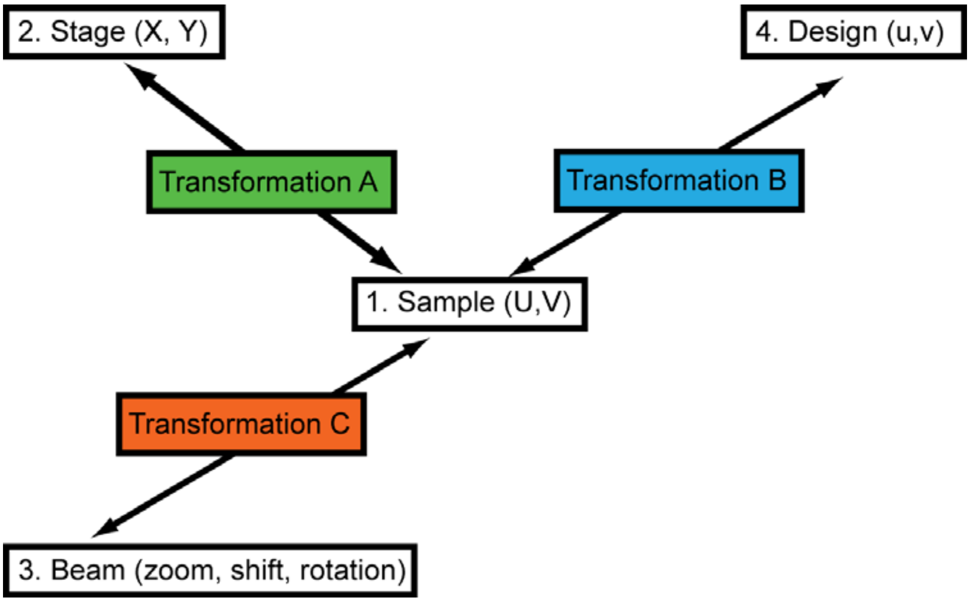


Figure 13-4: Sample to beam coordinates transformation C.

3 Alignment/Adjustment interrelation description

Since the marks used for an alignment are features on a sample, the sample orientation or dimensions on the sample affect the alignment itself. For example, if the sample is rotated by certain angle with respect to the stage, then after an alignment, the writefield will be rotated by the same angle (assuming that the stage and the beam are perfectly aligned to each other). Hence, one can use the information from the UV-adjustment procedure to roughly calculate the rotation and scaling of the writefield.

From version 4.0 onwards, a mechanism is implemented which takes this interaction into account. This mechanism consists of two steps.

- Before storing the writefield parameters to the database, the current UV-adjustment values are used to calculate new writefield parameters, which do not depend on the UV-adjustment values. Only these values are stored.
- Before setting a writefield with parameters from the database, the current UV-adjustment values are used to calculate effective writefield

parameters. These effective values do depend on the current UV-adjustment.

In the menu **Extras➡Settings➡Writefield Manager➡Use UV Transformation = ON/OFF**, the **UV Transformation** can be defined as either **ON** or **OFF**.

3.1 Writefield Alignment with and without a laser stage

Writefield Alignment may be carried out using two different procedures, depending on the type of stage installed.

When a Laser stage is installed, it can be used as a measuring tool. The user will drive to the corners of the sample with the beam blanked and the distances driven by the stage are then used for writefield alignment.

When no Laser stage is installed, writefield alignment can still be carried out with respect to **Zoom** and **Rotation** values. When a structure needs to be written with exact values, a calibration standard such as **Chessy** can be used to achieve writefield alignment.

The parameters **Zoom** and **Rotation** are required to obtain accurate stitching. In order to achieve an accurate overlay, the additional parameter **Shift** is required.

All writefield alignments are carried out using the **Scan Manager**. There are several procedures available, such as **Manual** and several **Automatic** procedures, with and without images, linescans etc.

The transformation quantities **Zoom U**, **Zoom V**, **Shift U**, **Shift V**, **Rot U** and **Rot V** may be displayed and modified in **Writefield Alignment**. In addition, the **Control bar** section gives access to all relevant commands for handling the transformation.

4 Beam Tracking Alignment description

Beam Tracking is a hardware function, which will address small inaccuracies in beam placement, arising from the resolution, hysteresis and acceleration effects of the sample stage. The stage itself cannot place the beam in exactly the position defined for each dot in a pattern at the nanometer scale, because of the above sources of inaccuracy. Beam tracking compensates these effects at each position in the pattern, by deflecting the beam over the very small distance from the current position that the stage has reached to the target position.

This beam tracking at each dot position is a very fast process, compared to the stage movement itself and so it does not significantly add to the time taken to complete the pattern, it simply ensures that the pattern is written accurately and precisely.

Within **Stage Control**, all target positions for the pattern are stored. At each dot position, a comparison is made between the target position and the obtained position achieved by the laser interferometer stage. The magnitude and direction of the small delta between the target position and the obtained position are used to determine the beam deflection that is required to place the beam in exactly the target position.

A beam tracking calibration routine is performed in the **Scan Manager**, in order to determine the expected magnitude of the delta between the target and the obtained positions. This ensures that the re-orientation of the beam exactly matches the stage movement discrepancy.

Only the **Zoom** and **Rotation** values are used for **Beam Tracking Alignment**. The **Shift** value is only used for **Writefield Alignment** procedures. During the beam tracking alignment procedure, a shift value is created, but it is added to the existing Shift value that was determined in the **Align Writefield** procedure.

Beam tracking is analogous to making a very fine adjustment to the alignment of the writefield and so the beam tracking alignment procedure is very similar in structure to the **Writefield Alignment** procedure in which the **Zoom** and **Rotation** values for the pattern generator also need to be calibrated.

Several iterations of the **Beam Tracking Alignment** may be required, to achieve the optimum coincidence in the target and obtained positions. The number of iterations will be reduced if the **Preset** button is used first.

The main difference between **Beam Tracking Alignment** and a **Writefield Alignment** procedure is that the **XY** coordinate system is used for Beam Tracking and the **UV** coordinate system is used for Writefield Alignment. The motor controller operates in the XY coordinate system and generates the signal for beam deflection and, therefore, the beam deflection itself is also in the XY coordinate system.

B Tasks

1 How to carry out a Writefield Alignment

The principle of a Writefield Alignment is simple. Some unique and easily recognizable features on the sample are selected as marks. Scanning over these marks with the beam causes each mark to appear at a certain position. The current positions of the features relative to their reference positions lead to the input parameters for calculating the writefield transformation.

The alignment procedure is embedded into the general sequence as follows:

- Focus on the sample surface.
- Adjust UV coordinates with respect to the stage XY.
- Select the desired writefield size in **Microscope Control**.
- Perform writefield alignment using the corresponding procedures in **Scan Manager**.

It is helpful to understand that a writefield alignment depends on the current transformation between sample and stage. Therefore, before starting a writefield alignment, you should firstly establish the coordinate transformation between the stage and the sample, i.e. perform a UV Adjustment.

Two different alignment procedures are possible, depending on whether a laser interferometer stage is installed or not. Alignments with and without laser stage are described.

To display writefield alignment procedures in the **Scan Manager** for a laser stage system, go to the file menu **Extras**➡**Settings**➡**Scan Manager**➡**Laser stage = On**.
When there is no Laser stage, the variable **Laser stage = Off** is selected and procedures for alignment with the sample Chessy will be displayed. This will be set correctly on installation.

1.1 Writefield Alignment without laser stage

If your system is not equipped with a laser interferometer stage, use a precise and well-structured sample, for example the **Chessy** sample. This sample serves as a measuring standard for the following writefield alignment.

It is possible to align parameters such as **Shift** and **Rotation** using a **Chessy**, but there are two limitations which make a stitching application impossible. Firstly, the Chessy and the sample would need to have the same UV adjustment and the same focus position (exactly the same height). Secondly, the stage must have an accuracy of a few nm to perform stitching.

Normally, the alignment for each writefield size is performed during installation, before using the system. These values are then stored in the database. This makes it possible to start work with these stored parameters without the prior need for alignment.

The following steps are necessary to perform a Chessy alignment. For a more detailed description, refer to the Operation Manual.

- Focus on a leveled sample.
- Perform a UV Adjustment with **Origin Correction and Angle Correction** using the Chessy sample. Section **Alignment/Adjustment interrelation** to understand why this step is necessary. (⇒ *Alignment/Adjustment interrelation description* on page 13-5)
- Place the adjacent corners of two squares near to the center of the beam. Record an image at this position.
- If an applicable alignment procedure in **Scan Manager** exists, select this procedure and drag and drop it onto the adjacent corner at the image center. The result is displayed in the figure below. If an applicable procedure does not exist, create one as described in the section **Defining a new alignment procedure.** (⇒ *Writefield Alignment procedures* on page 11-33), **Writefield Alignment Procedures➡Manual**.

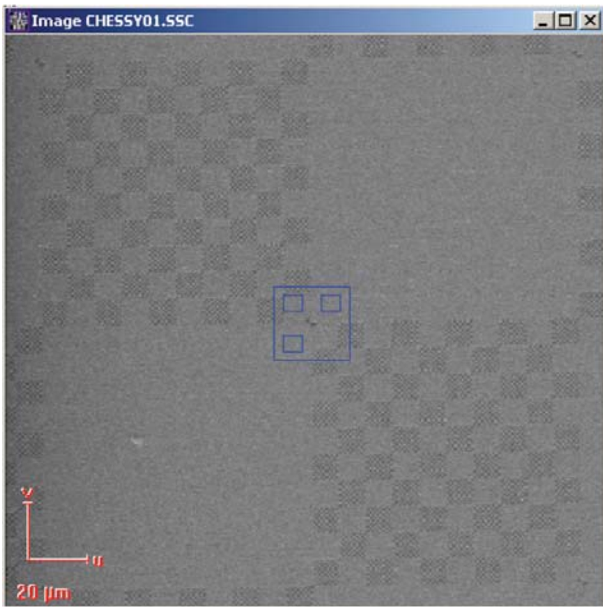


Figure 13-5: Image after drag and drop of an applicable procedure onto Chessy

- ▶ Select the Positionlist in which the alignment procedure has been stored and scan the corresponding position.
- ▶ In each image, place the blue marker on the mark by using the **Ctrl** key together with the **left mouse button**.
- ▶ At the end of the procedure, if the procedure was successful, **Apply** correction values.

1.2 Writefield Alignment with laser stage

If your system is equipped with a laser interferometer stage, it is possible to use the laser stage as measuring standard. The precision of the Laser stage allows a very precise alignment of the beam with respect to the sample, making stitching possible.

The internal procedure is not quite the same as the Chessy procedure described above. Here, only one mark is used. The mark is moved to several positions within a non-aligned writefield. At each position, a smaller image is recorded such that the difference between the current and reference positions can be determined. From the differences, the beam alignment values are calculated. The figure below illustrates this procedure.

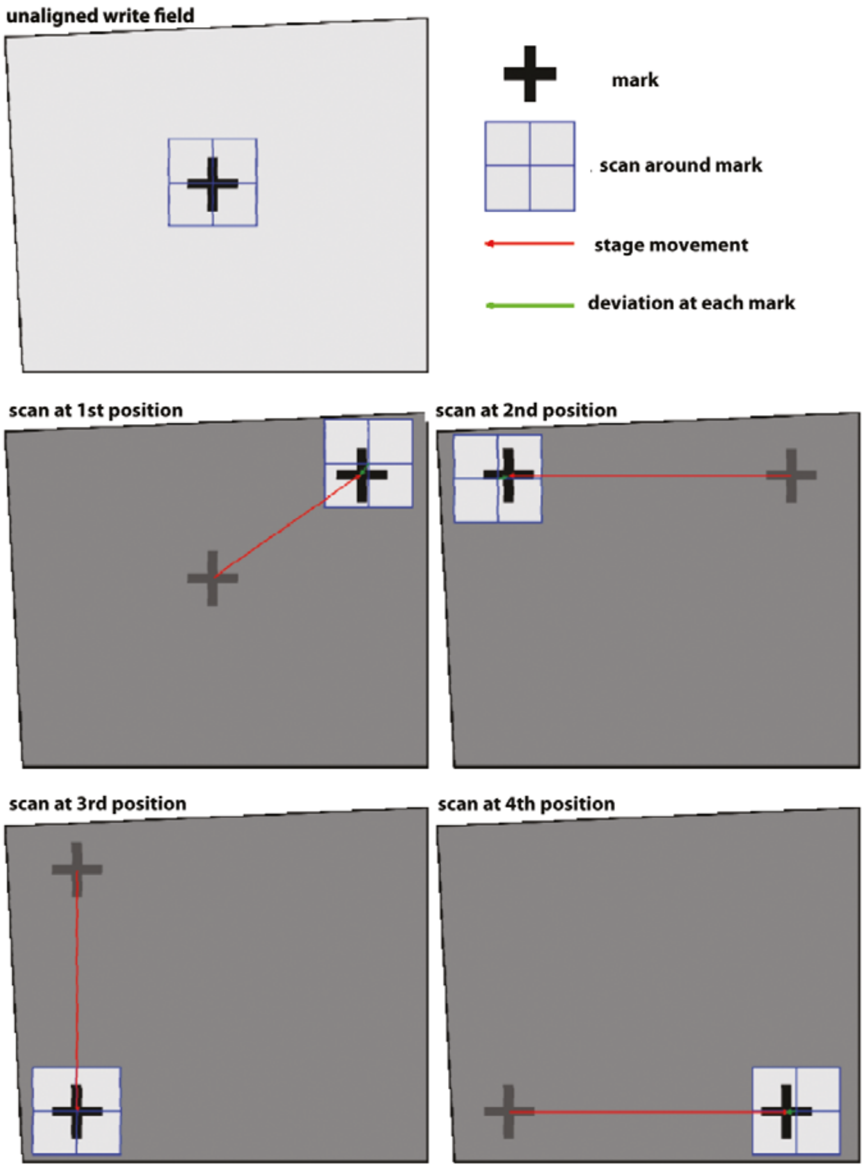


Figure 13-6: Writefield Alignment procedure used with Laser stage. One mark is moved, using the Laser stage, to different locations, so that the same mark appears in each scan.

Before this procedure can be executed, the following steps are required.

- Focus on a leveled sample.
- Perform a UV-Adjustment with **Origin Correction** and **Angle Correction** using the sample, (⇒ *Defining an Origin Correction* on page 12-9, ⇒ *Defining an Angle Correction* on page 12-9).
- Place the mark near to the current stage position. Record an image at this position.
- If an applicable alignment procedure in **Scan Manager** exists, select this procedure and drag and drop it onto the adjacent corner in the image

center (⇒ *Manual Writefield Alignment* on page 11-34), **Writefield Alignment Procedures➡Manual**. The result is shown in the figure below. If an applicable procedure does not exist, create one as described in the section **Defining a New Alignment Procedure**, (⇒ *Writefield Alignment procedures* on page 11-33).

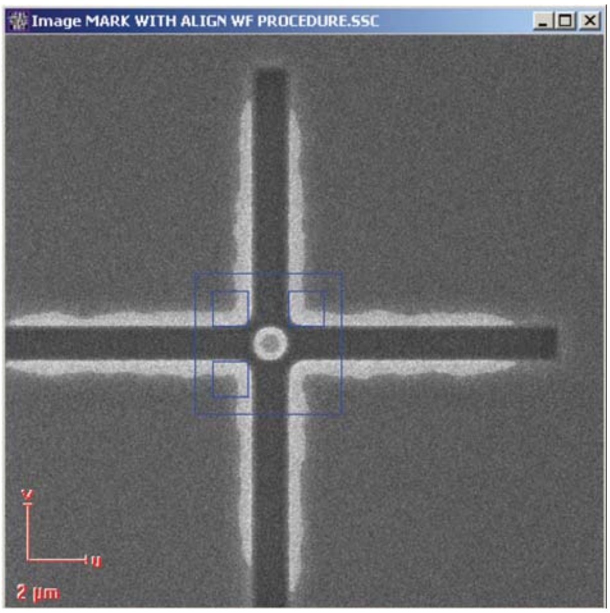


Figure 13-7: Image after drag-and-drop of an applicable procedure onto one mark.

- Select the Positionlist in which the alignment procedure has been stored and scan the corresponding position.
- In each image, place the blue marker on the mark, by using the **Ctrl** key together with the **left mouse button**.
- At the end of the procedure, if the procedure was successful, **Apply** correction values.
- Repeat this sequence with smaller scan fields until no decrease in correction factors is observed.

To use the beam tracking option, it must be activated by clicking the **Beam Tracking** button in the main tool bar of the Raith NANOSUITE Software. Once activated, beam tracking will be performed whenever there is a discrepancy between the target and obtained positions. A beam tracking signal from the motor controller is used as an input signal for the pattern generator.

On Off

Figure 13-8: Beam Tracking button

▶ Before performing beam tracking alignment, the **Preset** button should be clicked in the **Beam Tracking Alignment** window. This ensures that the starting point for the beam tracking alignment is the set of parameter values from the **Writefield Alignment**.

Click **Preset** for **Zoom** and **Rotation** before starting the beam tracking alignment. This will ensure that the user has a good starting point for these parameters at the beginning of the procedure.

If the **Preset** is not clicked, beam tracking alignment can still be carried out, but if the starting point is further away from the optimum values, more than one iteration may be required and the overall process may take longer to complete.

▶ You will see that the procedure has the same steps as the **Writefield Alignment** procedure. During the procedure, the user will not notice any difference, since the procedures are structured in the same way in the **Scan Manager**.

Within **Scan Manager**, the pre-written procedures and sub-procedures have a similar appearance. The user must set marks and the resolution and average values for gray scales. The marks are then placed into the writing field.

▶ The procedure will now be carried out, but the beam deflection in XY is controlled by the motor controller in this case and not by the pattern generator. The required beam deflection to place the beam at the target position is achieved using a dedicated signal from the motor controller.

Beam Tracking Alignment must be carried out every time that the **Writefield** has been changed or if a **UVW Adjustment** has been carried out, since each new Writefield Alignment will change the beam to sample alignment, as well as the **Zoom** and **Rotation** values. These new values will require different beam tracking parameters. The new beam tracking parameters may be only slightly different to the previous ones, but alignment should be repeated, to achieve optimum beam positioning performance at the nm scale.

Beam Tracking Alignment should be carried out after a Writefield Alignment procedure has been performed. This means that the adjustment marks for the Beam Tracking procedure will have exactly the correct size and orientation.

In **Writefield Alignment**, a combination of rotation of the beam, using the column deflection unit, with rotation via the Raith pattern generator, is often used. For example, if the **Column Rotation Step** (CRS) is set to 10 degrees, the following formula will calculate the rotation as follows:

CRS = Column Rotation Step

$$\alpha = \text{Writefield rotation angle}$$
$$\alpha = 23°$$

CRS=10°

$$\text{column rotation} = \mathrm{ABS}\left(\frac{\alpha}{\mathrm{CRS}}\right) \times \mathrm{CRS} = 2 \times 10° = 20°$$

$$\text{Pattern generator rotation angle} = \alpha - \text{column rotation} = 23° - 20° = 3°$$

Since **Beam Tracking Alignment** is carried out in the XY coordinate system, the beam tracking alignment must firstly counter-rotate the 20 degree column rotation carried out by the column deflection software. This rotation was part of the transformation to the UV coordinate system, which is the sample coordinate system. The Beam Tracking Alignment, however is carried out in XY so the rotation of the column must be subtracted first, to get to the XY basis.

Set **Extras**➡**Settings**➡**Writefield Alignment**➡**Use column rotation = ON** to enable the rotation compensation by column. The **Column rotation step** defines the angle at which the column rotation will be activated.

In this example, when the rotation angle has been set to 20 degrees in the column deflection software and the **Send** button has been clicked to activate it, clicking on the **Preset** button in **Beam Tracking Alignment** will return a **Rotation** value of -20 degrees in the Beam Tracking Alignment window.

C Software Reference

The user may call up **Writefield Control** by clicking the (Writefield Control) icon in the control bar.

Writefield Control consists of the following dialogs:

- **Writefield Manager** manages writefield settings and parameter sets, which the user can edit.
- **Writefield Alignment** enables the user to perform writefield alignment for **Zoom**, **Shift** and **Rotation**.
- **Beam Tracking Alignment** enables the alignment procedure for beam tracking, which can be installed as an option.
- **Beam Tracking Control** can be used by Administrators or service personnel to check the beam tracking functions.
- **Scan Manager** contains procedures to carry out Writefield Alignment, Image Linescans, Linescans, Images etc. Please refer to the **Scan Manager** chapter for detailed explanations, (⇒ *Writefield Alignment procedures* on page 11-33).

1 Writefield Manager dialog

The transformation quantities of the writefields and the magnification setting of the electron or ion optics are inter-related. For example, when setting up a writefield of 100 µm, the field of view must be larger than 100 µm. In addition, the correct transformation quantities may depend on the beam energy, working distance and also slight differences in linearity when using different amplification for different magnification regions, etc. Thus each writefield must be separately aligned for every magnification. Since writefield size, magnification and the transformation quantities depend on each other, they are stored together as a set in one database and each set is identified by name.

1.1 Tool bar

The following buttons are available in the tool bar.

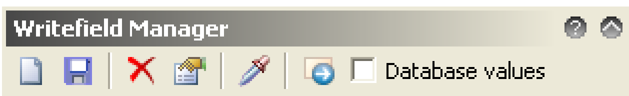
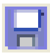


Figure 13-9: **Writefield Manager** tool bar

Control element	Function
	To create a new parameter set of writefield size with corresponding magnification and to assign a name to this parameter set, so that it can easily be identified. If no name is given, a name is automatically created using a combination of magnification and field size. By using names, it is possible to create several parameter sets with equal writefield size but different magnifications and vice versa.
	To store changes in the currently selected parameter set to the database (only red and blue entries are affected).
	To delete an existing parameter set
	To edit an existing parameter set
	To initiate reading of the current magnification from the column
	To activate the currently selected writefield size and magnification. If **Database values** are selected, the beam alignment values are read from the database before activating them. Otherwise, current values are still active.
	To set the **Writefield Manager** to the set Database values

To set up a new parameter set, it helps to understand that the product of writefield size and magnification should be constant. The current value of the product itself depends on beam control hardware, as well as the column.

1.2 Writefield Manager working area

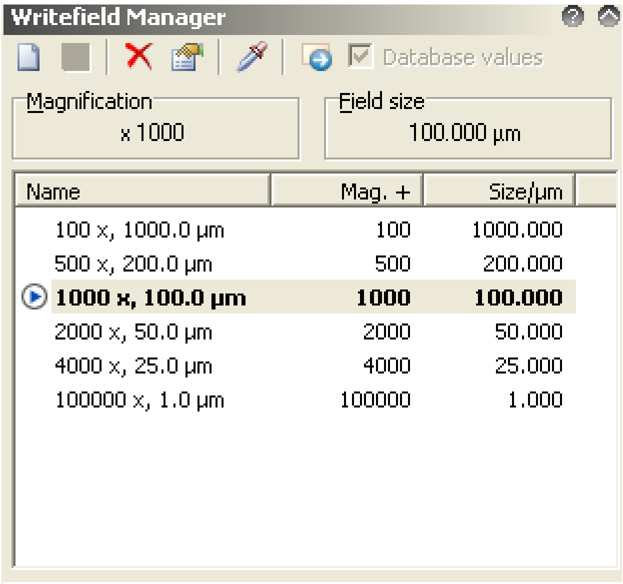


Figure 13-10: **Writefield Manager** dialog

The currently selected **Magnification** and the **Field size** are displayed at the top of the dialog. In the case of a magnification that does not fit to a selected writefield, the magnification is displayed in red. The list is used to select a new writefield.

Within this list, the various writefields are selected by name and the active writefield is marked by a blue arrow. If the alignment values stored in the database differ from the current alignment values, the corresponding entries in this list are displayed in blue.

A new entry is displayed in red, if it has not been stored in the database. Black entries are parameters for which database settings agree with the current setting in the system.

2 Writefield Alignment dialog

The **Writefield Alignment** dialog performs the alignment of **Zoom**, **Shift** and **Rotation** using mark alignment.

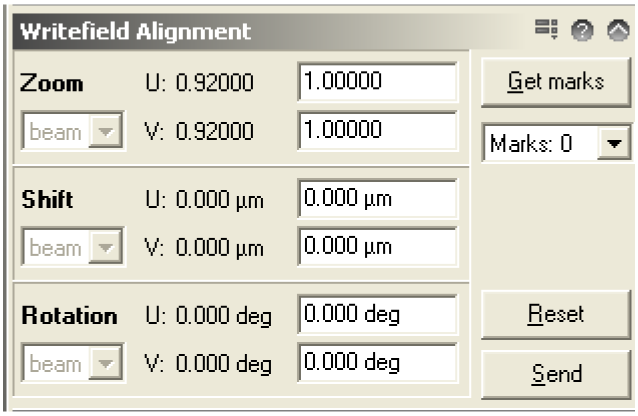


Figure 13-11: **Writefield Alignment** dialog

Control element	Function
Zoom U, V	Zoom for scaling of U and V axes. Possible values are in the range 0 to 1.
Shift U, V	Shift with respect to U and V axes can be positive or negative. Raith systems can compensate a large shift by applying the corresponding value to the transformation between sample and stage. The stage shift can be either switched **On** or **Off** in **Extras**➡**Settings**➡**Writefield Alignment**➡ **Use Stage Shift**➡**On** or **Off**.
Rotation U, V	Rotation relative to U and V axes can be positive or negative. The beam compensates rotation of the UV system, therefore the rotation angles of the UV adjustment and the beam have different signs. Raith systems can compensate any rotation. It is possible to compensate rotation by sending the rotation angle to the column software. This can be either switched **On** or **Off** in **Extras**➡**Settings**➡**Writefield Alignment**➡**Use Column rotation**➡**On** or **Off**.
Input boxes	The values given here are for the modification of the transformation quantities. The values for **Zoom** are multiples of the current values and values for **Shift** and **Rotation** are added to current values.

Control element	Function
Get marks	This command calculates the correction parameters from the Positionlist containing adjustment marks. Alignment procedures called from the **Scan Manager** perform this command automatically.
Mark	Select the mark from the dropdown list.
Reset	Resets the correction parameters to default values, which are **Zoom** = 1 and **Shift/Rotation** = 0.
Send	This will initiate the Writefield Alignment

2.1 Writefield Alignment Options menu

You can call up the **Writefield Alignment Options** menu by clicking on the **Menu** button in the **Writefield Alignment** dialog.

The user may define acceptable delta values and if these are exceeded, the new alignment will not be applied.

When a Writefield Alignment procedure with several marks has been selected, it may be that one or more marks are not recognized by the software. This may be due to a read error or because they have not been read in accurately enough. The user may define how many marks must be recognized in order for the Writefield Alignment procedure to be accepted and carried out. If there are too many read failures, the user may select to **ignore** the errors or **cancel** the patterning procedure.

Control element	Function
Options	Opens the **Writefield Alignment Options** dialog.
Alignment➡Set zoom to default	Set the current zoom value for U and V to default value specified in **Extras➡Settings➡Writefield Alignment➡Initial zoom U/V**.
Alignment➡Set shift to zero	Resets only shift U and V of the current alignment to zero.
Alig-nent➡Remove rotation offset	Sets rotation U = 0 and V = old rotation V - old rotation U.

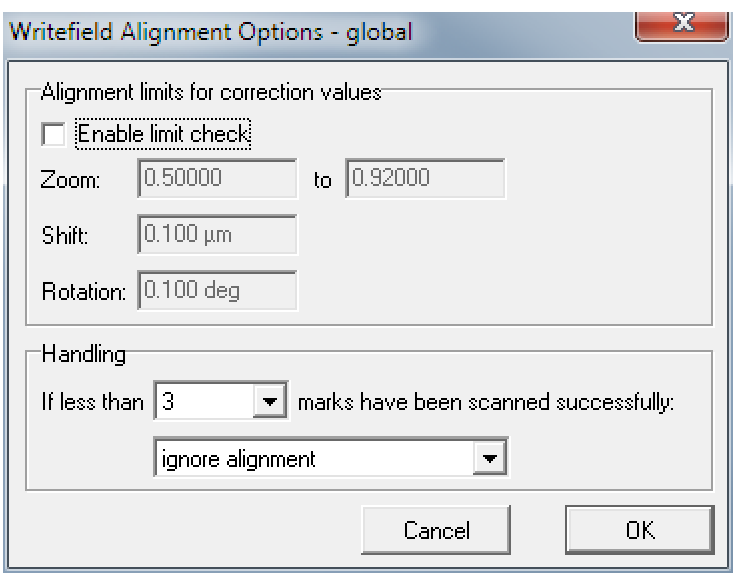


Figure 13-12: **Writefield Alignment Options** dialog

Control element	Function
Alignment limits for correction values subset:	
Enable limit check	To enable limit check facility
Zoom	The limit check values for the minimum and maximum zoom deviation can be entered.
Shift	The limit for the shift deviation can be entered.
Rotation	The limit for the rotation deviation can be entered.
Handling subset:	
If less than .. marks have been scanned successfully	The user can enter the minimum number of marks which have to be scanned and recognized successfully.
Action	The user can choose the action to be taken if the number of successfully scanned marks does not meet the defined requirement. - **Ignore alignment**: The procedure will continue with previous Writefield alignment settings. - **Cancel patterning**: The patterning procedure will be terminated immediately.

3 Beam Tracking Alignment dialog

Beam Tracking Alignment will be carried out for **Zoom** and **Rotation** values.

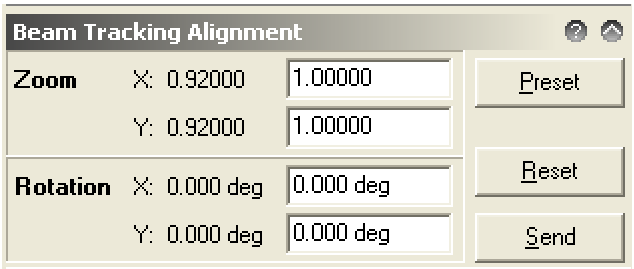


Figure 13-13: **Beam Tracking Alignment** dialog

Control element	Function
Zoom X, Y	Zoom for scaling of X and Y axes. Possible values are in the range 0 to 1.
Rotation X, Y	Rotation relative to X and Y axes can be positive or negative.
Preset	This will upload the latest values for **Zoom** and **Rotation** from the **Writefield Alignment** dialog. Only rotation compensated by the column software will be taken into account when **Preset** is activated.
Reset	This will reset the alignment to the default values.
Send	This will initiate the beam tracking alignment.

4 Beam Tracking Control dialog

This dialog is not visible to all users, only to **SYS level users**. This dialog enables testing of the deflection output voltage from the motor controller, which is used to achieve beam deflection to the target position from the obtained position. Test values may be entered into the fields and the output voltages can be measured to check that they are correct. This is a useful feature if difficulties with beam tracking have been experienced.

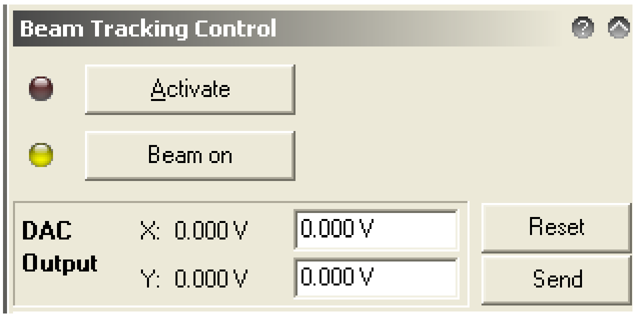


Figure 13-14: **Beam Tracking Control** dialog

Control element	Function
Activate	To activate the beam tracking control
Beam on	To switch the beam on
DAC Output X:	To display the X DAC output voltage
DAC Output Y:	To display the Y DAC output voltage
Reset	To reset the output voltage to the default value of zero
Send	To activate the entered values
Activate	To enter/leave path control mode

Use the **Beam on** button to switch the beam blanker **On/Off.** This is useful, since during beam tracking and FBMS procedures, the motor controller sends signals for the activation and deactivation of the beam blanker. This is in addition to the beam blanker functionality controlled from the pattern generator, which also sends activate and deactivate signals to the beam blanker.

When patterning FBMS structures, in which **Beam Tracking** is also activated, the beam will be unblanked before the start and blanked at the end of the patterning process via the motor controller.

Chapter 14
Patterning

This chapter gives an overview of the available patterning parameters for **Areas**, **Lines**, **Curved elements**, **Dots** and **FBMS elements**. Some background information is given to aid understanding of the patterning process.

The chapter consists of the following;

- Background information on the **Patterning Parameters** for standard elements and **FBMS Elements.**
  (⇒ *Setting patterning parameters for standard elements* on page 14-3),
- Explanation of the patterning principles of FBMS elements, using diagrams
  (⇒ *FBMS (option)* on page 14-6),
- Explanation of the patterning principles of MBMS elements, using diagrams
  (⇒ *MBMS (option)* on page 14-11),
- How to set the **Module Settings** to perform multi-stitching procedures
  (⇒ *Setting the Module Settings for single field patterning* on page 14-13),
- Step by step guide to carrying out patterning via a **Positionlist**
  (⇒ *Patterning via Positionlist* on page 14-15),
- Detailed software description of the **Patterning** dialog.
  (⇒ *Patterning dialog* on page 14-17),
  (⇒ *Patterning Parameter subdialogs* on page 14-22)
- Explanation of the **Patterning Control** window, once the patterning procedure is executed.
  (⇒ *Grayscale Bitmap (option)* on page 14-54)
- Description of the **Beam Current** dialog.
  (⇒ *Beam Current* on page 14-59).

The user may call up **Patterning** by clicking the (Patterning) icon in the control bar.

Patterning comprises the following dialogs:

- **Patterning Parameters** to:
  - Select the patterning parameters for **Areas**, **Lines**, **Curved elements**, **Dots** and **FBMS elements**, calculate the patterning parameters and the patterning time.

- **Beam Current** to:
  - Display the beam current.

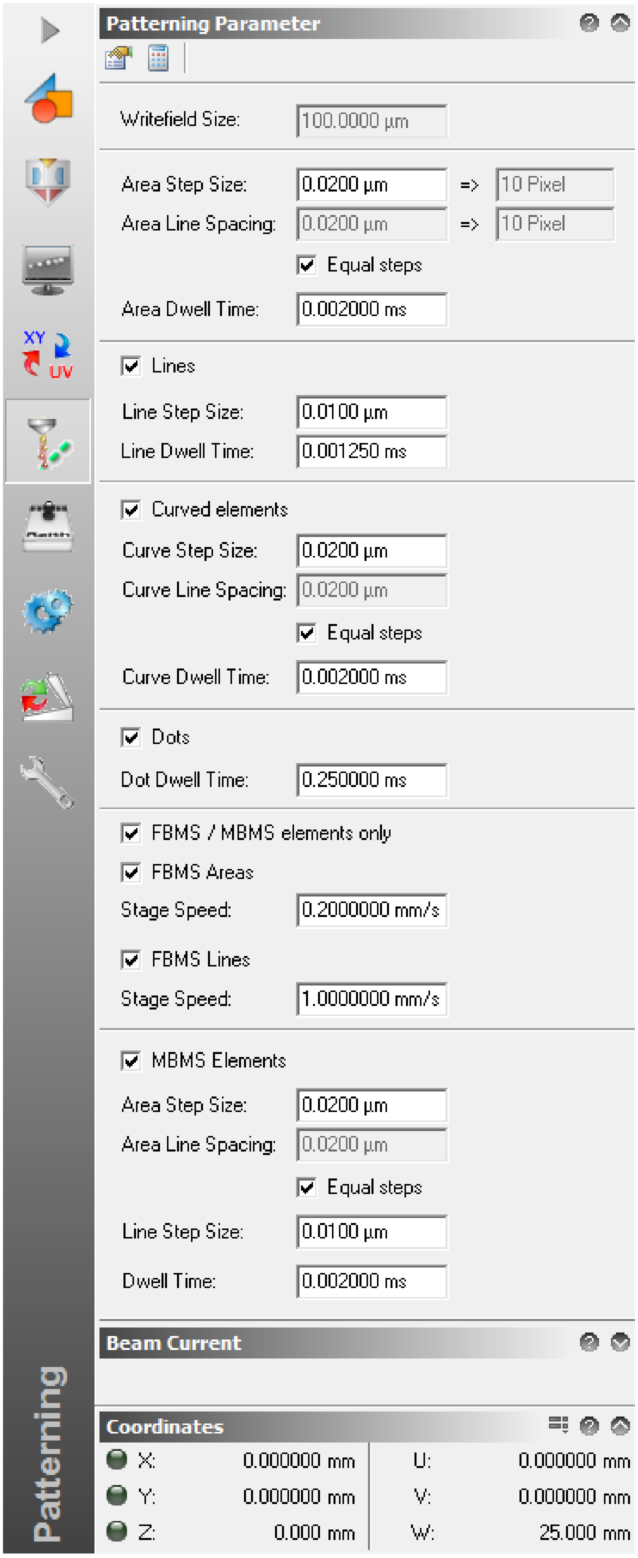


Figure 14-1: **Patterning Parameter** dialog.

A Functional Description

1 Setting patterning parameters for standard elements

The choice by the user of the patterning parameter values depends on the resist being used and so it is assumed that the user has some experience of resists and the required doses for each.

The geometric parameters that affect the patterning of a structure are the **Step Size**, which is the distance between dots, and the **Line Spacing**, which is the distance between each line of dots.

The minimum step size, also known as the pixel width, is hardware dependent and is proportional to the (2^{16} bit) DAC step resolution. The smallest step size can be calculated as:

$$\frac{\text{writefieldsize}}{2^{16}}$$

So, for example, for a 100 µm writefield, the smallest step, or pixel width is:

$$\frac{100\mu\text{m}}{2^{16}} \approx 1,6\text{nm}$$

In normal mode, without FBMS, the stage is fixed and the beam is moving. In this mode, the dwell time is a key factor in the patterning. The dwell time determines the writing time of the beam at each defined position on the sample. When the beam moves to each new position, the dwell time corresponds to the time during which the beam is held stationary at that position. The beam then moves to the next defined position and remains at that position for the duration of the dwell time.

1.1 Single Line mode

SPL (Single Pixel Line)

Figure 14-2: Single Pixel Line

To pattern a Single Pixel Line (SPL), the user selects the parameters **Step Size** and **Dwell Time.**

The beam is digitally positioned and the accuracy of the positioning is determined by the digital to analog converter (DAC) step resolution, as described above. The translation of the beam from each defined position to the next is very fast and the beam remains on whilst moving from one dot to the next. All patterns and lines, of any shape, including curved elements, are written as a series of dots. A rectangular area in normal mode can be written in one of two ways.

1.2 Line mode

When an area is written as a series of single lines, of defined line spacing, each line starts at the same end as the previous line. The beam is blanked between the end of one line and the positioning at the start of the new line.

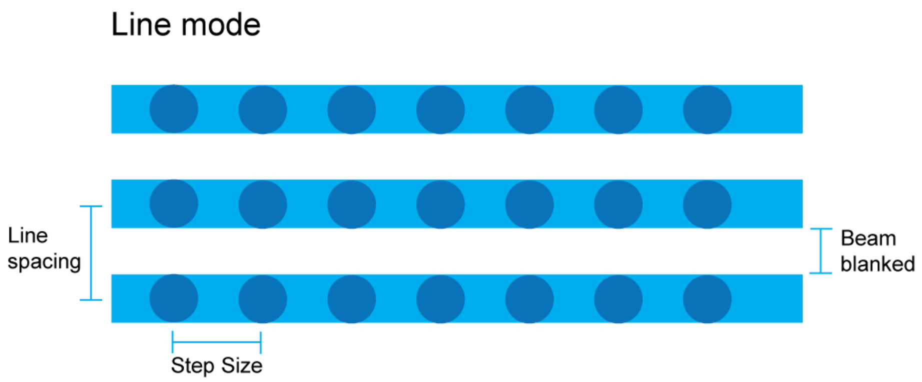


Figure 14-3: Area patterning in **Line mode**

1.3 Meander mode

Alternatively, a pattern may be composed of lines in a continuous meander. Once a line is complete, the beam moves perpendicular to the line direction, by a distance equal to the line spacing. The next line is written in the opposite direction. At the end of each line, the beam position is incremented by the line spacing and the next line is written in the opposite direction to the previous one. The beam does not need to be blanked between each line and will often result in faster patterning.

Meander mode

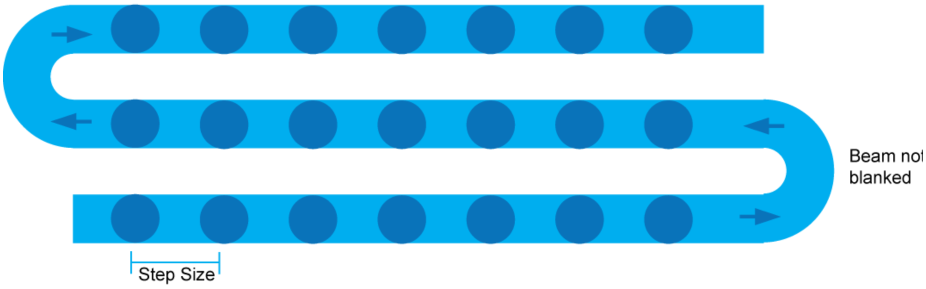


Figure 14-4: Area patterning in **Meander mode**

1.4 Dynamic compensation

When the blanked beam has been moved to the beginning of a new line within a structure and a settling time has been allowed, there is a short distance over which the speed of movement of the beam ramps up from rest to the normal speed. If this were to occur during the start of the patterning of the new line, a small length of the line would be over-exposed, since the beam would be traveling too slowly.

To avoid this error, the beam is first directed to a position close to, but not at, the first position for the patterning of the new line. The distance from the start position to the first point on the pattern is used as a short acceleration path for the blanked beam. When the line patterning is initiated, the first movement of the blanked beam is over this short acceleration distance, during which the speed ramps up to the required value. When the beam reaches the start of the pattern, it is up to the correct speed and the beam blanker is turned off for patterning at the start of the line, resulting in correct patterning at the very start of the line. Over-exposure does not, therefore, occur.

As well as a ramping up of the speed of the beam at the start, there is also a ramping down of the speed of the beam when movement along a line is ended. To prevent patterning errors caused by the ramping down of the speed of the beam, at the end of the patterning, the beam is blanked immediately and the beam is either stopped or moved to the next pattern. By blanking at the end of the pattern, no patterning errors are caused by the ramping down of the speed of the beam.

Dynamic compensation

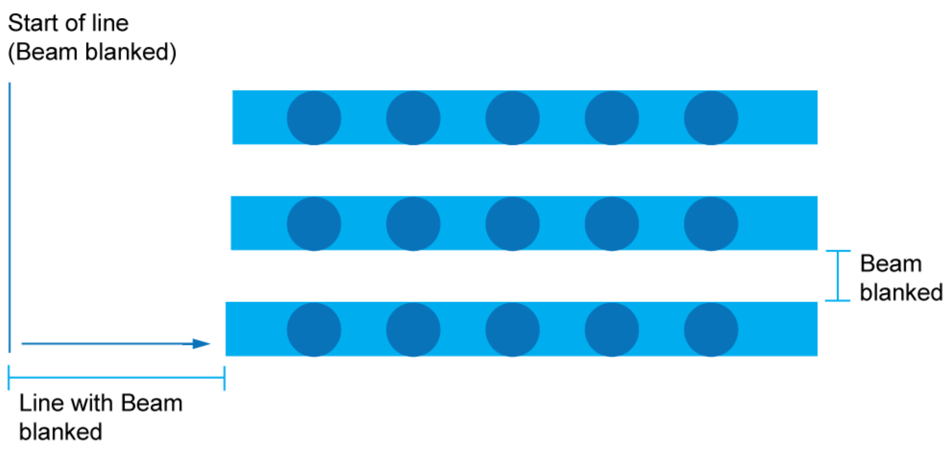


Figure 14-5: Dynamic Compensation

Dynamic compensation is not available for patterning in **Meander** mode.

2 FBMS (option)

FBMS (Fixed Beam Moving Stage) technology allows to expose smooth and stitch-error free paths of arbitrary curvature of any length, even up to several cm long, by maintaining the beam at a fixed position and then continuously moving the stage (and thus the sample) with respect to the beam. Raith's feature name for this unique lithography patterning mode is **traxx**.

Without FBMS, long paths (lines) are fractured into shorter lines that fit into a single write field. Each boundary between write fields is therefore subject to some amount of stitching error. FBMS removes all stitching errors of path structures because they are written in a continuous way with no breaks. Hence, FBMS is the ideal choice to fabricate optical devices like waveguides or zone plates or electrical devices, where even fracturing errors below 10 nm cannot be tolerated.

Two different structures are available using FBMS:

- **FBMS lines** (⇒ *FBMS Lines* on page 14-9) - paths with zero width and
- **FBMS areas** (⇒ *FBMS Areas* on page 14-9) - paths with a width larger than zero.

Both types of FBMS elements can be defined as straight paths and as curved paths.

Some GDSII designs may have a combination of FBMS elements and normal elements. In such cases, all normal mode elements are written first and then with the beam blanked, the stage is driven to the starting point of the first FBMS structure and FBMS mode is then enabled.

Patterning of FBMS and standard GDSII elements as well as patterning of FBMS lines and areas can be separately activated. Area and Line stage speeds can be defined individually, but are directly related to the patterning dose at a given beam current and are typically calculated by the patterning parameter calculator (⇒ *FBMS Patterning Parameter formula for Calculation Width* on page 14-23). Note that the real stage speed during the patterning can differ from the specified stage speed in the window. The stage speed is automatically adjusted whenever another dose factor than 1.0 is applied, and when the width of the FBMS area element differs from the **Calculation Width** chosen in the **Patterning Parameter** calculator window (⇒ *FBMS* on page 14-41).

2.1 FBMS beam deflection patterns

There are three different kinds of beam deflection patterns related to the FBMS:

- **Spot pattern** for FBMS lines,
- **Circle generator pattern** for FBMS areas and
- **User defined beam deflection** (⇒ *FBMS* on page 14-41).

In the spot pattern, the beam is simply kept at a fixed position in the center. The circle generator pattern is applied for FBMS area elements in order to achieve uniform area dosing for any possible direction of stage movement for the given width of the element. A single filled circle cannot be used for this purpose, since this would lead to a higher dose towards the center of the path. On the contrary, a single circle circumference is also not suitable, since this would lead to a higher dose towards the boundary. For this reason is special circle symmetric pattern is calculated by the software, where the distance between inner circles is greater than the distance between outer circles. The pattern results in an average step size perpendicular to the stage movement, which can be defined in the enhanced parameters for FBMS (⇒ *FBMS* on page 14-41).

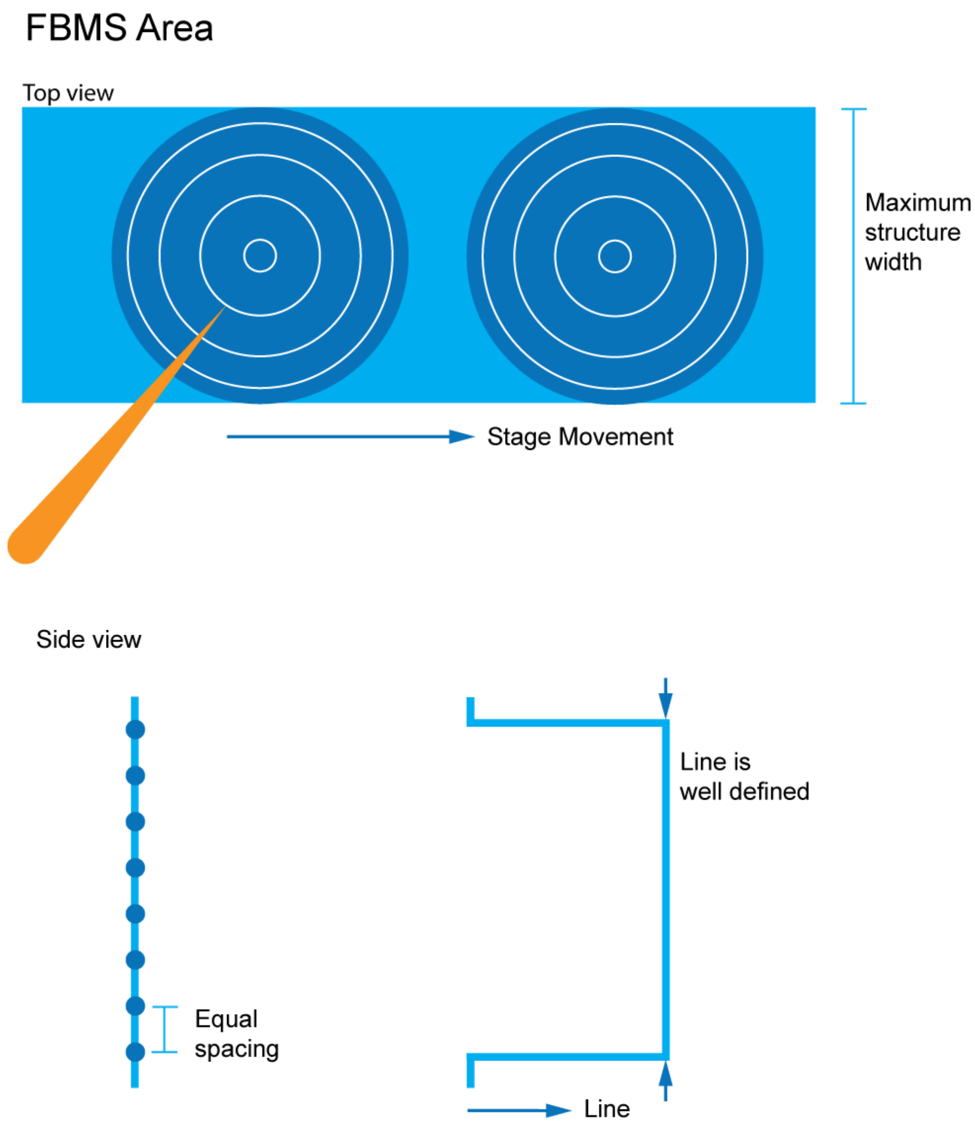


Figure 14-6: FBMS Area

The user defined beam deflection is a programmable mode that offers the possibility to define an individual dot pattern and use this together with FBMS lines instead of the single spot pattern. For further details on this mode see section enhanced parameter for FBMS (⇒ *FBMS* on page 14-41).

Both user-defined and circle generator pattern are basically addressed by the digital pattern generator as a series of single pixels. The maximum number of addressable pixels depends on the digital pattern generator type (version 6: 3400 pixel; version 7: 65000 pixel). The pixels are written without settling time but with a fixed dwell time per pixel. The dwell time is called FBMS dwell time and can be specified in the menu **Extras➡Settings➡Patterning**. The recommended standard setting for the Raith Turnkey System is 0.002 ms with respect to electromagnetic deflection. The minimum value that can be set is dependent on the digital pattern generator type.

2.2 Beam Tracking Alignment

In its most basic operation, FBMS describes a mode where an exposure is executed by moving the stage while holding the beam stationary. Nevertheless, in order to obtain perfect patterning results it is very important to achieve a uniform beam on stage movement, which means no velocity fluctuations and position deviations perpendicular to the direction of motion. For this reason, the beam is not really fixed, but is in fact deflected by a fast beam tracking: Variations in the stage movement are continuously compensated by measuring the deviation between actual and nominal (ideal) values of the stage's position, and calculating a feedback correction-signal which is applied to the beam position. Accurate compensation by beam tracking requires the alignment of the deflection applied by the motor control unit to the stage movement. This alignment routine is called Beam Tracking Alignment (⇒ *Functional principle of Beam Tracking Alignment procedure* on page 11-10). It has to be applied whenever a new writefield size is selected for patterning in FBMS mode, just as for standard patterning runs.

2.3 FBMS Lines

FBMS lines are paths with a width of zero (⇒ *Adding FBMS paths* on page 5-44). In “line mode“, the beam is kept at a fixed position while the sample is moving laterally with constant speed. Like for single pixel lines, there is a strong correlation between beam current, speed, and the resulting line widths. The resulting dose is calculated by the following formula:

$$\text{line dose} = \frac{\text{beam current}}{\text{stage speed}}$$

FBMS lines with less than 20nm line width can be written in line mode.

2.4 FBMS Areas

FBMS lines are paths with a width larger than zero (⇒ *Adding FBMS paths* on page 5-44). In “area mode“, an increase in line width up to several 10 µm is achieved by a repetitive and continuous lateral deflection of the beam in the circle generator pattern (⇒ *FBMS beam deflection patterns* on page 14-7), which is defined such that the applied dose is constant over the designed line width. Moreover, its circular symmetry guarantees the same line width regardless of the direction of the sample motion. In addition to the beam deflection, the sample is simultaneously moving along the designed path as in line mode. The area dose can be calculated by following formula:

$$\text{area dose} = \frac{\text{beam current}}{\text{stage speed} \times \text{line width}}$$

2.5 Dynamic compensation for FBMS

A dynamic compensation for FBMS can be activated within the menu command **Extras➡Settings➡Motor Control IEEE-488** by setting the parameter **FBMS line prolongation**. This parameter defines the way in mm which the stage will travel with a blanked beam before the FBMS path is being exposed. This additional movement of the stage ensures that the final speed is reached before the FBMS patterning is started. When the value is set to zero, the prolongation is switched off. The reasons for the prolongation are two-fold. It enables the stage to reach the required speed prior to the patterning, and it also avoids patterning errors from stage vibrations when the stage is first set in motion. The prolongation becomes more important at higher stage speeds.

2.6 FBMS split path angle

A path can be defined along a number of nodes. The line prolongation will only be initiated at the start of each line of the path.

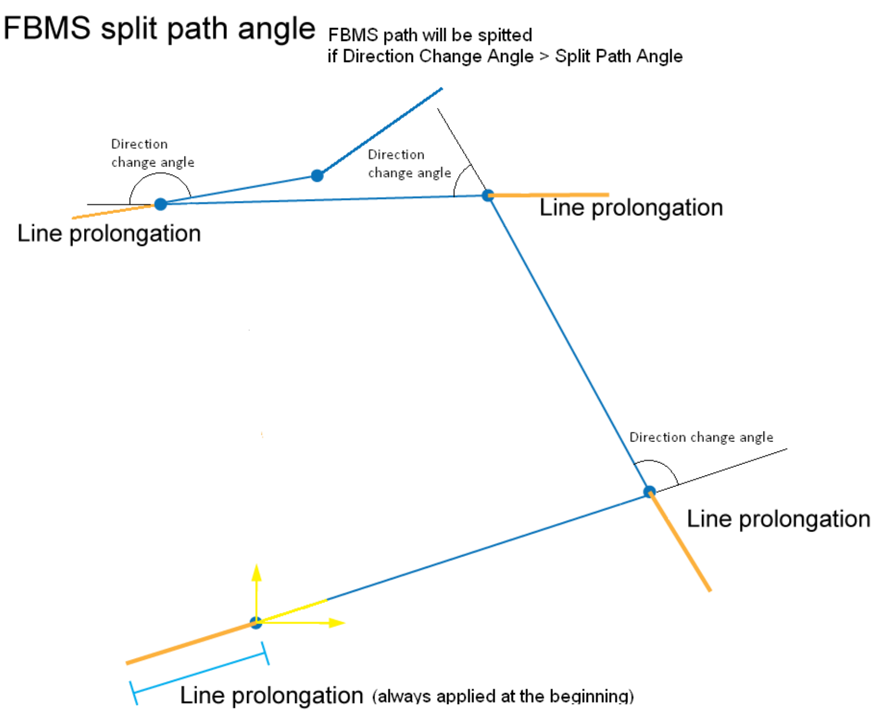


Figure 14-7: FBMS split path angle. The angles, denoted alpha, are defined between two patterning lines, shown in blue.

The angle alpha is defined between two lines. The angle will be defined in degrees and can vary from one split path segment to the next. After one segment of the line has been written, the line will be interrupted and the beam will be blanked. The stage will then describe a path to the next starting line with a blanked beam. Once the new angle has been reached, the line prolongation will be initiated again.

The user can define the minimum angle for the splitting. When this angle is reached or exceeded, the line prolongation will be initiated. Line prolongation is only needed when the path angle has reached a certain size.

When line prolongation is used frequently within a structure, it can lead to significantly longer overall patterning times, over the complete structure. If the timing is critical, the user may decide to set the line prolongation to zero.

The **Split Path Angle** can be defined in the **Module** status. The degree, by which the path splitting will be initiated, can be set by the user in **Extras➡Settings➡Motor Control: IEEE-488➡FBMS split path angle.**

Split Path Angle = 180^0 - Alpha

---

3 MBMS (option)

The optional periodixx feature is a unique lithography patterning mode and is based on Raith´s **Modulated Beam Moving Stage** (**MBMS**) technology. In MBMS patterning mode, the beam movement is defined such that the combination of repetitive patterning and synchronized continuous movement of the laser interferometer stage results in stitch-free, strip-shaped periodic structures (⇒ Figure 14-8). Hence, MBMS is the ideal choice to fabricate optical devices like elongated Bragg gratings, or photonic crystal waveguides, where even stitching errors of 10 nanometers and below can be critical for the device performance.

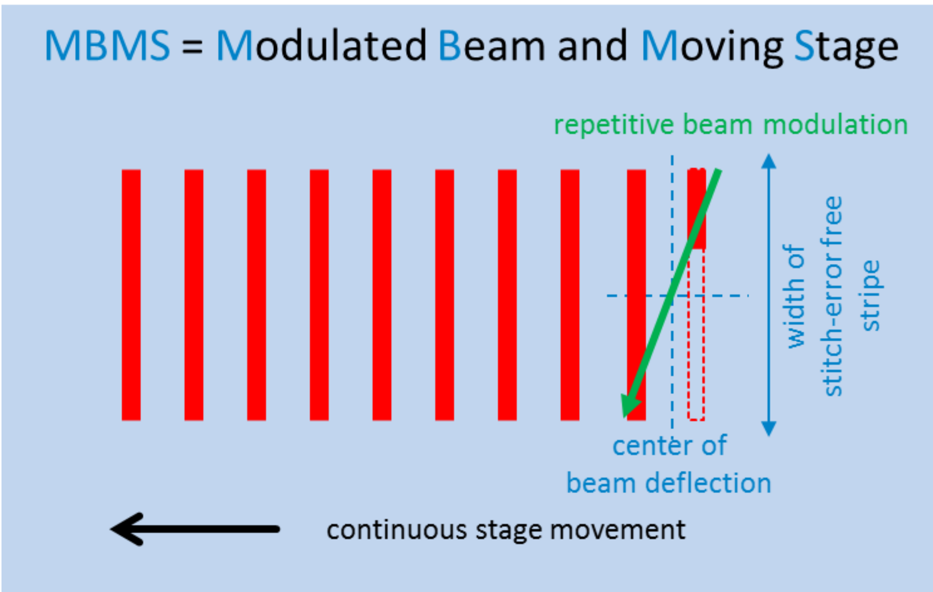


Figure 14-8: Principle of Modulated-Beam-Moving-Stage: Continuous stage movement with repetitive beam modulation, resulting in stitch-error free strips with a length of several mm or even cm.

Due to synchronization of beam and stage movement, also larger areas with periodic structures can be fabricated by stitching of adjacent strips showing superior stitching results. This is possible because 1) there is no stitching in direction of movement, and 2) stitching errors between strips are small due to the small deflections fields, e.g. deflection in direction of stage movement is typically smaller than the pitch itself.

The MBMS motor control unit allows excellent pitch control by means of stage velocity control in combination with beam tracking functionality. Variations in the stage movement are continuously compensated for by measuring the deviation between actual and ideal position of the stage and calculating a correction-signal, which is finally applied to the beam position.

3.1 Patterning parameters for MBMS elements

MBMS elements are defined as straight paths with a basic pattern that is repeated parallel and perpendicular to the path (⇒ *Creating MBMS elements (option)* on page 5-46). The base pattern can contain any GDSII element, e.g. polygons, lines, and dots. During exposure, a pixel pattern is calculated from the basic pattern, which includes a shift of each pixel that compensates the continuous stage movement (⇒ *Visualization* on page 14-53). There is only one dwell time available for the pixel from all elements in MBMS mode. Dose factors for area and line elements can be used within the basic pattern, but are transferred into scaling of the pixel distance for this reason. There is no dose scaling for single dots in the base pattern. Within the **Patterning Parameters** window, MBMS elements can be activated or deactivated, and area and line step size and the dwell time can be set. In order to find suitable parameters, you may use the MBMS tab in the Calculator window (⇒ *MBMS Parameter Calculation* on page 14-26).

In MBMS patterning mode, settling times are generated by adding blanked pixels. A dedicated settling time factor for MBMS elements can be separately defined, and can reduced compared to conventional exposures and with respect to individual layouts in order to minimize the overall patterning time (⇒ *Settling Time* on page 14-38).

3.2 Patterning properties in the Positionlist

Just as for other elements, patterning parameters for MBMS elements can be set for individual Positionlist tasks by opening the **Patterning Properties** dialog via the context menu command **Properties** of the **Positionlist**. (⇒ *Patterning via Positionlist* on page 14-15). In addition to parameters defined in the **Patterning Parameters** window, the Patterning Properties allow to set the **Pitch Scaling Factor** (the factor of 1 means no change in scaling). This factor is responsible for the scaling the pitch for MBMS elements inside individual positionlist entry by changing the stage speed during the patterning process. The reason is the limited absolute pitch accuracy due to local variation of the laser interferometer stage accuracy, which is in the nm range, but typically reproducible. This allows sub-nm control of the pitch in a limited area by applying a previously determined (via test pattern) Pitch Scaling Factor for the exposure with respect to the stage position.

In MBMS patterning mode, settling times are generated by adding blanked pixels. A dedicated settling time factor for MBMS elements can be separately defined, and can reduced compared to conventional exposures and with respect to individual layouts in order to minimize the overall patterning time (⇒ *Settling Time* on page 14-38).

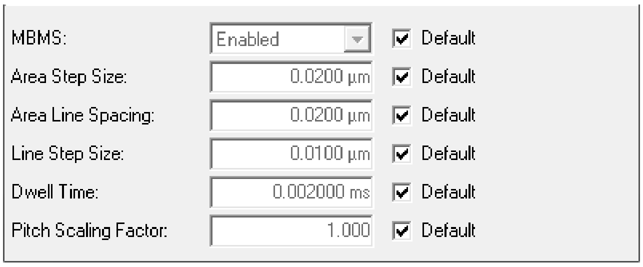


Figure 14-9: **Patterning Properties** of **MBMS** elements in the Positionlist.

4 Setting the Module Settings for single field patterning

The **Single Field** mode can be enabled via the main menu **Extras**➡**Settings**➡**Patterning**. Double click to view all parameter settings.

Single Field is a patterning mode developed for users who do not have the laser interferometer stage installed on their system. Without this option, it is not possible to stitch several writing fields. The user has therefore to use the **Single Field** option for the patterning procedure.
When the option is selected, only one writefield can be written. Only a GDSII structure, either smaller than, or equal to, one writefield is accepted.

When the user selects a variable, the software will show a description of this variable at the bottom of the window.

Double clicking on the variable **Single Field** will open a new dialog, in which the user can switch the Single Field mode On or Off.

Please read the section **Module Setting** in the chapter **User Interface** for a more detailed description, (⇒ *Module settings* on page 4-31).

4.1 Stitch wait time

In between writefields, a stitch wait time is implemented to pause until the stage is not longer vibrating after the last movement. The user can adjust the value for the stitch wait time.

In **Extras➡Settings➡Patterning**, set the variable **Stitch wait time** to the number of milliseconds that should elapse, following stage movement to the new stitch field, before the patterning in this stitch field is initiated.

B Tasks

1 Patterning via Positionlist

The **Patterning** icon in the control bar opens the corresponding dialogs for the definition of the default parameters that are used during a patterning procedure, such as step sizes, dwell times and selection of patterning modes.

The patterning procedure itself is executed via the **Positionlist**. The advantage is that within a Positionlist it is possible to define additional parameters, such as the location on the sample at which a structure should be written and the working area. A Positionlist can also store more then one patterning procedure, enabling batch execution. For each patterning procedure, different patterning parameters can be assigned. Within the Positionlist, the user can overwrite the default patterning parameters as defined in the **Patterning Parameter** dialog, to adapt them to special applications or to find new optimized parameter settings.

It is recommended to read the chapter **Positionlist** as well as the chapter **Scan Manager**, to familiarize yourself with the way that procedures from the Scan Manager can be inserted via drag and drop into the Positionlist, as well as how the parameters can then be changed within the Positionlist via the **Properties** dialog. The chapters also explain how parts, or all of the Positionlist can be executed, (⇒ *Positionlist* on page 9-1, ⇒ *Scan Manager* on page 11-1).

- Execute the patterning procedure via the **Positionlist** whenever multiple structures need to be written or when repeated structures with the same, or different parameters should be written. The Positionlist enables the execution of batch patterning procedures, which will be fully automated, the stage driving to each new location in turn.
- In order to generate an automated patterning sequence of one or more structures at different locations on the sample, firstly load the **GDSII database** containing the structures.
- Open a new Positionlist via the menu **File**➡**New Positionlist**.
- Within the GDSII database, select the structure that is to be written. Drag and drop it at any location within the Positionlist window. Using this technique, the structure will be added as a line entry to that Positionlist. Each line of the Positionlist contains a task.
- To write multiple GDSII structures, repeat this procedure.
- Alternatively, you can use a **Positionlist Filter**, such as **Matrix Copy**, to generate additional patterning tasks.

When using the drag and drop function for insertion of a patterning procedure into the Positionlist, the software stores the following settings within the Positionlist:
- the current writefield size
- the current UV position
- the current GDSII working area
- if patterning of lines, curved elements and dots is switched on or off.

- ▶ The user can now redefine these parameters within the Positionlist for every task, by opening the **Patterning Properties** dialog via the context menu command **Properties** of the Positionlist. This enables the editing of the patterning parameters. The patterning of the GDSII structure will be performed by applying these parameters.
- ▶ For a quick graphical inspection of selected layers and the working areas, double click on the line entry in the Positionlist. This will automatically open the **GDSII Viewer** and displays the selected structure, layers and working areas.
- ▶ It is possible to execute either all or part of the Positionlist. The complete patterning list can be stored and recalled for use or modification at any time.

When the user selects the procedure **Automatic with Linescans** from the **Scan Manager** and inserts it into the **GDSII Editor** via drag and drop, this will then be patterned as **Layer 61**. This is the same layer as used for the automatic mark scans.

The procedures **Automatic with Images** will be patterned as **Layer 62** and **Manual with Images** will be patterned as **Layer 63**. This is the same layer as used for manual mark scans.

When the user is selecting the layers to be patterned in the **GDSII Editor**, the appropriate layers have to be either activated or deactivated.

GDSII writefield mark scans are created within the **Scan Manager**, taking into account the **GDSII mark scans**, either manual or automatic procedures can be called up. These procedures may also include linescans or images. The parameters can be edited within the Scan Manager.

The procedures can then be inserted into the **Positionlist** via drag and drop.

C Software Reference

1 Patterning dialog

Whatever pattern you have in mind, it is very important to have some background information about how the GDSII design is transferred into a pattern and how the various parameters affect the patterning.

The aim of this section is to provide information about patterning parameters. To understand this in the complete context of instrument operation, it is useful to remember that the following steps have to be executed before writing a pattern.

- Using the **GDSII Editor** and associated elements, create design structures.
- Set up the column, i.e. select the acceleration voltage, the beam current, and the writefield size.
- Define design parameters, i.e. the layers, the working area, and their location on the sample.
- Define the **Patterning Parameters**, i.e. Step Sizes, Dwell Times, Settling Times, and selection of different modes.
- Start the patterning.

Within this reference manual we do not describe how to set up patterning in a step-by-step way. To learn more about this, please consult the operation manual.

1.1 Tool bar

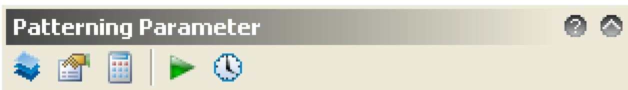


Figure 14-10: Tool bar of **Patterning Parameter**

Control element	Function
	If several layers have been designed with the GDSII Editor, the user can select one or more layers to be written. Click this button and a dialog appears with a listing of all layers. Select the layers to be written. Deselect layers by an additional mouse click.
	The Enhanced Parameters Settings allow the user to define more advanced values for the patterning.

If the user does not initiate the patterning via the Positionlist, but would prefer to initiate the patterning at the current position, the **Play** button will start the patterning. Only the selected structure will be written.
This procedure is not required if a Positionlist is used for the patterning, since the Positionlist includes all of the required information about the structures and layers.
Start patterning and **Patterning parameter calculation** buttons can be enabled via the menu **Extras**➡**Settings**➡**Patterning** by setting the parameter **Enable Patterning Controls** to **ON**.

Control element	Function
	The software will calculate the remaining patterning parameters from the values entered by the user.
	The selected patterning will be initiated.
	This will calculate the overall patterning time for the selected pattern.

2 Working area

The working area shows the patterning parameters for the selected write-field size. The parameters can be entered for **Areas**, **Lines**, **Curved elements**, **Dots** and **FBMS Areas** and **Lines**.

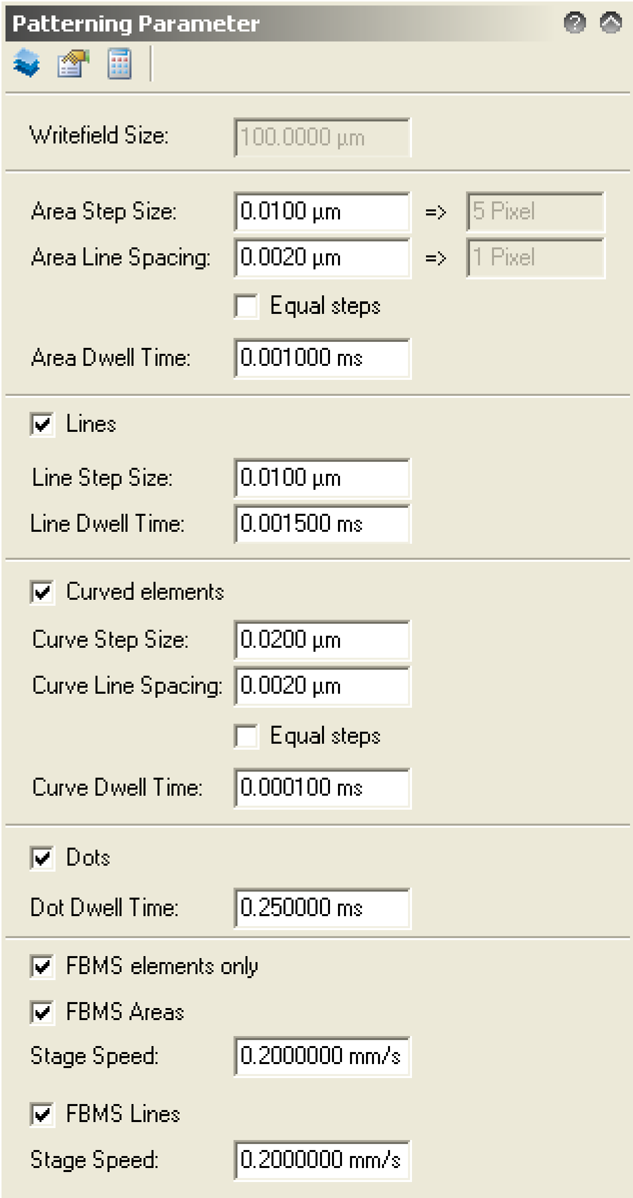


Figure 14-11: Patterning Parameters

Control element	Function
Writefield Size	Information is displayed about the currently selected writefield. A square writefield is assumed, in which width is equal to height. The following patterning parameters can be edited and calculated within the Patterning. An optional setting within a Positionlist is possible.

Control element	Function
Area subset:	
Area Step Size	The **Step Size** for the spacing within each line can be defined in µm. This parameter defines the distance between patterning dots. It must be an integer multiple of the **Basic step size**. When a step size value in µm is entered, the related step size in Pixels will be calculated.
Area Line Spacing	The **Line Spacing** between each line can be defined in µm. When a step size value in µm is entered, the related step size in Pixels will be calculated.
Equal steps	When this checkbox is checked, the **Step Size** and the **Line Spacing** will be equal.
Area Dwell Time	The **Dwell Time** for the area can be entered by the user. This parameter defines the time that the beam waits at each step when exposing an area.

The minimum dwell time depends on the hardware and is automatically set. The user will not be allowed to set a smaller value than the minimum dwell time. Dwell times can be selected in steps of 1 ns for the Raith High Speed Pattern Generator and in 100 ns steps for the Raith Quantum board. If the user input does not fit exactly into this dwell time raster, the Raith NANOSUITE software automatically selects the next larger value.

Control element	Function
Lines subset:	
Lines	Select this option to enable the patterning of single pixel lines. After enabling, the parameters **Step Size** and **Dwell Time** for these lines are available.
Line Step Size	The **Step Size** within each line can be defined in µm. This parameter defines the distance between patterning dots. It must be an integer multiple of the **Basic step size**.
Line Dwell Time	The **Dwell Time** in ms for the line may be entered by the user. This parameter defines the time that the beam waits at each step when writing a line.

Curved elements subset:

Control element	Function
Curved elements	Select this option to enable the patterning of curved elements, such as circles, ellipses, arcs or segments. After enabling, the parameters **Step Size** and **Dwell Time** for **Curved elements** are available.
Curve Step Size	The **Step Size** within each curve can be defined in µm. This parameter defines the distance between patterning dots in µm. It has to be an integer multiple of the **Basic step size**.
Curve Line Spacing	The line spacing between each curved line can be defined in µm.
Equal steps	When the checkbox for equal steps is checked, the **Step Size** and the **Line Spacing** will be equal.
Curve Dwell Time	The **Dwell Time** for the curve can be entered by the user. This parameter defines the time that the beam waits at each point when writing a curved element.
Dot subset:	
Dots	Select this option to enable the patterning of dots. After enabling, the parameter **Dwell Time** for dots is available.
Dot Dwell Time	The **Dwell Time** in ms for the dot can be entered by the user.
FBMS subset:	
FBMS elements only	Select this option to enable only the patterning of FBMS elements.

Do not check this checkbox, if you wish to write both standard elements and FBMS elements.

Control element	Function
FBMS Area stage Speed	The stage speed in mm/s for FBMS Areas can be entered by the user.
FBMS Lines Stage Speed	The stage speed in mm/s for FBMS Lines can be entered by the user.

The **FBMS elements only** checkbox is also useful, even when a structure only consists of FBMS elements, since the instrument software will be more time-efficient when the GDSII structures are being uploaded. The software will normally upload standard structures first, followed by FBMS elements. The software rasters the complete writefield to search for normal elements, which is a waste of time if the user knows that there are no normal elements. FBMS elements are usually larger structures, over a relatively large area, so the complete working area could be quite large. A search for non-existent normal elements could therefore be rather time consuming. If the **FBMS elements only** checkbox is checked, the patterning can start quickly, since reading in of the FBMS elements is quite fast.

3 Patterning Parameter subdialogs

The following subdialogs can be opened to edit further parameters

The user may call up the **Calculation** by pressing the button.

3.1 Patterning Parameter calculation

The following sections explain the Patterning Parameter calculation.

3.1.1 Formula for Patterning Parameter calculation

The **Patterning Parameter Calculator** will calculate the patterning for **Areas**, **Curved elements**, **Lines**, **Dots** and **FBMS** structures. The user inputs the beam current to be used.

The user enters the value for the **Area Dose** or **Line Dose.** This value depends on the resist used. The user would normally know the dose values for each resist. Beam current can be measured directly from the instrument via the Beam Current module, or typed into the **Beam Current** input field.
The formula for the calculation is shown and if the numbers are displayed in red, an error in the calculation has occurred and the user can not click **OK** to save the parameter.

The four input parameters for area patterning show four buttons next to the related input field. The user may enter the values in any four of these five fields. The fifth parameter will be calculated such that all five parameters fulfil the equation:

$$\mathrm{AreaDose} = \frac{\mathrm{BeamCurrent} \times \mathrm{AreaDwellTime}}{\mathrm{StepSize} \times \mathrm{LineSpacing}}$$

To perform the calculation, choose the button behind it. Now that these five parameters fulfil the given equation, the corresponding title of the tab changes from red to black.

The parameters for lines and dots may be calculated in the same way. In this case, the following equations are used to calculate the parameters for line patterning.

$$\mathrm{LineDose} = \frac{\mathrm{BeamCurrent} \times \mathrm{LineDwellTime}}{\mathrm{LineStepSize}}$$

and for dots:

$$\mathrm{DotDose} = \mathrm{BeamCurrent} \times \mathrm{DotDwellTime}$$

If all three parameter sets for **Area**, **Curved elements**, **Line** and **Dot** patterning fulfil their corresponding equations, the **OK** button is enabled and the values will be written back into the **Patterning Parameters** window.

3.1.2 FBMS Patterning Parameter formula for Calculation Width

The **Calculation Width** is entered by the user and corresponds to the minimum (narrowest) possible expected width of the FBMS path. The Calculation Width is used in the formula:

$$\mathrm{AreaDose} = \frac{\mathrm{BeamCurrent}}{\mathrm{StageSpeed} \times \mathrm{CalculationWidth}}$$

As the formula shows, the thinnest lines are written at the highest stage speeds.

The user should always enter a calculation width value as close as possible to the thinnest expected line in the GDSII pattern, in order to estimate accurately how fast the stage must move when writing the thinnest lines in the structure.

The **Calculation Width** is simply a reference value, which relates width, stage speed and dose. Stage speed will be automatically adjusted by the software depending on the patterning being carried out. For example, if the user has entered a Calculation Width of 2 µm, but the current layout width is 4 µm, the stage speed used for the 4 µm layout will be half the reference **Stage Speed**, which is displayed in the same window.

If the user enters a **Calculation Width** that is too large, the stage speed actually used will be in proportion to that high value. If another FBMS element within the GDSII database has a much narrower design width than the calculation width, the stage speed must be adjusted to a much higher speed, which may exceed the stage speed limit, resulting in incorrect patterning of the structure.

If the Calculation Width entered is an accurate reflection of the thinnest width expected in the GDSII structure and if the highest (worst case scenario) stage speed is calculated to exceed the patterning speed limit, the software will warn the user that the stage speed may exceed its limit and will prompt the user to change the settings accordingly.

If the user enters a value which is lower than necessary, the stage speed will be too low and the optimum system performance will not be achieved.

3.1.3 Patterning Parameter Calculation dialog

Choose the **Calculation** button from the tool bar to calculate the patterning parameters for the doses of **Areas**, **Curved elements**, **Lines**, **Dots**, **FBMS Areas**, **FBMS Lines**, and **MBMS** elements. Please note that the parameters for Lines, Curved elements, Dots, FBMS and MBMS elements are only available if the corresponding option within the patterning is selected. All parameters will be taken from the **Patterning Properties** window and they will be written back into it, when choosing the **OK** button.

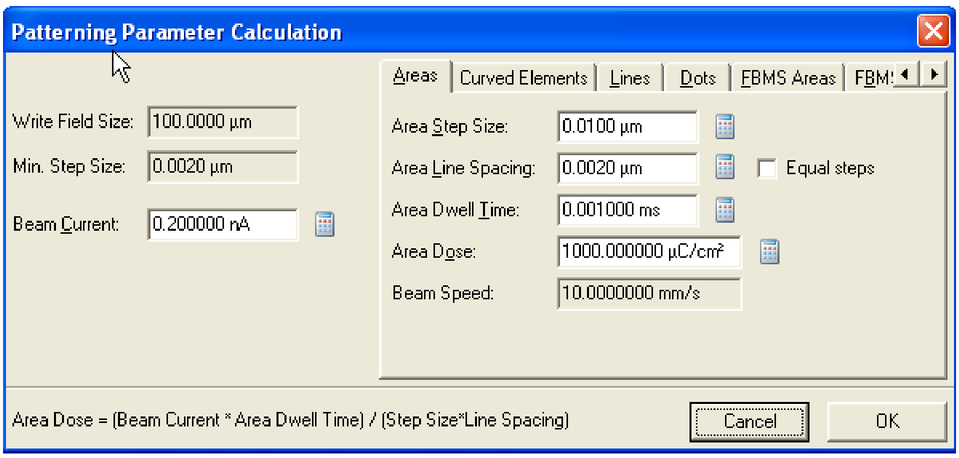


Figure 14-12: Patterning Parameter Calculation

On the left hand side, the user will see the following control elements.

Control element	Function
Writefield Size	This value is given for information purposes only.
Min Step Size	This value is given for information purposes only.
Beam Current	For most systems, a software module is available to read in the beam current from a current meter. Its value is then automatically displayed here. If such a module is not available, use an external current meter to measure the current and enter its value in nA. The same beam current is used to calculate the patterning parameters for **Areas, Lines** and **Dots**.

On the right hand side, the user can select one of the following tabs:

Areas, Curved elements, Lines, Dots, FBMS Areas and **FBMS Lines**. The corresponding entered parameters will be displayed, such as **Step Size**, **Line Spacing, Dwell Time, Area Dose** and **Beam Speed,** depending on the tab selected. The **FBMS elements** tab will be explained separately, since its parameter set differs from the other tabs.

3.1.4 FBMS Calculation

In order to find the most suitable parameters, you may select **Calculator** within the **Patterning** window for **FBMS Areas** and **FBMS Lines**. Here the correct Stage Speed/ Beam Current combination can be evaluated and selected for a given resist sensitivity. In the **FBMS Area Calculation** window, the parameters **Stage Speed**, **Calculation Width** and **Area Dose** are displayed.

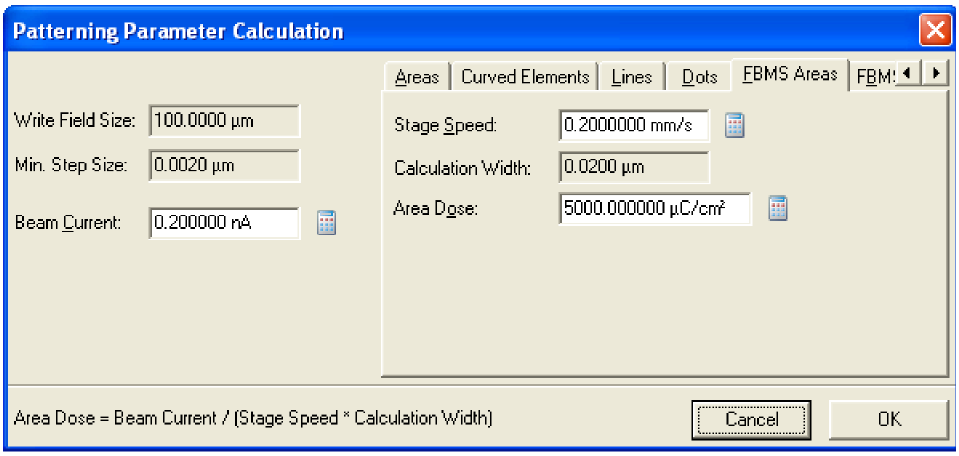


Figure 14-13: **Patterning Parameter Calculation** dialog, FBMS Area tab

Control element	Function
FBMS Area	The Area mode calculator proposes a suitable stage speed for all curved paths which have a non zero width and which will therefore be exposed in beam shape mode. Input parameters are the **Beam Current** and the **Area Dose** as described in standard patterning.
FBMS Line	The calculator scheme will be used to control curved paths with zero width to be exposed in beam spot mode. Important input parameters are the Beam Current and the **Line Dose**, as already described in the normal line patterning mode. Typically, this window is used to calculate the optimum stage speed for patterning. The results are transferred automatically to the **Patterning** window. Using the input dose value and the beam current, the FBMS calculator calculates the speed for the stage that will deliver the required dose.

An experienced user may wish to input the stage speed directly in some instances for special applications, using the **Stage Speed** input field in the **Patterning Parameter Calculation** window.

3.1.5 MBMS Parameter Calculation

Select the **MBMS** tab in the **Patterning Parameter Calculation** window to calculate the dwell time which is used for MBMS elements. Note that there is only one dwell time which is used for all area, line, and dot elements that are defined in the base element (⇒ *Patterning parameters for MBMS elements* on page 14-12). Select **Use as Base** option to use either Area or Line settings to calculate the dwell time. Eventually use the other selection to calculate a step size that results in the required dose for the other kind of elements at the given dwell time.

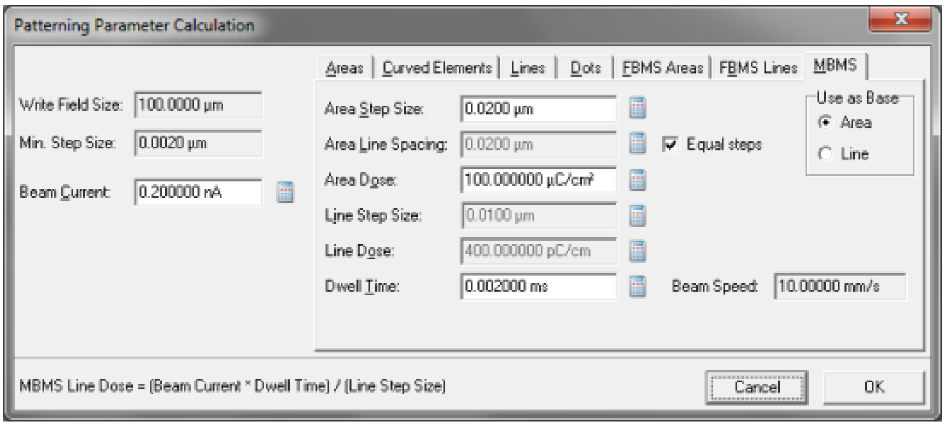


Figure 14-14: **Patterning Parameter Calculation** dialog, MBMS tab

3.2 Estimating Patterning Time

The user may call up the **Patterning Times Calculation** by pressing the button.

In the **Patterning Parameter** window, click the **Times** button to view information about the expected total patterning time. An **Estimated Patterning Times** dialog box shows the times for individual activities and the total time for the overall process for all selected layers and includes the time required for the execution of the scripts and macros in the last but one line.

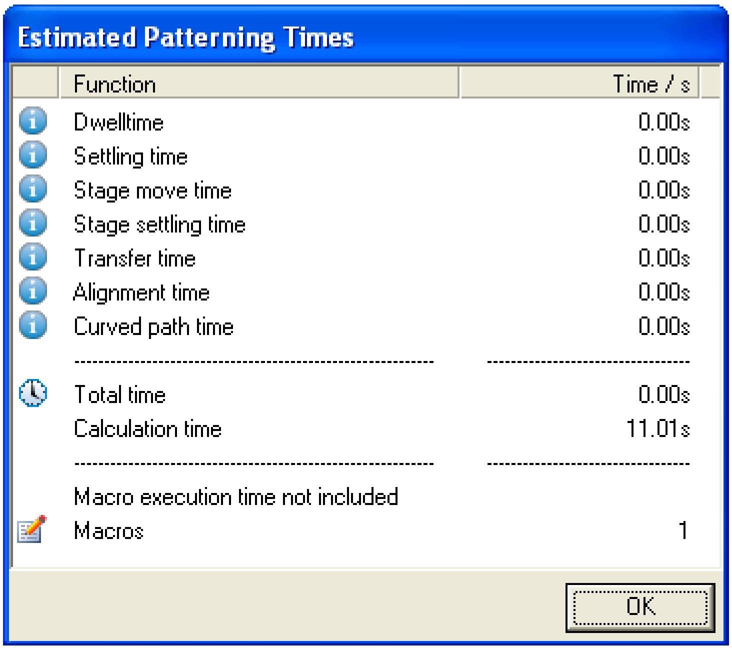


Figure 14-15: **Estimated Patterning Times**

3.3 Enhanced Patterning Parameters

Choose the **Enhanced Parameter Settings** button from the tool bar to edit the Enhanced Patterning parameters. A **Patterning Details** window will open.

3.3.1 Loops

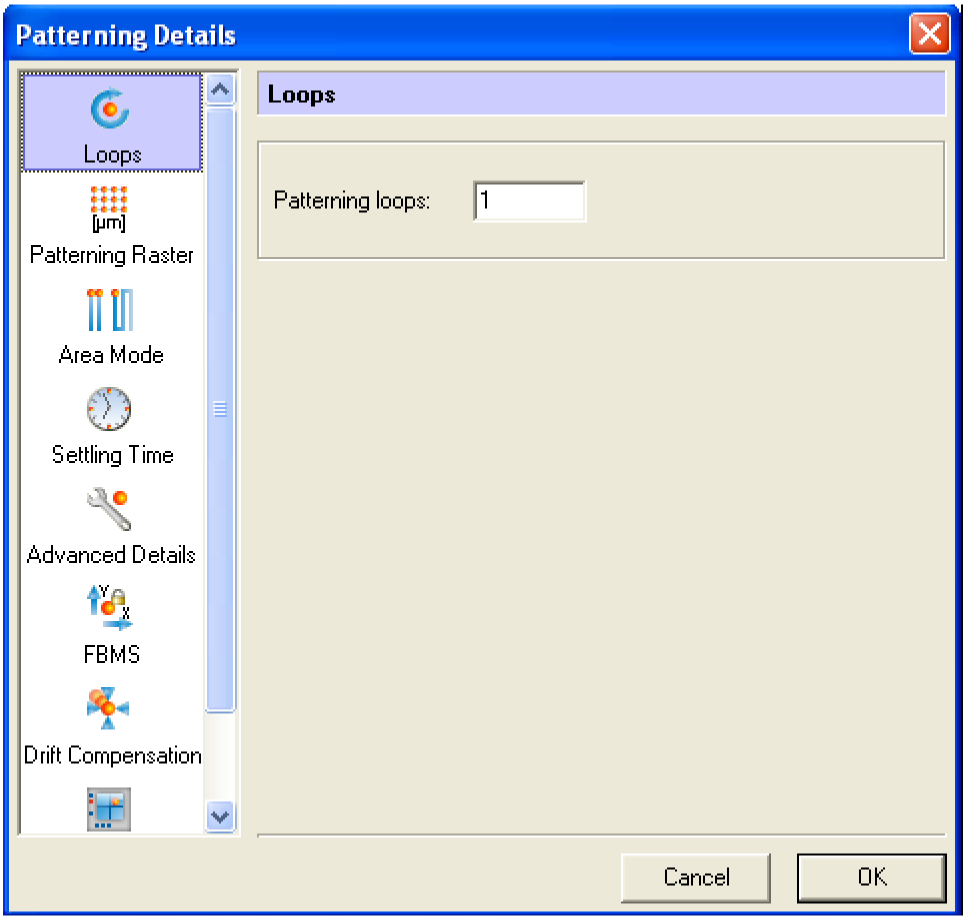


Figure 14-16: **Patterning Details** dialog, **Loops** tab

Control element	Function
Patterning Loops	The number entered within this text box defines how many times the complete structure will be written. Usually, a pattern is written just once, but it may be necessary, e.g. to avoid charging effects, to perform the patterning in several loops with lower dose. Selecting **1** for the **Loop** value will cause the patterning to be performed only once. If this parameter is set to zero, the patterning will start an endless loop until it is stopped within the corresponding window.

3.3.2 Patterning Raster

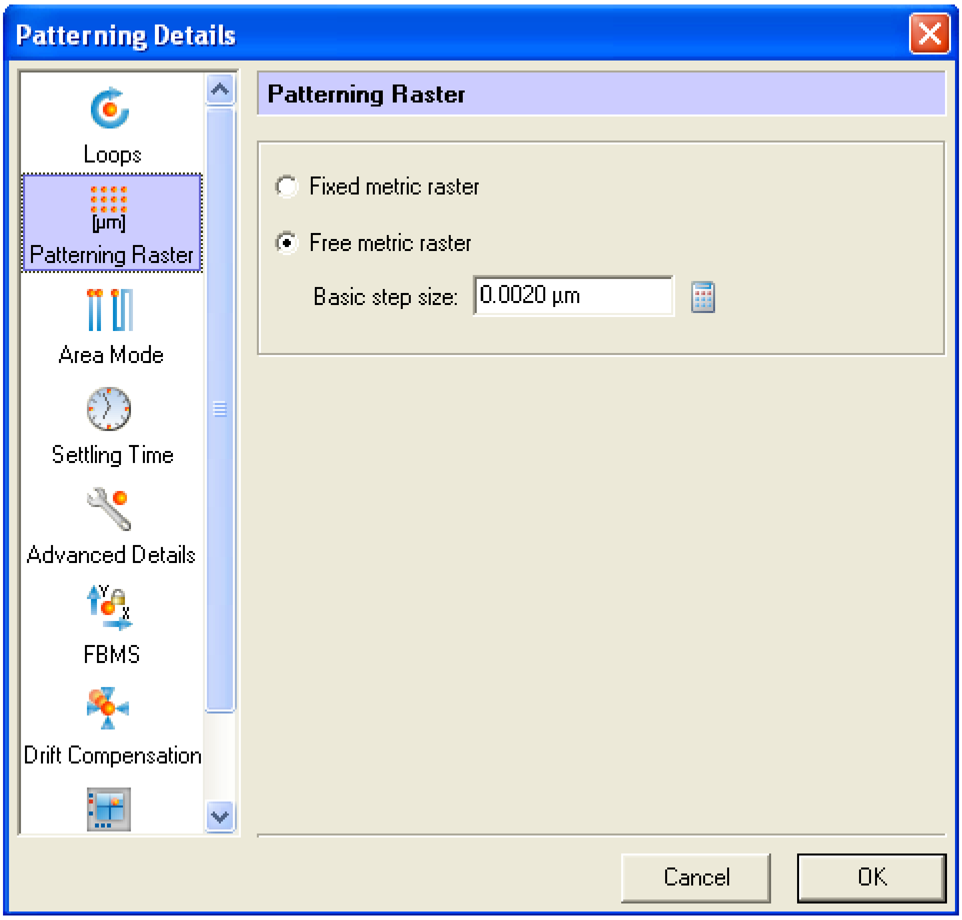


Figure 14-17: **Patterning Details** dialog, **Patterning Raster** tab

Control element	Function
Patterning Raster	**Fixed metric raster** The user may select either fixed metric raster, in which the software will calculate the raster parameters, or may select user definable raster, which is particularly useful when exposing small features. **Free metric raster** For example, if a rectangle with a 25 nm dimension is to be patterned, the user may select a **Basic step size** for the raster of 5 nm. This means that an integer number of steps corresponds exactly to the required dimension.

The patterning raster determines the minimum distance between two adjacent dots. This minimum distance is called the **Basic step size** and depends on the currently used writefield size. Select **Fixed metric raster** to automatically use the smallest **Basic step size**. Use **Free metric raster** mode to define the **Basic step size** manually. It can be any value greater than the **Basic step size** calculated in **Fixed Metric** mode;

writefield size/ number of pixels

For example, the step size for a 100 µm writefield is 1.6 nm and 0.8 nm for a 50 µm writefield. The **Basic step size** can be increased in 0.1 nm steps. Sometimes it is useful to work with **Free metric raster** to achieve optimal patterning results.

If several elements with dimension of 5 nm are to be written, it is best to define the **Basic step size** as 2.5 nm. This guarantees that all elements get the same number of dots. If one would use, for example, 2 nm, it may happen that due to rounding, one element would contain 2 lines and another one would contain 3 lines.

3.3.3 Area Mode

The filling of areas is defined here. In **Line** mode, the beam writes each line by always driving the beam from one direction. The best accuracy in patterning is achieved by this method, but the patterning takes longer, due to larger beam settling times and the need for dynamic compensation at the start of each line.

When the blanked beam has been moved to the beginning of a new line within a structure and a settling time has been allowed, there is a short distance over which the speed of movement of the beam ramps up from rest to the normal speed. This would lead to non-uniform patterning at the start of each line. **Dynamic Compensation**, explained later in **Advanced Details**, eliminates this potential error, but adds to the total patterning time.

In **Meander** mode, the beam is moved in alternating directions from line to line. This mode results in shorter patterning times. When using this mode, the dynamic compensation must be switched off.

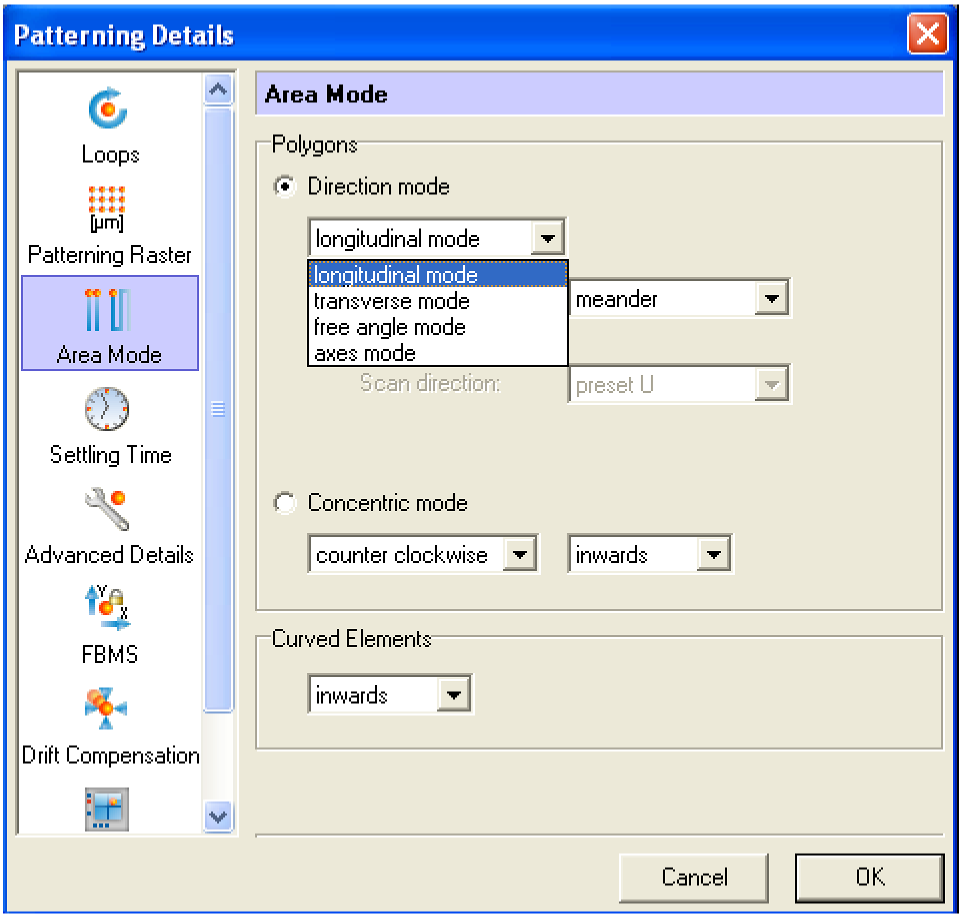


Figure 14-18: **Patterning Details** dialog, **Area Mode** tab

Control element	Function
Polygons	The user can define the following parameters: Direction mode: - **longitudinal mode** - **transverse mode** - **free angle mode** - **axes mode** Vector mode: - **line** or **meander** - **concentric mode**: - **clockwise** or **counter clockwise** - **inwards** or **outwards**
Curved elements	**inwards** or **outwards**

Longitudinal/Transverse mode

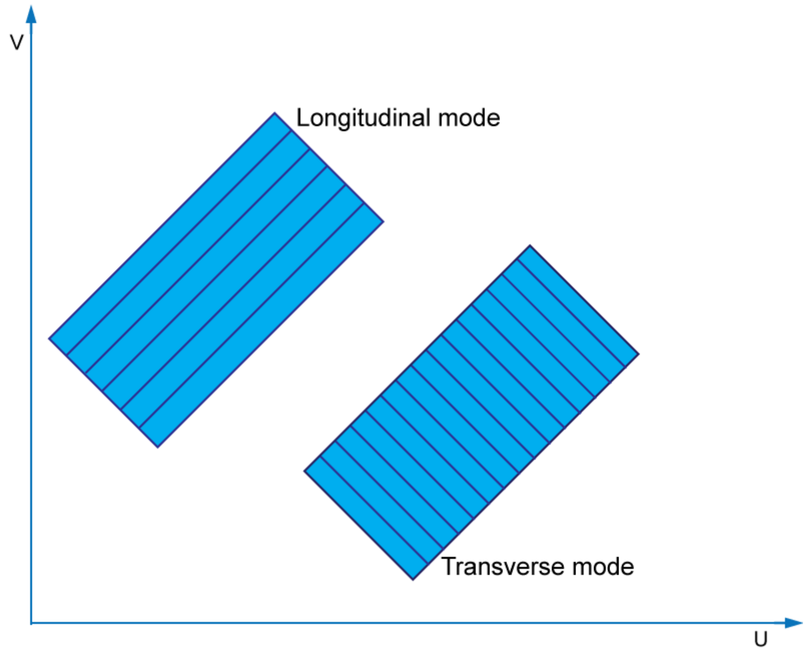


Figure 14-19: Longitudinal / Transverse mode

Mode	Function
Longitudinal mode	Lines will be written parallel to the longest edge of a feature, separated by the defined **Line Spacing**.
Transverse mode	The direction of the line will be perpendicular to the longest edge and all other lines will be parallel to this first line, separated by the defined line spacing.

Free Angle mode

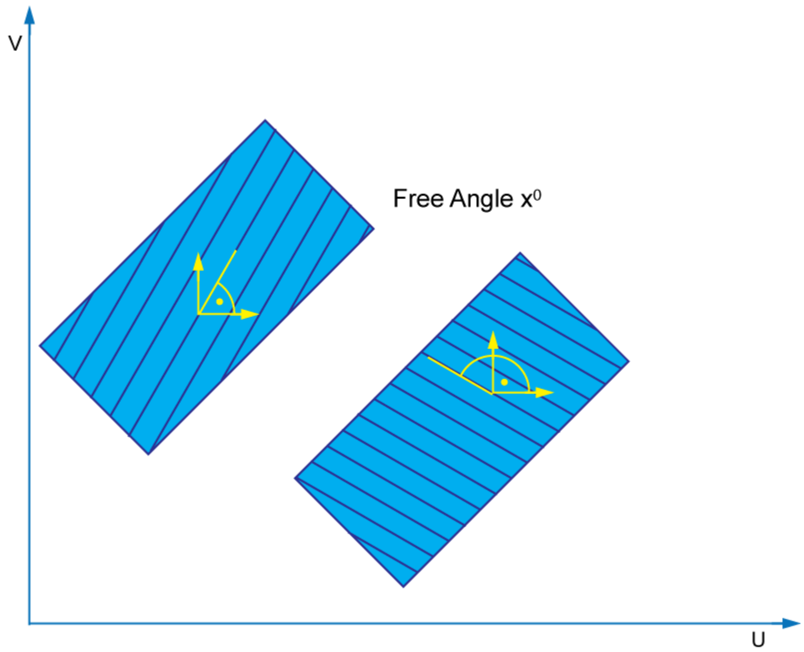


Figure 14-20: **Free Angle** mode

Mode	Function
Free angle mode	The user may select an angle, relative to the U direction, for the direction of the patterning lines. All lines will be described at this defined angle.

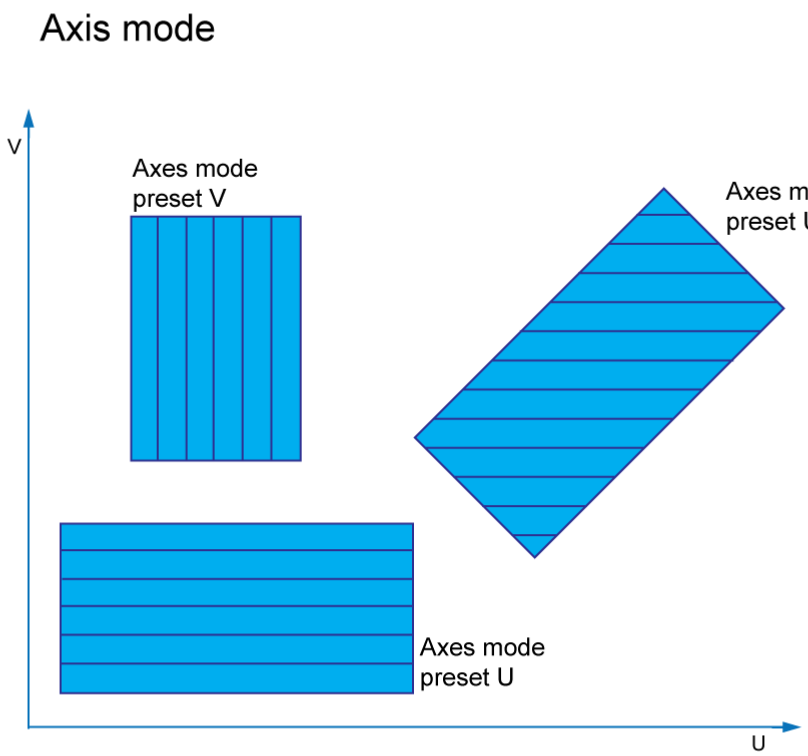


Figure 14-21: **Axis mode**

Mode	Function
Axis mode	Areas are filled by lines parallel to either the U direction or the V direction. If the structure has no edge parallel to either U or V, it is recommended to use the longitudinal or transverse mode for the patterning, resulting in a structure with a cleaner, or better defined, edge. Longitudinal patterning is generally faster, especially in meander mode, since there are fewer lines in total, of greater mean length and there are fewer start/stop events throughout the patterning procedure.

Area Line and Area Meander

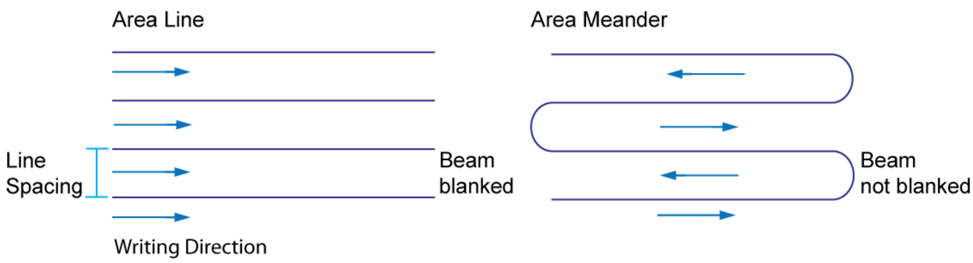


Figure 14-22: **Area line** and **Area meander**

Control element	Function
Scan Direction	When filling an area, the beam writes the area line by line. The direction of these lines is called the scan direction. In **Axis** mode, the user may pre-define the U and V directions.
Vector mode	**Line** The raster may be described as a series of lines or as a meander pattern. If Line mode is selected, the patterning takes the form of a series of single lines, of defined line spacing, each line starting at the same end as the previously written line. The beam is blanked during repositioning between the end of one line and the start of the new line. **Meander** Alternatively, if a Meander pattern is selected, after the first line is completed, the beam moves perpendicular to the line direction, by a distance equal to the line spacing. The next line is written in the opposite direction. At the end of each line, the beam position is incremented by the line spacing, perpendicular to the line direction, and the next line is written in the opposite direction to the previous one. In this mode, the beam does not need to be blanked between each line.

Concentric mode

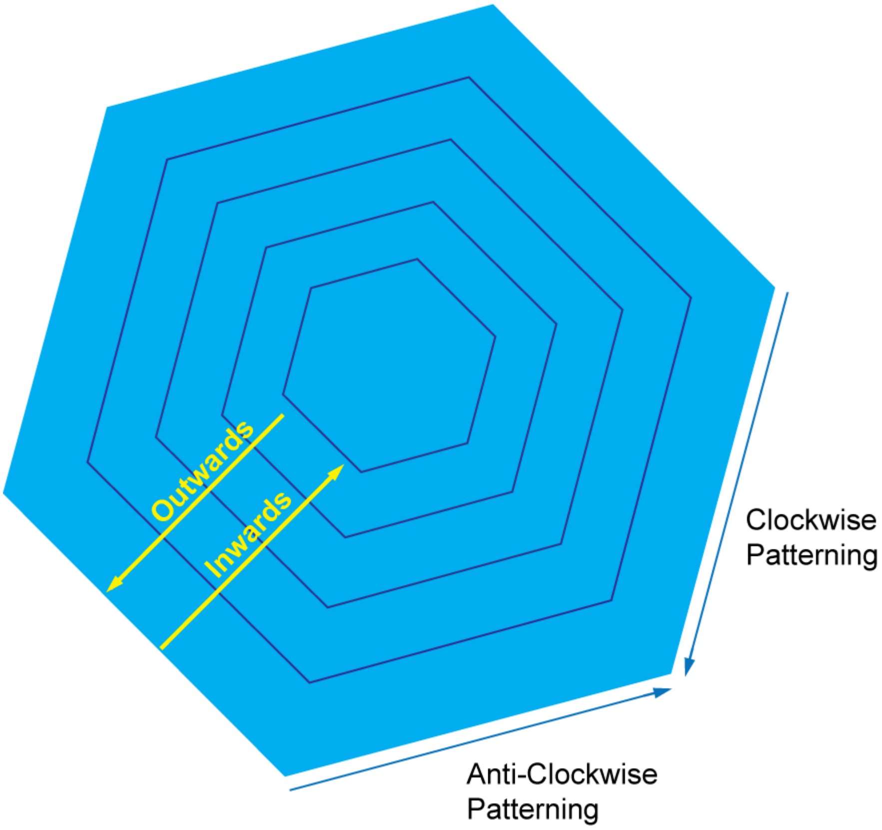


Figure 14-23: Concentric mode

Control element	Function
Concentric Mode	Any structure form may be patterned in this mode. The outer shape of the feature is defined as a series of straight lines between nodes, or defined points, forming a complex polygon. The whole area of the feature is patterned using smaller and smaller analogs of that shape. The patterning may be from the inside, moving outwards, or from the outer edge moving in. In addition, the patterning may be either clockwise or counter-clockwise around the feature.

Control element	Function
Curved elements	In addition to the patterning of straight lines, rectangular and other straight-sided element shapes, curved lines and patterns with curved facets may also be written. This allows the patterning of full circles, circle segments, bisected circles, complete circumference lines and arcs. A curved path may consist of any number of arcs of differing radius, up to a maximum of 1024 nodes (definition points). During the patterning, additional points will be calculated and placed in between arcs, to achieve the exact shape. The width of a path may also be defined. The minimum width is achieved by defining the **Width** parameter as zero, which leads to patterning in spot mode. Other widths may be defined in microns.

Curved paths may be created manually using the **GDSII Editor,** setting individual points on the curved path and editing the shape retrospectively.

It is important to calculate the doses for the patterning of circles differently to the calculation used for rectangular areas.

The line spacing for the area defines the distance between each line of dots. When a curved element is written, the first line defined is the outer curved edge. Subsequent lines are described inside this first curved line. Each written line is part of the outer circumference of a circle, each circle having the same center point. The line spacing is equivalent to the difference between the radii of each adjacent circle. The shape may be written either from the outer curved edge moving inwards, or from the inside, moving outwards towards the curved edge.

For all area elements, the default values for all patterning modes are defined in the **Patterning Details** dialog. To edit these values individually for a single GDSII element or group, go to the **Patterning Attributes** dialog.

3.3.4 Settling Time

The **Settling Time** is the time allowed for the beam to become stationary after a beam deflection to a new location in a write field. The optimum settling time varies according to the distance that the beam is deflected, i.e. the farther the distance deflected, the longer the corresponding Settling Time. A small beam deflection within a structure will require a shorter settling time than a beam deflection over a larger distance between structures.

The **Settling Time** can be set manually or calculated automatically.

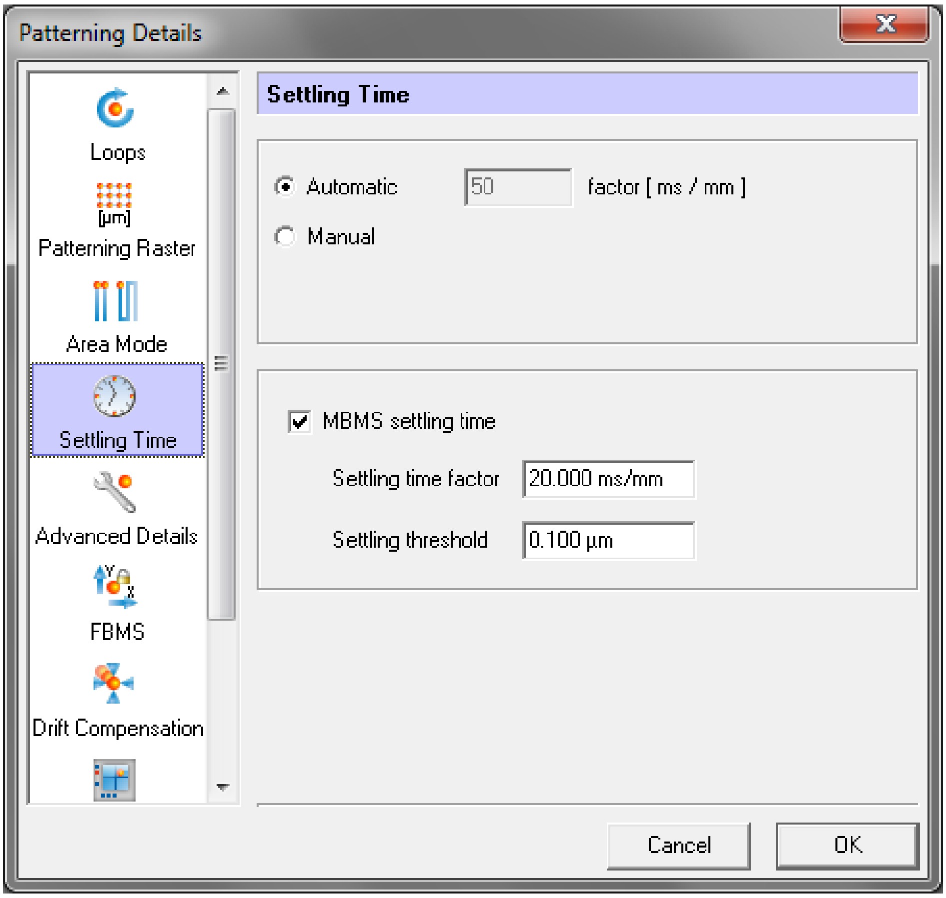


Figure 14-24: **Patterning Details** dialog, **Settling Time** tab

Control element	Function
Automatic Settling Time	The user may choose to use an automatic settling time, which is the recommended (default) setting. The automatic settling time will set the most appropriate settling time for each beam deflection within the patterning procedure. The patterning will therefore be as fast as possible, since the settling time is never longer than needed. The quality of the patterning will also be optimum, since none of the settling times will be too short. Automatic settling time uses a reference value for the algorithm. The reference value has units of ms per mm. The settling time calculated is directly proportional to distance moved. The entered value determines the number of milliseconds of settling time per mm of travel. The reference value for the automatic settling time may be adjusted in the main menu **Extras**➡**Settings**➡**Virtual Beam Control** by setting the variable **Auto Setting Factor** to an appropriate value.

To reduce the time taken for patterning, the reference value for the **Auto Settling Factor** may be reduced, but this should be done only in conjunction with observations of the quality of patterning outcomes, not simply based on the need for faster operation.

Control element	Function
Manual Settling Time	The user may select a single, constant settling time for all beam deflections within the pattern. Setting a value of **0** will result in no settling time applied after any beam deflections, which may be appropriate in the case of fast electrostatic columns. For electromagnetic systems, selecting a manual settling time that is too short will most likely result in errors at the beginning of a new structure, since the start of a new structure may follow a relatively large beam deflection from the end of the last structure.

There are several potential disadvantages in the selection of a manual settling time and the user should be familiar with these possible consequences before using a manual settling time. The manually set settling time must be long enough to be suitable for all beam deflections, including deflections over relatively large distances between structures. Manual settling time will consequently often result in longer total patterning times.

Some electrostatic columns may require no settling time, since the deflection of the beam within and between structures is very fast. Even so, it is recommended that the user gains some experience of the quality of patterning outcomes before deciding that settling time will always be set to zero. It is recommended that a settling time is always used for electromagnetic beam deflection.

If MBMS is installed (which requires the optional Raith feature **periodixx**), the **MBMS settling time** can be defined in the **Settling Time** tab.

Control element	Function
MBMS	The user may select or deselect, if a settling time for MBMS elements is applied.
factor	Value in ms/mm, which is used to calculated the settling time with respect to the jump distance between elements.
Settling threshold	If the jump distance between two elements is smaller than this value, no settling time will be applied.

MBMS elements are always exposed outwards. Therefore the exposure begins in the center of the MBMS elements, which allows reducing the MBMS settling time factor for individual applications compared to conventional exposures. The required smallest factor for highest placement accuracy depends on the layout and needs to be tested individually by the user in order to fully minimize the overall patterning time.

In MBMS patterning mode, the settling time is implemented by inserting blanked pixels. The number of pixels per period becomes larger if the **MBMS Settling Time Factor** is increased. Therefore, it is recommend to ensure that the maximum number of pixels is not exceeded after changing the (⇒ Figure 5-29 on page 5-49).

3.3.5 Advanced Details

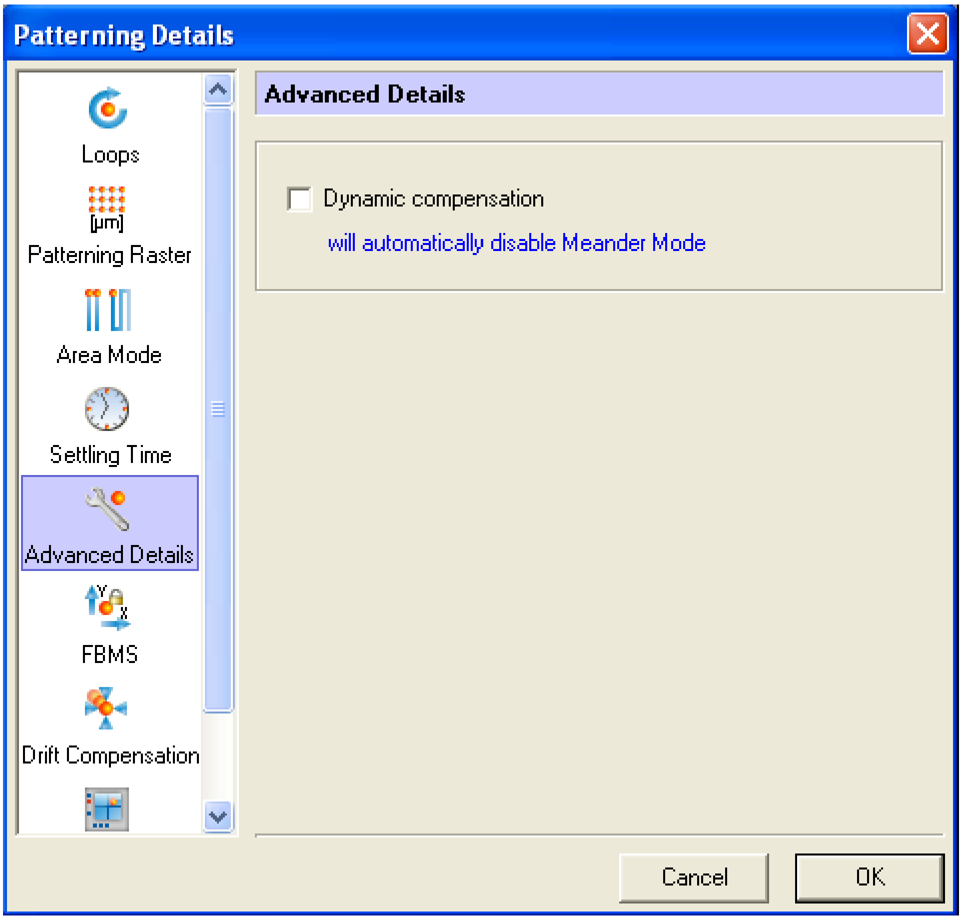


Figure 14-25: **Patterning Details** dialog, **Advanced Details** tab

Control element	Function
Dynamic compensation	Required for electromagnetic beam deflection only.

It is also possible to switch off the **Dynamic compensation** applied to the beam. All Raith systems have an algorithm to compensate dynamic effects, which occur when steering the beam, whereas not all Raith attachments have this option. Thus, only for Raith systems is it possible to switch this mode on. If dynamic compensation has been switched off, much faster patterning can be expected, but this may well be with a reduced placement accuracy.

3.3.6 FBMS

The two tabs are available for FBMS, **Circle generator** and **User defined beam deflection**.

Circle generator tab

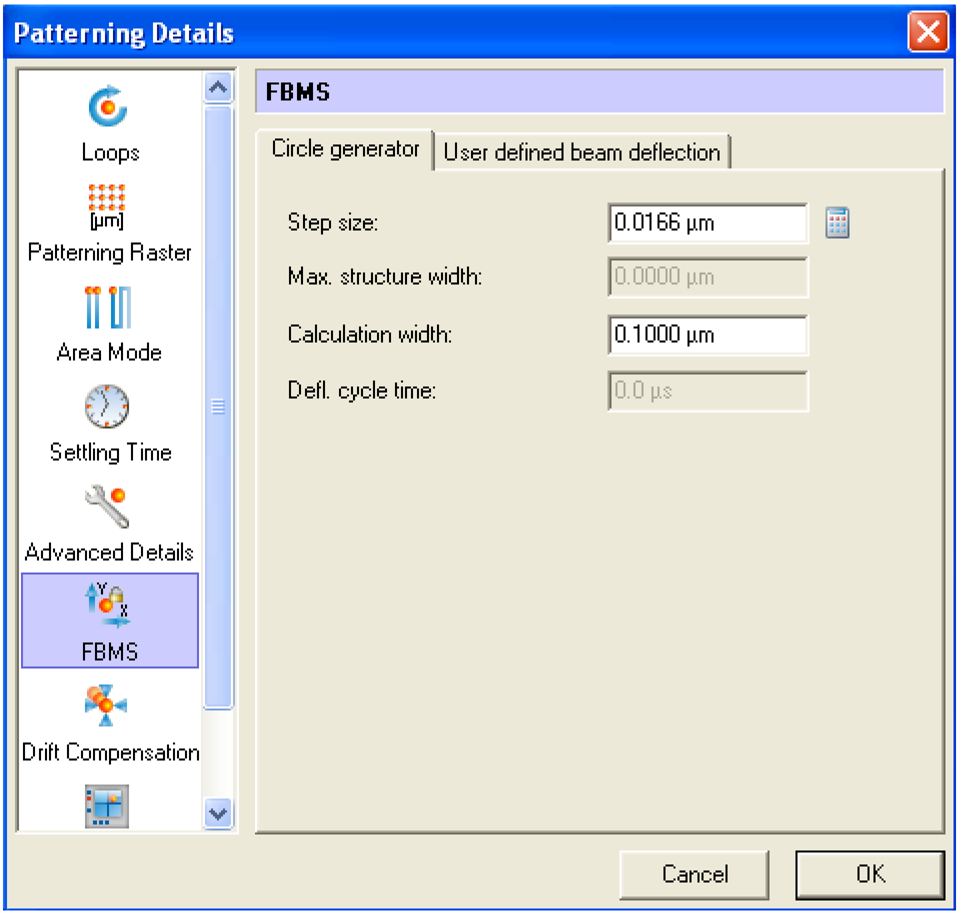


Figure 14-26: **Patterning Details** dialog, **FBMS**, **Circle Generator** tab

Control element	Function
Step size	May be entered by the user. It determines the distance between the dots on each circle circumference.
Max. structure width	Automatically calculated by the software and is dependent on the given Step size.
Calculation width	May be set for calculation of the corresponding cycle time.
Defl. cycle time	This value is automatically calculated and is the dwell time used. It can only be set in **Module Status**.

The circle generator allows the definition of a line as a **Curved Path** with a pre-defined width with larger process latitude than in dots mode. The line width can be specified directly in the GDSII design by the curved path parameter **Width**.

If the parameter **Max. structure width** has a non-zero value, the pattern generator creates a high speed circulating shape, while the stage is moving along a path. Due to the high speed of the beam in its circular movement, (single pixel addressing is typically done in the 0.5 to 1 MHz regime) the resulting beam dose profile is almost stationary relative to the moving stage.

The circle-symmetric pixel pattern used is automatically calculated by the system. The parameter that can be set by the operator for this mode is the **Step size**, which gives the average step size perpendicular to the stage movement. It therefore corresponds to the area step size for conventional GDSII elements and should be set with respect to the chosen aperture and resist type as in normal area patterning mode.

As in the **Patterning Parameter Calculator** window, a **Calculation width** can be set for calculation of the resulting cycle time. Thereby, the operator may check if the cycle time is reasonable with respect to the area stage speed.

The area definitions are taken from the **GDSII Editor**, where the design parameters are created and saved. The FBMS circle generator must calculate the concentric circle movements, including the radius of each circle and the step size between each dot that will achieve the required homogeneous patterning, depending on the resist. The step size for the pattern determines the distance between dots on each circle circumference, but the software will actually determine the pattern of dots on each circle.

User-defined beam deflection tab

The user-defined beam shape mode is activated by checking the checkbox **Use user pattern**. If checked, all FBMS paths will be written with the user-defined pattern. The result is a mixed patterning of a fast, continuously repeated deflection by the pattern generator, in which the center position of the deflection follows the path of the FBMS single-pixel.

In this tab you can assign, in the **User pattern file** field, an ASCII data file, which defines the deflection of the beam. The beam deflection values in this file may be changed by the user.

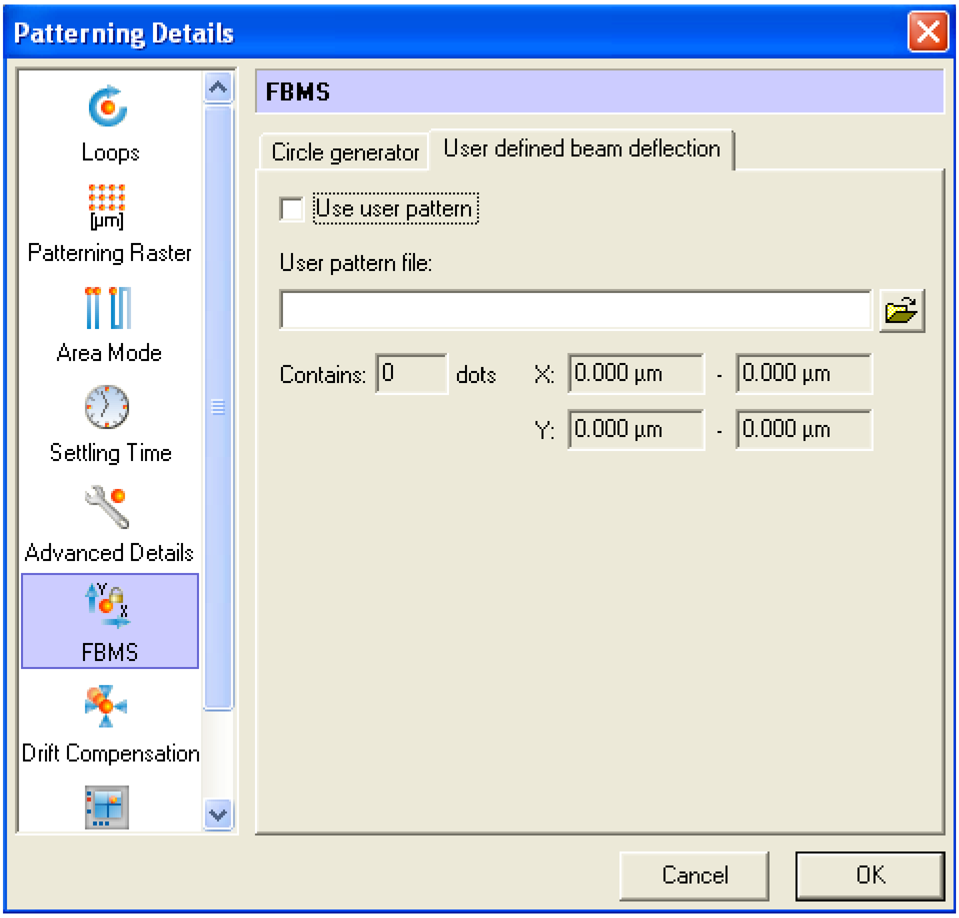


Figure 14-27: **Patterning Details** dialog, **FBMS**, **User defined beam deflection** tab

Control element	Function
Use user pattern	Check the checkbox to use the user pattern.
Use pattern file	Select a pattern file.
Contains (n) dots	The software will automatically calculate the number of dots, as defined in the assigned ASCII file.
X dimension	The software will calculate the X dimension of the user-defined shape.
Y dimension	The software will calculate the Y dimension of the user-defined shape.

Only experienced users with advanced skills in the technique should change the beam deflection values.

Furthermore, the operator may define user-specific beam shape patterns for FBMS writing. The pattern could be either a different circle generator pattern defined by the user for dedicated patterning tasks, or a special geometric pixel pattern for further sophisticated applications.

The pixel pattern is defined in a text file with a series of dots that are defined in Raith's ASCII format. Thereby Raith's **GDSII Editor** can be used to create the pattern as a GDSII file. The required ASCII file is easily generated by applying the ASCII export filter to the GDSII dot file. The only relevant dot data in the ASCII file for the **FBMS Shape mode** are the pixel XY coordinates, which define for this mode the offset to the center position 0,0. Further parameters of the dots in the ASCII file like **Dose**, **Layer**, and **Width** are simply ignored.

In order to use a specific ASCII file for the beam shape mode, its directory location and file name must be set in the **FBMS Parameter** window.

The number of pixels in the ASCII file and the maximum coordinate offset in X and Y are automatically displayed after defining the filename and folder location. The maximum number of pixels in **Beam Shape mode** is 3400. If the file contains more than 3400 pixels, pixel number 3401 and all following pixels will simply be ignored. Moreover, the maximum offset in X and Y has to fit the scan field range. In order to have sufficient additional scan range for beam tracking during FBMS writing, a reasonable maximum offset is half of the maximum scan deflection.

3.3.7 Drift Compensation

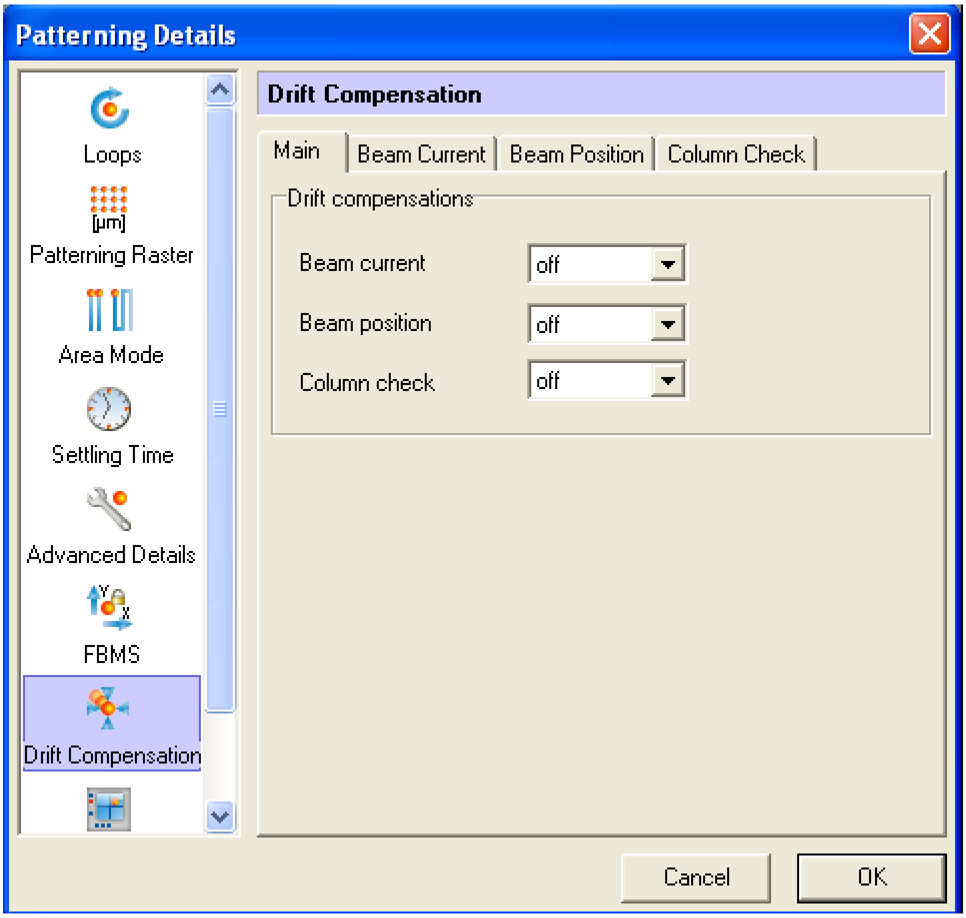


Figure 14-28: **Patterning Details** dialog, **Drift Compensation**, Main tab

There are several tabs available:

- **Main**
- **Beam Current**
- **Beam Position**
- **Column Check**

This facility is very useful when multi-field patterning is carried out. When a very large structure is defined, it is sometimes useful to divide it into a number of individual fields, for sequential patterning as a series of stitched writefields. When writing for long periods, drift effects can occur.

Main tab

Control element	Function
Beam current	The beam current drift compensation can be switched off, or may be performed after completing a defined number of stitch fields, or after a defined time.
Beam position	The beam position drift compensation can be switched off, or may be performed after completing a defined number of stitch fields, or after a defined time.
Column check	The column check can be switched off, or may be performed after completing a defined number of stitch fields, or after a defined time.

Beam Current tab

There can be a small drift in the beam current during the writing of large patterns, which will mean that the chosen dwell time is no longer appropriate to achieve the required dose, since the dwell time is chosen based on the beam current value. This drift will result in under-exposure or over-exposure of the pattern and differences in the patterning in different stitched regions.

A beam current drift correction may be selected by the user, either to be completed after a period of time (**Beam current➡time**) or after completion of a certain number of stitch fields (**Beam current➡stitch field➡n**). Typically, drift correction may be carried out every 15 minutes.

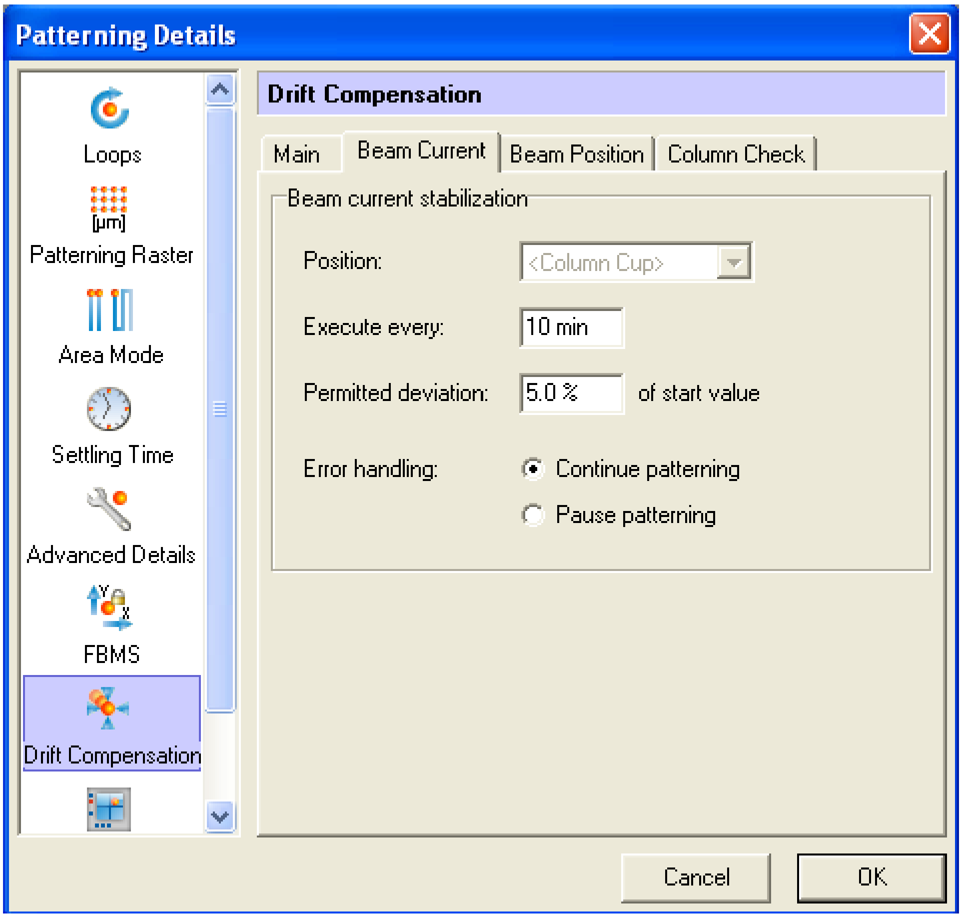


Figure 14-29: **Patterning Details** dialog, **Drift Compensation**, **Beam Current** tab

Control element	Function
Position	Position name as defined in the **Stage Control** window
Execute every	Minutes or stitch field number
Permitted deviation	Enter (in %) the permitted deviation from the starting value.
Error handling	- **Continue patterning** or - **Pause patterning**

The automatic beam current drift compensation will not adjust the values in the beam current module. The compensation will not change these global instrument settings. The **Patterning Parameter** window will not, therefore, give an in-situ indication of any drift compensation. The beam current drift compensation adjusts the patterning parameters during the patterning.

The user may also decide on the action to be taken if the RSD% is equal to or greater than the defined value, either to continue or to pause the patterning. If the RSD% is within the defined limits, the system will measure the current and will calculate the dwell time for all element structures automatically.

When the relative standard deviation exceeds the permitted limit and if the user has chosen to continue the patterning, the pattern will be completed without changing any values. If the user chose to pause the procedure, a warning message will prompt the user that the drift is outside the permitted range and the procedure will be paused.

Beam Position Drift tab

There can be a drift in the relative positioning of the stage and the beam. It has the effect of a drift in the placing of the beam on the sample and therefore inaccurate writefield alignment. Beam position drift correction will carry out a writefield alignment correction at some interval determined by the user. This correction may be carried out after the completion of a number of stitched fields, or alternatively, after a user-defined period of time.

All automatic writefield procedures, listed in the **Scan Manager**, may be executed automatically, by selecting from the list of pre-written procedures. The software will carry out all patterning using updated values arising from any corrections carried out by the beam position drift correction. Beam position drift is also discussed in the chapter **Beam to Sample Alignment**, (⇒ *Beam to Sample Alignment* on page 13-1).

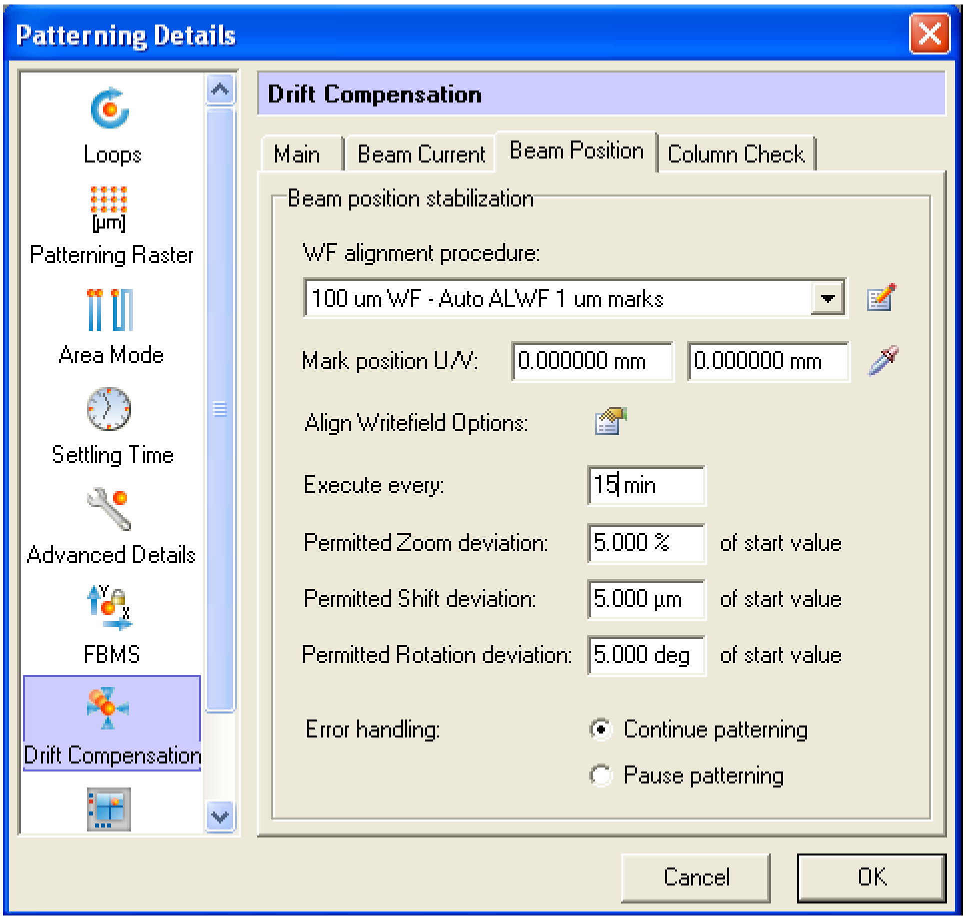


Figure 14-30: **Patterning Details** dialog, **Drift Compensation**, **Beam Position** tab

Control element	Function
WF alignment procedure	Select a writefield alignment procedure from the data-base.
Mark position U/V	Defines the position at which the WF alignment procedure must be executed.
Align Writefield Options	Accesses the **Writefield Alignment Options** dialog, (⇒ *Tool bar* on page 11-16).
Execute every	Select a number of minutes or number of stitch fields.
Permitted Zoom deviation	Enter the permitted deviation (%) from the starting value for **Zoom**.
Permitted Shift deviation	Enter the permitted deviation (%) from the starting value for **Shift**.

Control element	Function
Permitted Rotation deviation	Enter the permitted deviation (%) from the starting value for **Rotation.**
Error handling	**- Continue patterning** or **- Pause patterning**

When **Execute every nth minute** is chosen, the user may enter the new values individually for each parameter for the **Writefield Alignment**, including beam shift and rotation. If the relative standard deviation (RSD%) is within the permitted limits, then the new **Writefield Alignment** parameters are used. If the RSD% exceeds the permitted limits, the patterning will either be paused, if that is the selected option, or continue, using the old values, if that is the selected option.

The Raith protocol tool will display all drift compensation values and outcomes. All of the details of the calculated values for each pattern may be viewed if required.

Column Check tab

The column check will only function on certain column types. The high tension voltage (EHT), as well as the gun status may be checked automatically. The check on the gun status is simply to determine if the gun is on or off.

The EHT and extractor voltage are quantifiable parameters and the user may input a permitted range, or the relative standard deviation (RSD%) of each one. If the permitted RSD% is exceeded, the user may pre-define the action to be taken. The user can choose to either pause the patterning, or to continue the patterning.

In the beam current module, one of several options can be selected from the dropdown list. The column check drift can be completed after writing a number of fields or after a user-defined time. The user may then define the acceptable RSD%, relative to the starting value.

Please read the chapter **Column Control** for a detailed description of the Column parameters (⇒ *Column Control Tool bar* on page 7-20).

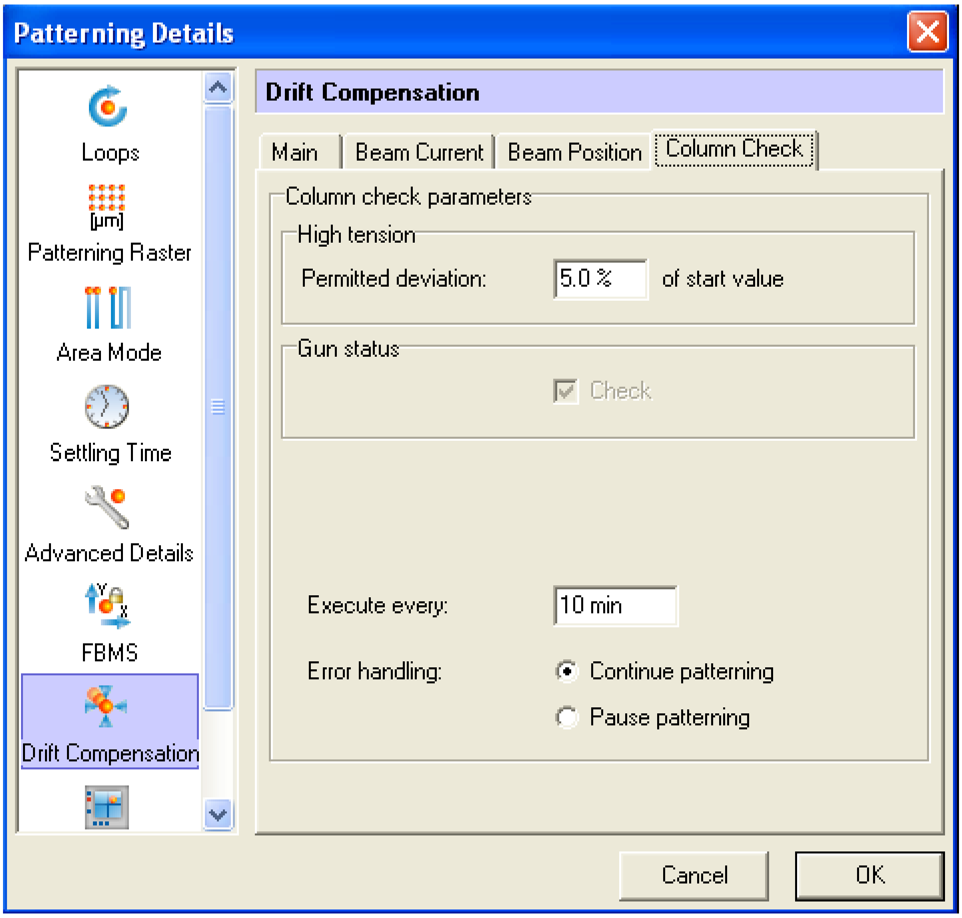


Figure 14-31: **Patterning Details** dialog, **Drift Compensation**, **Column Check** tab

Control element	Function
High tension Permitted deviation	Enter the permitted deviation (%) from the starting value for High tension.
Gun status	Check if the gun status should be verified.
Execute every	Enter how often the **Column Check** should be carried out.
Error handling	- **Continue patterning** or - **Pause patterning**

3.3.8 Visualization

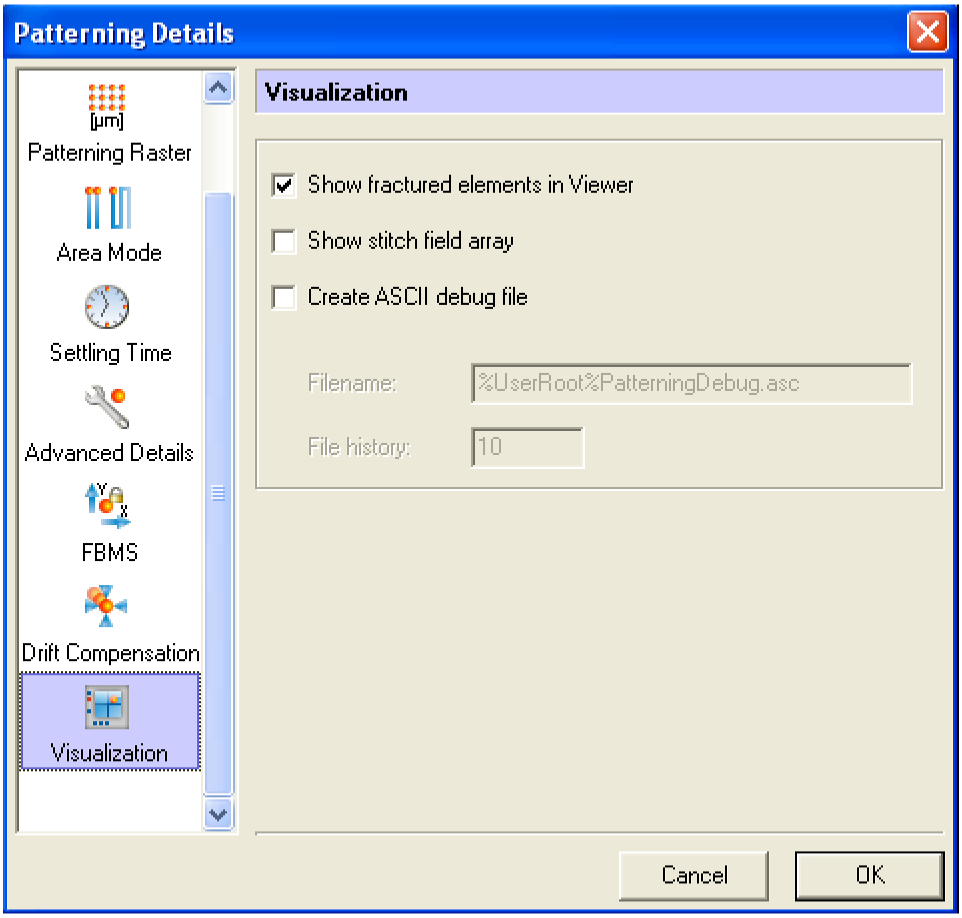


Figure 14-32: **Patterning Details** dialog, **Visualization** tab.

Control element	Function
Show fractured elements in Viewer	This enables the visualization of the currently fractured elements as an overlay over the GDSII viewer.
Show stitch field array	This enables a coarse view of the currently processed writefields.
Create ASCII debug file	This writes the fractured elements into one file using the ASCII format.
Filename	The file name and location of ASCII debug file.
File history	Shows how many different files can be stored in the file history. The user can change this value by adjusting the variable **Number of ExpoDebug files** in the menu **Extras➡Settings➡Patterning.**

During the patterning, within the **Beam Patterning Control** dialog, a table may be displayed called **Show stitchfield area**. The software will rasterize the complete design working area and will show the progress in real time of the patterning process.

In **Beam Patterning Control**, the stitchfield will be shown. The table shows the working area with each single writefield. Those in green have been read and those in white will be read later, (⇒ *Beam Current* on page 14-59).

To display the **Beam Patterning Control,** the patterning has to be initiated.

If **Create ASCII debug file** is **On**, a file will be created and stored in **<Raith root >➡User➡<user name>**. The output of the pattern, fractured into elements and clipped to writefield borders, will be named **patterning 1**, **patterning 2** etc. up to **patterning 10** and will be stored in this patterning history.

The ASCII data file may be imported and viewed in the GDSII Database, via **File➡Load ASCII**. The structures that are preprocessed by the software into basic elements ready to be sent to the pattern generator, will be displayed. Errors in the design may be located in this way, prior to starting the patterning.

This is very useful for checking the fracturing of elements, i.e. the division of elements into smaller individual writefields. This is performed when a structure will span more that one work field.

These ASCII debug files are usually only viewed if any problems have occurred during patterning, such as a failure to write some structure elements or misplacement of any structures. The ASCII file may then be used to troubleshoot the problem, but switching on this option will affect the performance of the patterning process because writing data to a file is quite slow.

If MBMS is installed (which requires the optional Raith feature **periodixx**), two additional ASCII debug files are generated: **PeriodiXXDebugLines.asc** and **PeriodiXXDebug.asc.**
PeriodiXXDebugLines.asc contains the base element after fracturing into lines.
PeriodiXXDebug.asc is the final pixel pattern of a complete period, including shifts between pixels that compensate for the continuous stage movement during the patterning.

3.3.9 Grayscale Bitmap (option)

The Grayscale Bitmap is a special patterning feature in the Raith NanoSuite software that requires the 3Lith license. The size of bitmap files that are referenced inside the GDSII structure is limited to about 13000x13000 pixels and depends on operating system. The Grayscale Bitmap allows import of bitmap files with 8 bit gray scale resolution. The assignment of intensity to patterning dose can be done linear or via contrast curve.

NOTICE

Adding bitmaps to GDSII structure takes place in **GDSII Editor**, **Bitmap Properties** dialog (⇒ *Adding bitmaps as reference*).

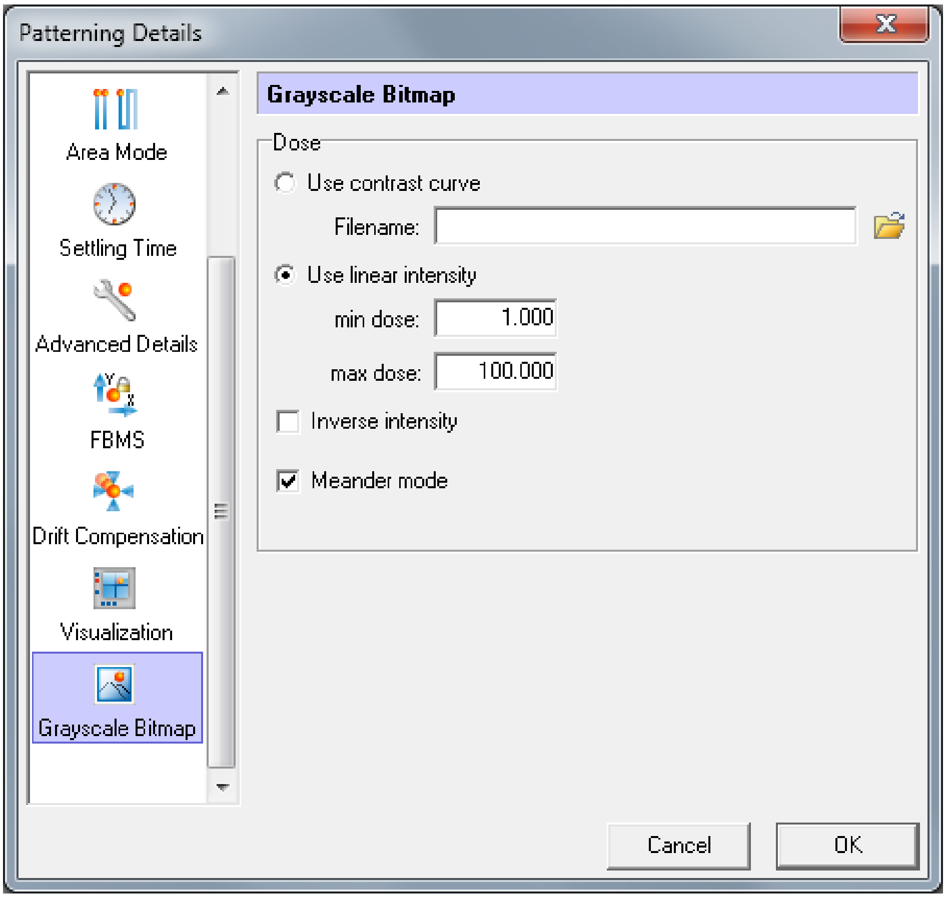


Figure 14-33: **Patterning Details** dialog, **Grayscale bitmap** tab.

Control element	Function
Use contrast curve	Optional, you can define the dose using a contrast curve. The contrast curve is a text file, in which the assignment of the dose values to the resist thickness is defined. The following format is used per row: Dose value as a real figure > TAB sign > resist thickness as a real figure. You can find some examples for contrast curve definitions in the **<Raith root> directory > Proxy folder**.
Use linear intensity	This intensity is transferred into a dose for the corresponding GDSII element. **Min dose** and **max dose** specifies the minimum and maximum dose factor for minimum and maximum gray scale value in bitmap correspondingly.
Inverse intensity	Select this option to apply the smallest intensity to the maximum dose factor and vice versa.
Meander mode	Select this option to pattern bitmap in meander mode. By default, bitmap are patterned in line mode.

4 Patterning Control

Once a patterning task has been inserted into the Positionlist via drag and drop, the patterning procedure can be executed.

A **Patterning Control** window will open to show the progress of the overall patterning procedure. This window will be displayed once the patterning has been started. Patterning may be paused or stopped via this window and the current patterning status is shown. The user can view the progress of patterning, since elapsed time and events are shown. GDSII structures are read in fractured and sent to the pattern generator.

Once the clipping process is complete, all of the data are loaded into the buffer. Once the elements to be written are uploaded, they will be patterned sequentially in the defined order. All of the structure positions are stored, allowing initiation of the movement to the start position of each structure. **Drift compensation** is carried out throughout the execution, as defined by the user (⇒ *Drift Compensation* on page 14-46).

The number of writefields to be patterned will be displayed. The current writefield is indicated, so that the user can see the progress made so far. The total elapsed time for the completed structures will also be shown.

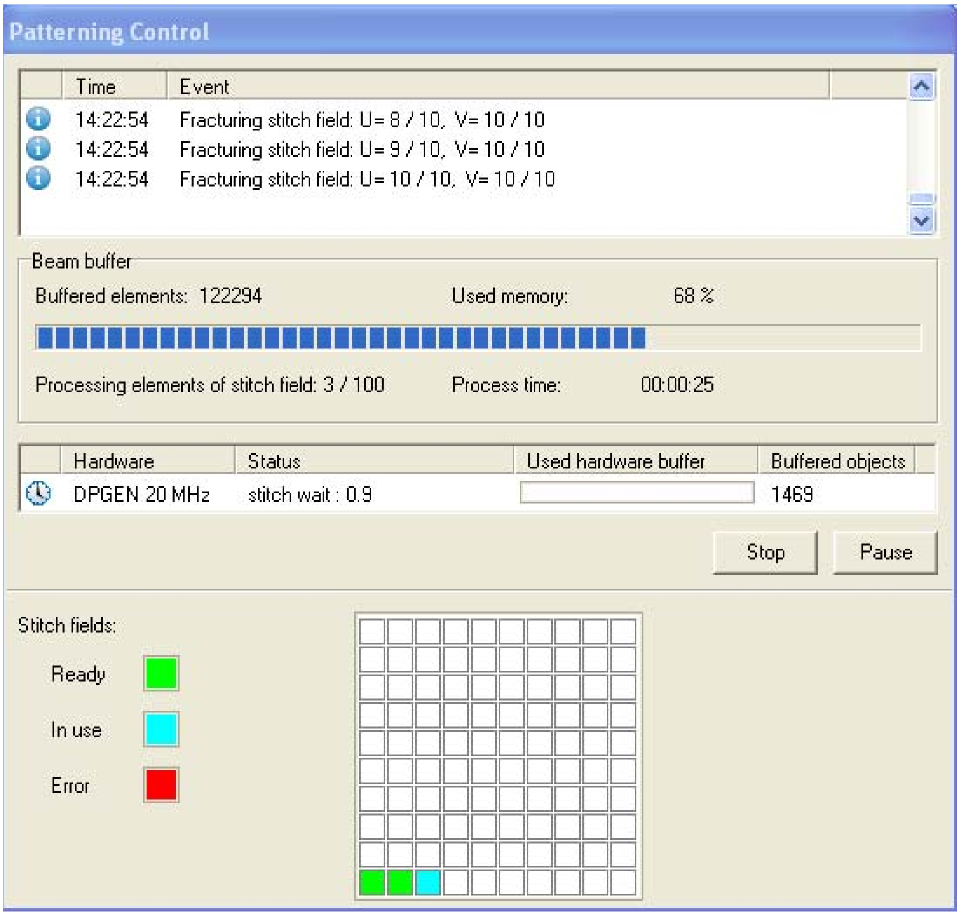


Figure 14-34: **Patterning Control** dialog

Control element	**Function**
Time / Event	The time and the current event are displayed
Beam buffer subset:	
Buffered elements	Shows the current number of buffered elements. At first, the amount of data stored in the buffer may be large and it will slowly decrease as the elements are written. The capacity of the buffer is displayed, along with the number of objects still to be loaded.
Used memory	The percentage of currently used memory is displayed.
Processing elements of stitch field	The number of currently processed stitchfields and total number of stitchfields are displayed.
Hardware	The currently used hardware will be shown.

Control element	Function
Status	The Status of the current activity is displayed, so the user can see if the system is patterning, waiting for a stitching procedure etc.
Used hardware buffer	Used hardware buffer is always displayed. This buffer is used for the current patterning. The buffer capacity and current usage are shown.
Buffered objects	The number of objects being sent to the hardware are displayed.
Stop, Pause	The patterning process can be paused or stopped using this button. Once the patterning procedure has been stopped, this window will be closed. As soon as the window is closed, the user is alerted that the patterning is stopped.
Stitch fields	The schematic shows the stitching process in progress.
Green	Ready
Blue	In use
Red	Error

When the patterning is stopped using this window, patterning will not be stopped immediately. Any objects that are stored in the hardware buffer will be written. Only when the buffer is empty, will the patterning procedure be stopped, followed by the automatic closure of the window.

For example, a simple structure such as a rectangle, hardly requires any hardware buffer space. It will be loaded very quickly and even when the user gives the command to stop the patterning, the structure is likely to have been already transferred to the buffer.
If the structure is written using a long dwell time, it might take several minutes before the structure has been written and the window is closed.

5 Beam Current

The Beam current measurement will be displayed.

Figure 14-35: Beam current measurement

Control element	Function
Display field	The larger value is the current value and the smaller value beneath is the last measured value.
Units	The user can change between units by selecting the unit from the dropdown list. The software will automatically recalculate the value in the new unit and display it.
Measure	Clicking on the **Measure** button will initiate a new measurement of the beam current.
<current position> field	From the dropdown list, the user can choose where the beam current should be measured. Most sample holders include a Faraday cup to measure the beam current. The stage will automatically drive to the specified position to measure the current and if the **drive back** checkbox is checked, the stage will drive back to the previous stage position.

Whenever a beam current measurement is performed, the reading will be saved in the patterning parameters, so that the dwell times for Areas, Lines, Dots etc. must be recalculated via the **Patterning Parameter Calculation** dialog.

Chapter 15
Automation

This chapter gives an overview of how scripting can be used to implement Automation functions.

The following topics and tasks are included:

- Background information on Java scripts, Visual basic scripts and Macros
(⇒ *Functional Description* on page 15-3),
- Creating a completely new script, (⇒ *Creating a new Script* on page 15-5) or opening an existing script and editing it
(⇒ *Opening a Script* on page 15-5),
- Explaining how single functions in Java script can be tested, (⇒ *How to test single functions in a Java script* on page 15-7) and how the Raith Scripting Help can be used for guidance
(⇒ *Using Raith Scripting Help to write a script* on page 15-8),
- How to work with Macros
(⇒ *Opening and editing a Macro* on page 15-9),
- Functionality of drag and drop into the Positionlist
(⇒ *How to drag and drop a script into a Positionlist* on page 15-10),
- Explanation of the Library folder, where all scripts are saved
(⇒ *How to view the Library folder* on page 15-11),
- Detailed description of the Automation dialog and all tool bar dialogs
(⇒ *Automation dialog* on page 15-12).

You can call up **Automation** by clicking the (Automation) icon in the control bar

---

The dialog is displayed to the right, next to the control bar. The text **Automation** will appear on the control bar.

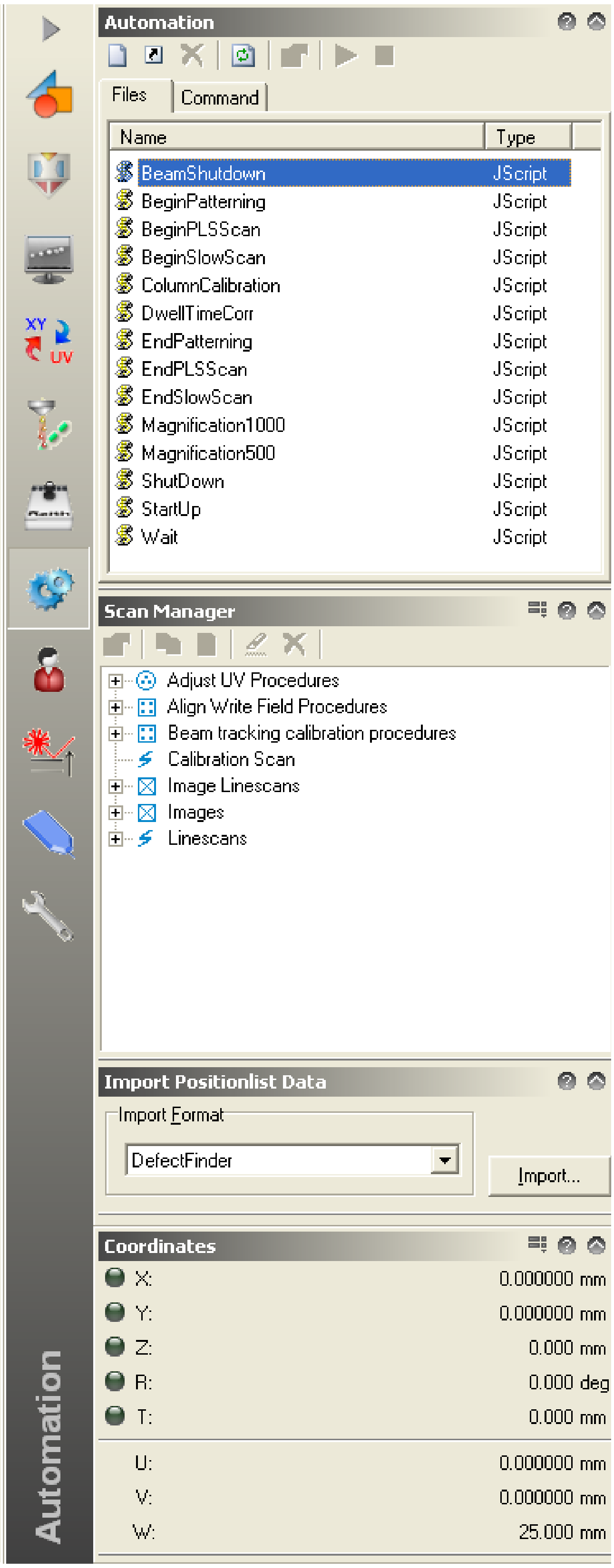


Figure 15-1: Automation

A Functional Description

Java script and Visual Basic script are well known programming languages and it is assumed that the user has some prior experience of using them.

To learn about Windows scripting and the syntax of Java script, please refer to Microsoft Windows Script Technologies documentation, which can be downloaded from the Microsoft official web site: *http://www.microsoft.com/downloads/details.aspx?familyid=01592C48-207D-4BE1-8A76-1C4099D7BBB9&displaylang=en*.

The scripts are utilized within the Raith software platform, in which they are run within the context of the instrument software, not on the basis of Windows™. It is therefore possible to use generic script commands not only within Java or Visual Basic, but also to drive the Raith software directly.

The **Automation** window has been designed to give the user direct control of the Raith software. A particular instrument procedure may now be modified and controlled by the user, giving the freedom and flexibility to program complex procedures.

Macro is Raith's own macro interpreter, allowing macros to be called up within the Raith software.

Pure stand-alone macros are rarely used these days. It is recommended that you use Java and Visual Basic scripting, since these languages provide more opportunities to use programming tools, conditional commands, the use of logic, as well as the creation of loops and the linking of different scripts, as in a programming language.

Using Macro, macro commands may only be called up sequentially.

The Raith script library contains a series of generic scripts written in Java script, but it is equally possible to use combinations of Java script, Visual Basic script and Macro commands. Macro commands may be also embedded in Java scripts.

Java script is a programming language, in which commands can be looped and linked. Exceptions and other logical conditions can also be generated. A macro can be executed within Java script. Using the pre-defined **App.Exec** method, any macro commands may be embedded within Java script.

Whenever logic is required, a programming language which is syntax-based, such as Java script, is required. Using Java script, the user may create loops, link to other commands and include conditional actions, depending upon a value being true/untrue. Comments may be highlighted in a different color within the script and interpreter messages may be issued.

B Tasks

1 Creating a new Script

Proceed as follows to create a new script.

▶ Click on the **Create a new file** button.

This will open a dialog box, in which the user may select from one of the following three file types:

- Java script
- Visual Basic script
- Macro

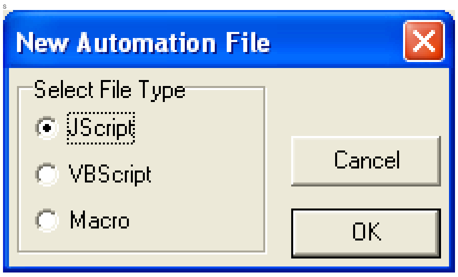


Figure 15-2: Creating a new script file

2 Opening a Script

Proceed as follows to open a previously written script.

▶ Click on **Open Script File** button.

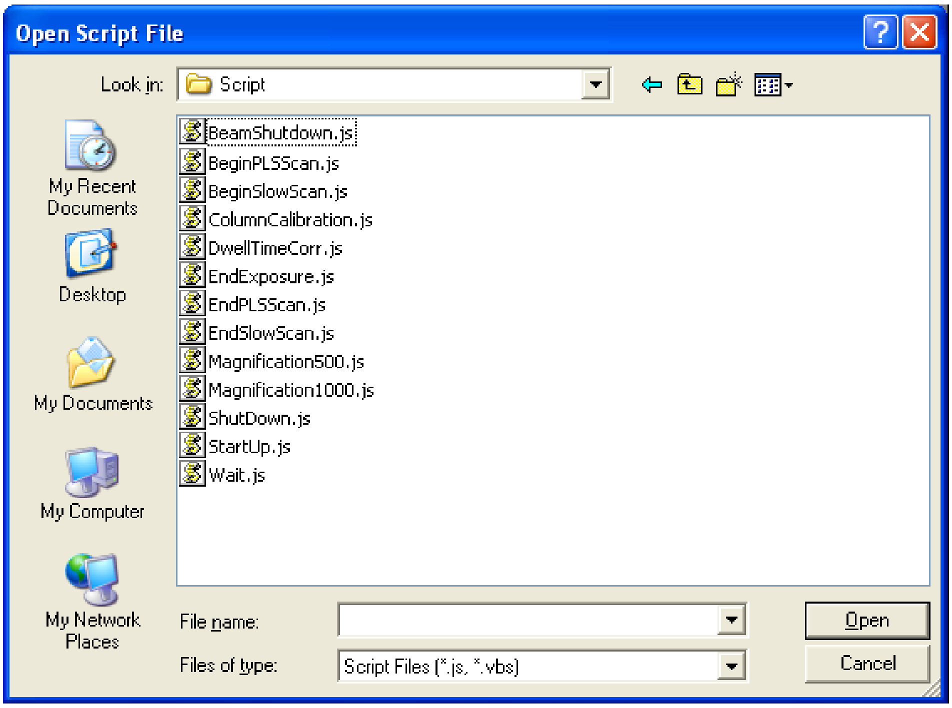


Figure 15-3: Opening a Script file

- When you click on the **Open Script** button, the corresponding dialog box shows the user directory. Each registered User has a folder called **Scripts.** This folder contains any scripts saved by that user.

All users have access to the generic scripts and any newly created scripts. As examples, the generic scripts **BeginPatterning, BeginPLSScan, EndPLSScan, EndPatterning, BeginSlowscan** and **EndSlowscan** are very specific commands which initiate certain key tasks and functions. These are located in the User's Script folder. When, for example, a Positionlist is scanned within the Raith software, the script **BeginPLSScan** is called up. The user may change these scripts to achieve some specific actions connected to predefined events mentioned above.

Besides creating new scripts, a user may select any script from the user directory and adapt it to specific requirements. For example, a script may include a command which sends an e-mail alert to the user at key points, such as when a Positionlist is first initiated, or completed.

As a further example, a script may save a Positionlist before completion. This is particularly useful when the user wants to avoid data loss.

- Double click on an existing script in the **Automation** window, e.g. **Wait.** The script will be displayed.

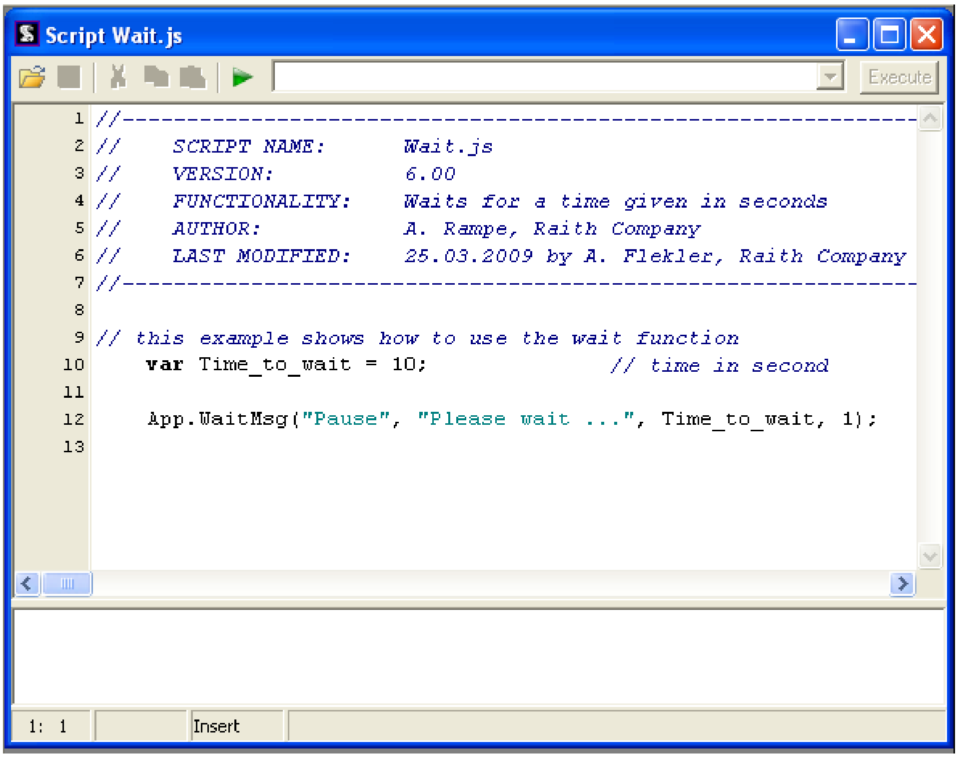


Figure 15-4: Selecting an existing script

3 How to test single functions in a Java script

- To test a single Java script function within the **Script Editor**, proceed as follows. Select the Script that you wish to test. The script must contain function declarations only, but no function calls. Such scripts are a kind of scripting library that provides other scripts with function declarations.

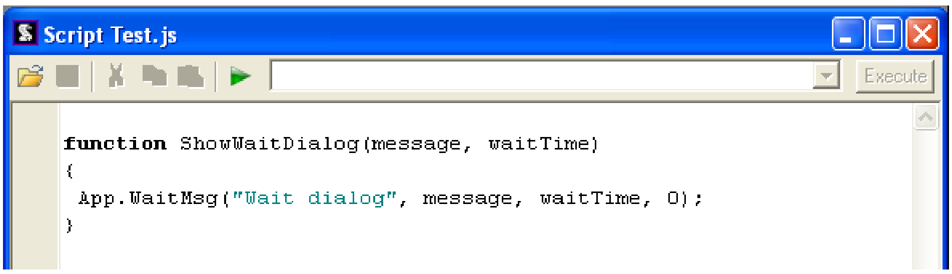


Figure 15-5: Selecting a **Java script** file for testing

- Click on **Run** . This will initiate the Interpreter and all functions inside the script will be loaded in the dropdown list inside the tool bar. The name of the function is now displayed.

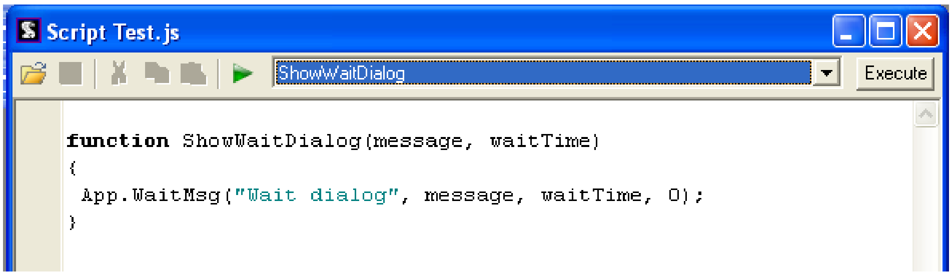


Figure 15-6: **Run** the selected script

- Click on **Execute**. Only the selected function will now be executed.

A new dialog will now open, showing the **Procedure Parameters**.

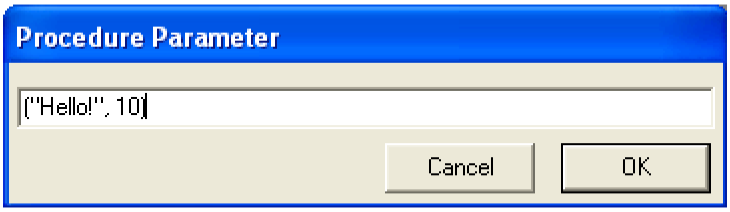


Figure 15-7: Procedure Parameters

Enter the required value for an existing function parameter in brackets, separated by a comma and click on **OK**.
Now the procedure will be executed by the script engine.
Only the function parameter syntax itself (in brackets) should be copied and pasted.

---

4 Using Raith Scripting Help to write a script

Within the Raith directory, script files written by Raith are available. The **Raith Scripting Help** file describes all Java script and Visual Basic script objects, methods and ways in which Java script can be utilized. This Help file is continuously updated, to include descriptions of all available scripts. In addition, examples are given, explaining Java scripts and the tasks that they perform. Macros are documented within the **Commands** dialog, (⇒ *Command tab* on page 15-14).

Proceed as follows to obtain help on scripting.

- In order to access the Raith Scripting Help, go to **Start menu➡Programs➡Raith➡Scripting Help.**

5 Opening and editing a Macro

Proceed as follows to select an existing macro command.

▶ Click on the **Commands** button, which will open a list of all available macro commands in the software.

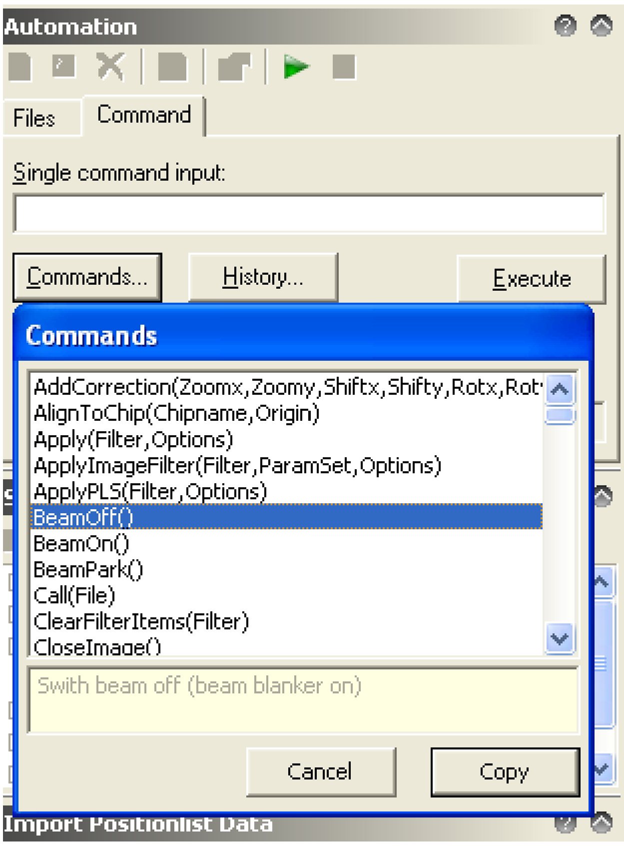


Figure 15-8: Selecting an existing macro command

Macros are direct commands, which can be called up via the **Commands** button. This will open a new window with a listbox in which all available Macro commands are listed. A desired macro can be selected and may then be copied and pasted.

The selection and implementation of a macro is straightforward. The user selects a command from the list and a description of the selected command will be displayed in the text box below.

For example, selecting and executing the **BeamOn()** macro will switch on the beam.

To create a new macro file, you must go to **Create new file** and select **Macro**. Once a new macro has been created and saved, it will be listed under the **File** tab.

A macro file may contain one or more macro commands. These will be executed sequentially in a batch process.

6 How to drag and drop a script into a Positionlist

When a new script has been created, it may be dragged and dropped into the Positionlist. Click on **Scan** to execute the Positionlist.

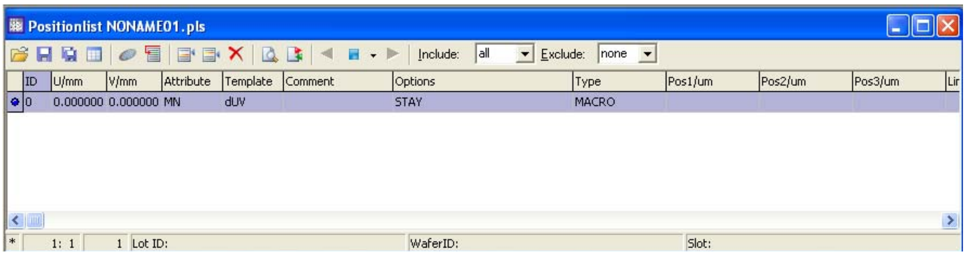


Figure 15-9: How to drag and drop a script into Positionlist

If a new script has been created and included in the Positionlist via the drag and drop and subsequently cannot be executed, go to the main menu bar **Scan**➡**Execute macro** in order to find the cause of the error.

The checkbox next to **Execute macro** must be checked, otherwise the macro in the Positionlist is deactivated. Check **Execute macro** and scan the Positionlist again. This will now successfully execute the macro in the Positionlist.

Upon installation, the Positionlist scanner in the Scan menu will be activated via the checkbox. During the development of applications in using the instrument, the user may sometimes choose to deactivate certain functions by unchecking them. For example, the user may choose to scan the full Positionlist without executing the macros in the Positionlist. Alternatively, the user may choose to scan the full Positionlist without patterning.

To learn more about the Positionlist, please read the chapter Positionlist, where all functions are explained in detail, (⇒ *Positionlist* on page 9-1).

7 How to view the Library folder

- Within Windows Explorer, **<Raith root>➡Lib** is the **Library** Folder, in which a number of scripts are available. The user may select and view any of these scripts.

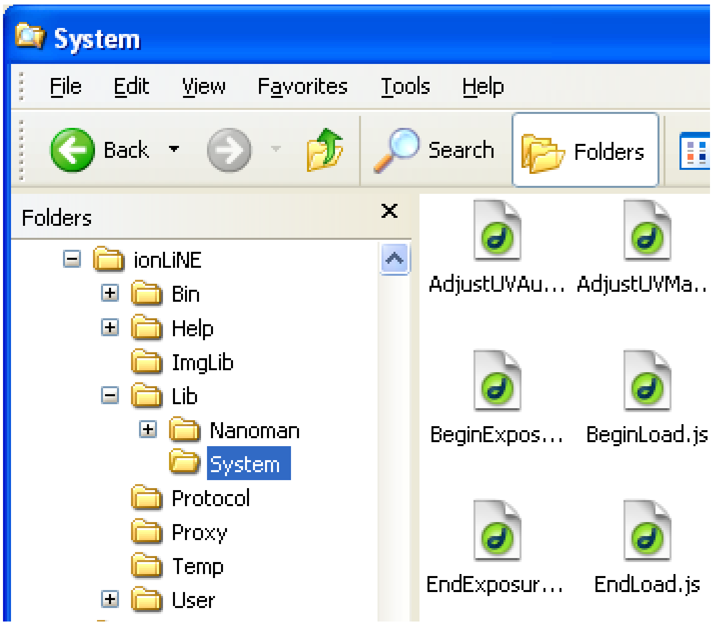


Figure 15-10: Viewing the Library folder

- Viewing library files can be useful. The user may, for example, like to know which pre-defined Java script functions are already available and can be applied in User Scripts.

On installation, there is a subfolder **<Raith root>➡Lib**. Within this library, there is a list of functions written in Java script. All users have access to this library folder. It is of vital importance that the user does not edit, rename or copy/paste any of these library files, which control important instrument functions. These files are loaded every time the software is initiated.

If the user renames or copy/pastes any of these files, they would not be recognized by the software the next time the software is initiated. The functions that they perform would therefore not be possible and an error message will be displayed.

The user may view these files, but they are generally rather complex scripts and any editing is very likely to have consequences for the operation of the instrument. A typical library script is the start-up and shut-down of the column. This is a much more complex script than the user might expect it to be and it is written by Raith specialists.

C Software Reference

1 Automation dialog

Automation comprises the following dialog:

Automation dialog for

- Administration of previously written scripts in Java script, Visual Basic script or Macro.
- Creating and saving new scripts.
- Scripting to enable user access to direct control of Raith software.
- Open and edit previously written Scripts in Java script, Visual Basic script or Macro
- Create new scripts
- Execute macro commands
- Drag and drop scripts into a Positionlist

2 Tool bar

You can use the tool bar to select the commands to open or create new Scripts.

You can access the current executable commands via the context menu.

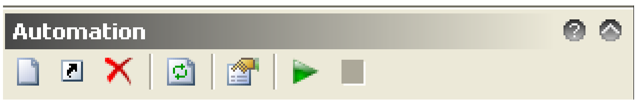


Figure 15-11: Tool bar

Control element	Function
	To create a new script.
	To create a link to another script, which can be located in any folder on the hard drive.
	To delete the selected script or link.

Control element	Function
	To refresh the file list of all previously written scripts in the Script folder. Any newly created scripts will now be displayed.
	To edit the selected script or macro file
	To load the selected script into the script engine and execute it.
	To stop the running script

It is possible to create a link to an existing script file. If a link to a script has been created, a link button will be displayed next to the file name. The button for scripts is color-coded, indicating **Java script**, **Visual Basic script** or **Macro script** types. It is possible to create a link to any script via **Create link.**

3 Working area

The working area is divided into two subsets:

- **File** - A list of all Java script files will be displayed.
- **Command** - A list of all macros will be displayed.

3.1 Files tab

When selecting the **Files** tab, all previously written Java script files will be displayed, from which a script can be selected.

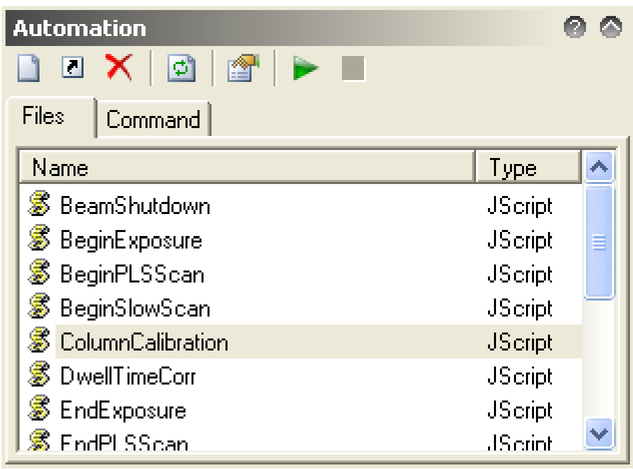


Figure 15-12: **Files** tab

All files within the User's **Script** folder will be displayed. If the user wishes to copy and paste a file within Windows Explorer in its own user directory, **<Raith root >➡User➡<User name>➡Script**, clicking the button, will update the list of scripts.

3.2 Command tab

The **Command** tab enables the administration and testing of single macro commands.

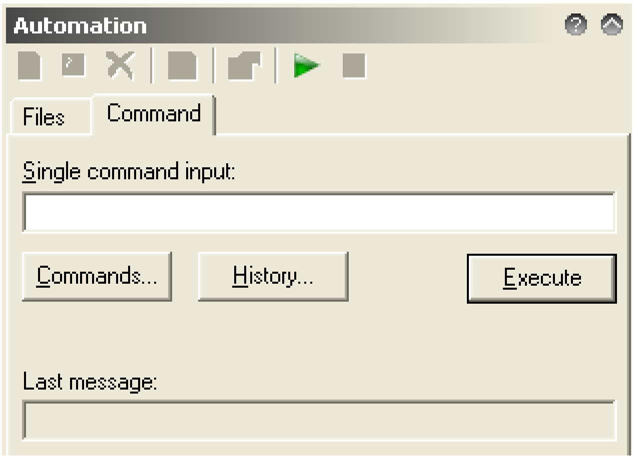


Figure 15-13: **Command** tab

Control element	Function
Single command input	A single macro command can be typed into the field.
Commands	To list all available macro commands
History	To list the History of the previously used macros
Execute	To execute the selected macro
Last message	To show the last message interpreter

A message can be displayed showing if a selected command has been executed successfully or not. In the event of a syntax error, an error message will be displayed, (e.g. **Syntax not found**).

4 Automation subdialogs

The following subdialogs are available from the Automation dialog.

4.1 Script Editor

To open the **Script Editor**, click on the **New** button, then select either **Java script**, **Visual Basic** script or a **Macro**.

After selecting a script type from the **New Automation File** dialog box, a new window, **Script Noname** will open, within which it is possible to edit scripts. The file name extension reflects the selected file type. The button next to the file name changes according to the selected file type.

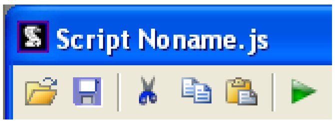


Figure 15-14: Opening the **Script Editor**

Control element	Function
	To open a new dialog box, displaying the contents of the most recently selected folder, where previously written scripts are stored.
	To save the script if changes have been made.
	To cut and paste parts or all of the scripts into another script
	To copy and paste parts or all of the script into another script
	To paste parts or all of the scripts into another script
	To run the script

5 Script Editor context menu

Right mouse click in the **Script Editor** dialog will open the following context:

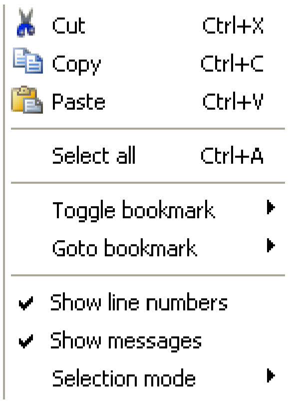


Figure 15-15: Context menu of the **Script Editor**

Context menu item	**Function**
Cut (Ctrl+X)	To cut and paste parts or all of the script into another script.
Copy (Ctrl+C)	To copy and paste parts or all of the script into another script.
Paste (Ctrl+V)	To paste parts or all of the script into another script.
Select all (Ctrl+A)	To select all of the script ready to be cut, copied or deleted.
Toggle bookmark	To toggle quickly between lines of code that have been bookmarked.
Goto bookmark	This is similar to above Toggle bookmark, but enables the user to choose a specific bookmark.
Show line numbers	To display the line number of each line of code on the left-hand side of the script.
Show messages	These are messages which are displayed by the scripting engine (for example, an error will be displayed with the line number at which the scripting error can be found).
Selection mode	To select from a range of modes via the left mouse button. **Normal**: Normal selection **Column**: All columns can be selected **Line**: All lines can be selected

Chapter 16
Working with Wafers

The functionality and principles of working with wafers will be explained, particularly the wafer layout.

The **Wafermap** representation and addressing is most convenient at the large scale wafer level. For more precise addressing tasks within a chip, the **Chipmap** representation contains a zoomed chip from a wafermap.

This chapter contains the following topics and tasks:

- Background information about the **Wafermap** and **Chipmap** windows (⇒ *Overview of Wafermap* on page 16-2),
- Performing an adjustment (⇒ *Performing an adjustment routine* on page 16-7),
- Software description of the **Wafermap** and **Chipmap** windows (⇒ *Wafermap window* on page 16-15,

  ⇒ *Chipmap window* on page 16-19),
- Explanation of mouse commands (⇒ *Mouse and keyboard commands* on page 16-41),
- Interaction with other modules (⇒ *Interaction with other modules* on page 16-42).

A Functional Description

1 Overview of Wafermap

A wafermap is defined and stored as an ASCII data file, which is normally indicated by the extension < *.wlo >. If the user is importing defect data via **KLA** or **Tencor** format files, using the **Import** window, the **Wafermap** layouts are automatically constructed from the imported data file and stored within the subdirectory **WAFER** of the user's directory.

The **Waferlayout** can be viewed. A background image can be used to represent the wafer holder, the mounts, clamps and position of the Faraday cup etc., for illustration and orientation purposes. In addition, all positions on the wafer holder can be addressed in UV coordinates.

The schematic below shows a typical waferlayout, with all the parameters which can be edited in the **Edit Waferlayout** subdialog - **Wafer** tab, (⇒ *Edit Waferlayout subdialog - Wafer tab* on page 16-21) and in the **Edit Waferlayout** subdialog - **Chip** tab, (⇒ *Edit Waferlayout subdialog- Chips tab* on page 16-26).

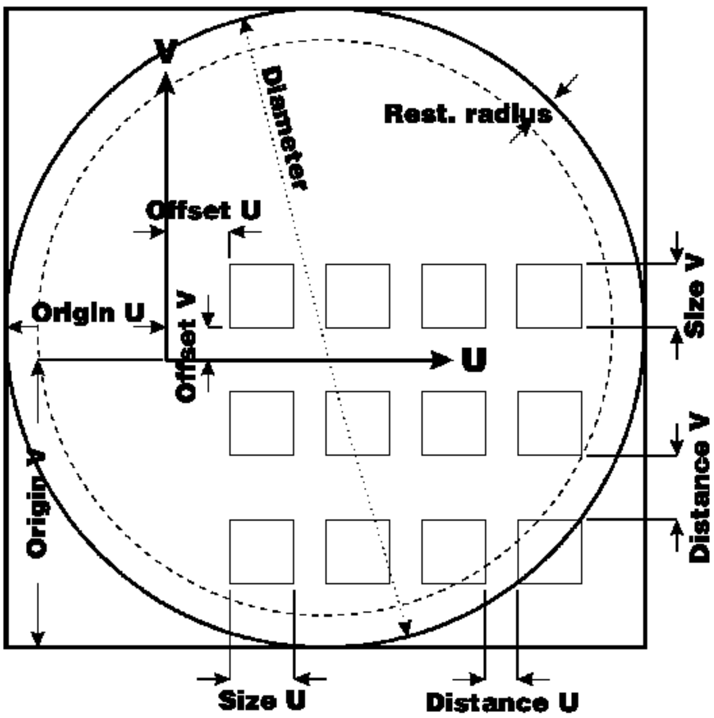


Figure 16-1: Schematic of waferlayout

The schematic explains graphically the meanings of the parameters to be entered (all metric values are given in mm). The outer diameter of the wafer can be entered. Chips which are not completely located within the wafer are not included in the wafermap layout, but they may be counted in the chip numbering sequence.

The restriction radius is the width of a circular margin encompassing the chip area. Chips which are located inside the circular margin, in whole or in part, are not drawn in the wafermap layout, but they may be counted in the chip numbering sequence.

The value for the origin of the UV (sample) coordinate system is the distance, in U and in V, between the UV origin and the lower left corner of a square enclosing the wafer circle. Typical values are either **0,0** (UV origin is the lower left corner of the square) or **Radius, Radius** (UV origin is the wafer center). Set the origin according to the conventions that you are using for the general UV coordinate system. You may, however, also adapt this origin later on, simply by defining corresponding offset values for the U and V coordinates.

Uncheck the unpatterned wafer to display the **Chip** tab, which will illustrate the Chiplayout.

A GDSII design structure can also be loaded and overlaid on either a single chip or over the full wafer. When full wafers are being used, the UV coordinates system is used for addressing the positions. The user can choose, for example, chip center as the position for the overlay of the GDSII structure. Zooming into the chip, the GDSII structure can be clearly observed within the wafermap.

2 Wafer patterning

The following letters can be used in the **Template** column of the Positionlist.

Table 16-1: Template column of Positionlist

Control element	Function
c	To drive to the UV chip coordinate locally, within the current die, relative to the lower left edge of the chip
C	To drive to the UV chip coordinate relative to the center of the chip.
s/S	To skip, do not drive to this position
d	To initiate a relative drive command
- UV	To invert the coordinates
e/E	To rotate eucentrically (rotation first, then drive XY to adjust)

2.1 Quick summary of wafer patterning

Use **Ctrl** + **left mouse** button to select the chip, then **right mouse** button to store it to the Positionlist (select a reference point).

After each selected chip, insert the corresponding patterning into the Positionlist via drag and drop.

Enter either **CUV** or **cUV** into the **Template** column of the patterning rows.

For a step by step guide to wafer patterning, please refer to the task, (⇒ *Setting up automated wafer patterning* on page 16-9).

3 Preadjustment principles

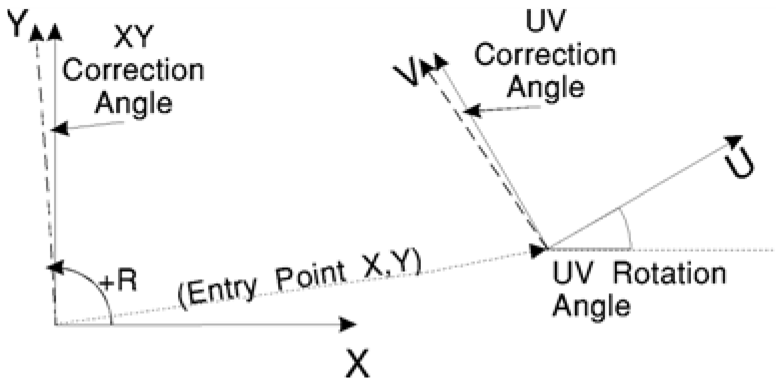


Figure 16-2: Principle of Preadjustment

The **Preadjustment** function calculates a new rotation angle and entry point for the UV coordinate system.

For this calculation, you must input values for some variables before performing any Preadjustment.

For calculation of the UV rotation angle, the program needs information about the orientation of the wafer on the sample holder and information about the orientation of the wafer in the waferlayout.

This information is taken from the variables **Module status➡[Adjust]➡Flat Angle X and ... /Flat Angle U**. Flat angle X defines the orientation of the alignment flat on the sample holder. It is the counter clockwise angle between the positive X direction and the direction of the alignment flat. Flat angle U is the corresponding angular difference between the positive U direction and the normal line on the flat in the waferlayout. This value is generally calculated from the waferlayout by the software itself.

Do not change the **FlatAngle U** manually.

The UV rotation angle is calculated using the following formula:

$$a = f_{fax} \cdot (a_{fax} + o_{fax}) + f_{fau} \cdot (a_{fau} + o_{fau}) + f_{rot} \cdot (a_{rot} + o_{rot})$$

with

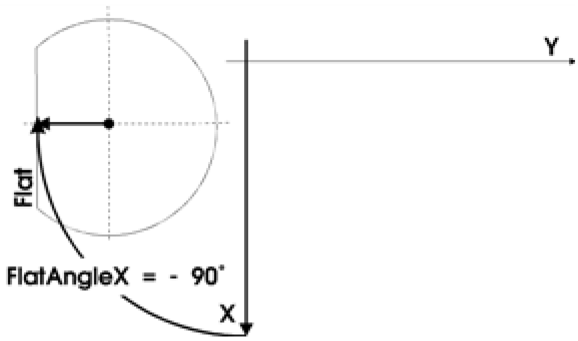


Figure 16-3: Calculation of **FlatAngle X**

a	: Calculated UV rotation angle
a_{fax}	: Flat angle X
a_{fau}	: Flat angle U
a_{rot}	: Position of motor R
f_{fax}	: Factor for flat angle X
f_{fau}	: Factor for flat angle U
f_{rot}	: Factor for position of motor R
o_{fax}	: Offset for flat angle X
o_{fau}	: Offset for flat angle U
o_{rot}	: Offset for position of motor R

These factors and offsets for calculation of the UV angle are accessed via **Module status➡[Adjust]**. They may be used to meet specific needs, but normally they should not be changed.

To calculate the UV entry point, set the variables **Module status➡[Adjust]➡ Prealign Center X** and ... **/Center Y** to define the X and Y coordinates of the wafer center position on the sample holder.

The required data for Preadjustment parameter calculation can be imported, for example, from the project file and from the waferlayout file. The project file may contain information that is specific to the stage or the sample holder, whereas the waferlayout file defines the characteristics of a certain wafer. Whenever a new wafer is placed on the sample holder, the correct wafer information for Preadjustment is loaded automatically by selecting a matching waferlayout file.

It is also useful to store the adjustment flags within the waferlayout file to automatically load these positions for the correct **3-Points Adjustment** routine. This can be activated within the database (.db) with the help of the Raith support team.

B Tasks

1 Performing an adjustment routine

▶ Open the **Wafermap** and apply the preadjustment routine.

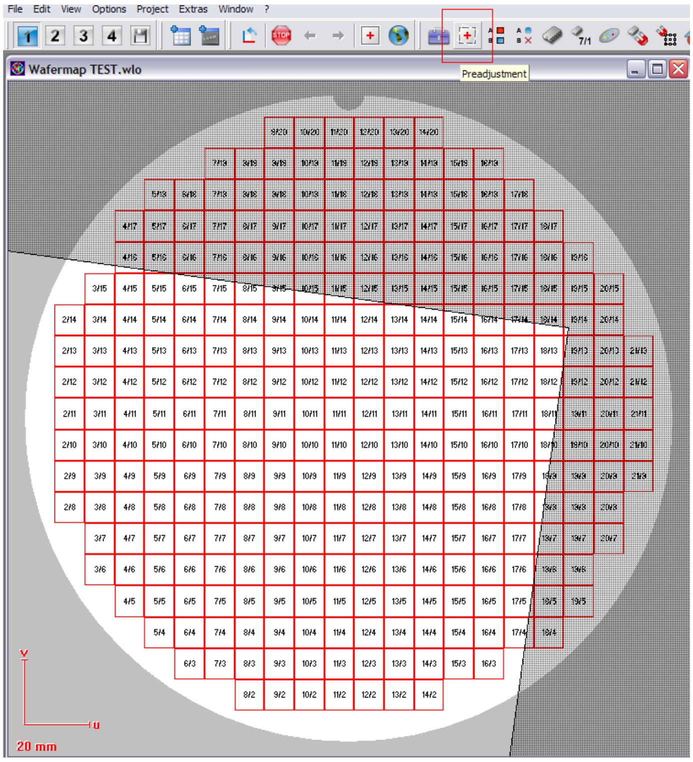


Figure 16-4: Visualization of preadjustment procedure within a Wafermap.

The procedure for determining the wafer center must be completed before the angular orientation can be measured, as described in the section **Deskew Marks**.

▶ Access the **Waferadjust** window by choosing the command **Edit➡Unpatterned wafer adjustment**.

▶ Set the 3 perimeter marks to those locations which can be easily accessed by your wafer stage. For this, you can vary in tandem the individual angular locations of the marks and the angle of the set.

Begin with the adjustment of the exact wafer center by using the perimeter marks.

▶ Choose mark #1 using the **Goto** button. The stage will automatically move close to the wafer boundary under the specified angular orientation.

▶ After the stage has reached its destination, move the stage manually so that you can see the wafer boundary crossing the center of the mark displaying the current stage position. For this step, it is not important that you take particular care with the angular orientation. If necessary, perform a backlash operation and readjust. Then choose the **Read** button.

- Alternatively, if an image is available, use the **Blue Flag** and drag it onto the wafer boundary within the image. The first checkbox acquires a checkmark, which indicates that this mark has been processed.
- Repeat these steps for mark #2 and mark #3.

Now perform the **Deskew** procedure, which is explained here for a wafer flat.

- Choose the **Goto** button of deskew mark #1. The stage will move to that location, based on the current adjustment.
- After the stage has reached its destination, move the stage manually so that you can see the wafer boundary crossing the center of the mark displaying the current stage position. For this step, it is not important that you take particular care with the exact position along the flat. If necessary, perform a backlash operation and readjust. Then choose the **Read** button.
- Alternatively, if an image is available, use the **Blue Flag** and drag it onto the wafer boundary within the Image. The first checkbox acquires a checkmark, which indicates that this mark has been processed.
- Repeat these steps for deskew mark #2.
- Choose **Adjust** to apply this new UV-Adjustment.

Please note that after this adjustment procedure, the addressed particles should be within a range of a few hundred microns or less. In some cases, a 3-points adjustment at visible particles or a post-adjustment can be useful to improve accuracy and speed of the review procedure. This depends strongly, however, on the precision of both the stage and the defect inspection tool, delivering the coordinate set (**Positionlist**).

Control element	Function
Adjustment	Command common to all UV windows, described in section **UV Windows**. Please note that the resulting mark location depends on the current catch status.
Measure	Command common to all UV windows, (⇒ *UV windows* on page 4-4).

2 Setting up automated wafer patterning

When setting up automated wafer patterning, it is important to be aware of the hierarchical structure:

1. Open a Positionlist and from there, open the assigned wafermap.
2. The wafermap contains the chip arrangement on the wafer and the link to the bitfailmap (memory chiplayout).

If you wish to create a fixed link between the Positionlist and the wafer-map, follow the steps above. When the Positionlist is open, open the wafermap with the tool bar, to open a new wafermap. This way, the wafermap will be linked to the Positionlist. This will also enable the catch mode (magnet button) within the wafermap.

- Open a Positionlist.
- Click on the **Wafermap** button in the tool bar to open a wafermap
- Move the stage to the chip on which you wish to perform the patterning.

Once the stage has been driven to the required chip, a task can be performed using the **Chip Coordinate** system. A Positionlist task is designated as using the Chip Coordinate system, by entering either a lower case '**c**' or a capital '**C**' in the **Template** column of the Positionlist. The stage will now not drive in the **Global** coordinates system, but instead the **Local** chip coordinate system, for the chip at which the stage is currently positioned.
If several tasks are to be performed on that chip, the lower case 'c' or upper case 'C' is also placed in the Template column of each task in the Positionlist. All tasks will be performed using chip coordinates, until a task is called up which has no letter 'c' in the Template column. The software will then revert to the **Global** coordinate system.

- Whenever one or several procedures should be carried out within a chip, for example patterning or mark recognition, either one task or several tasks can be grouped in the Positionlist, one after the other. Move the stage to any position on the chip on which the task should be performed. Once the current stage coordinates are anywhere within the required chip, any task in the Positionlist can be executed automatically. Chip addressing is performed via the wafermap.
- Open a wafermap and use the **Ctrl** key and the **left mouse** button to select one or multiple chips within that wafer, on which the procedures should be performed.
- For multiple chip selection, hold down the **Ctrl** key and click with the **left mouse** button on the first chip, then on the next chip and the next chip

and so on. The selected chips are displayed with a blinking frame to illustrate that they have been selected.

- Once all required chips have been selected, place the mouse cursor in any of the selected chips and press the **right mouse button**. Click on **Selection** and **Store to Positionlist**.

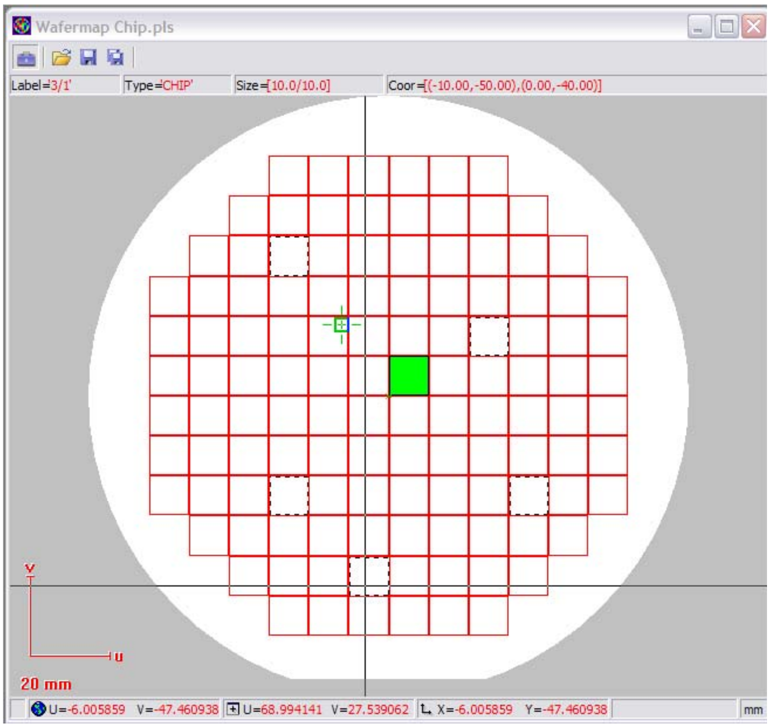


Figure 16-5: Chips selected on a wafermap

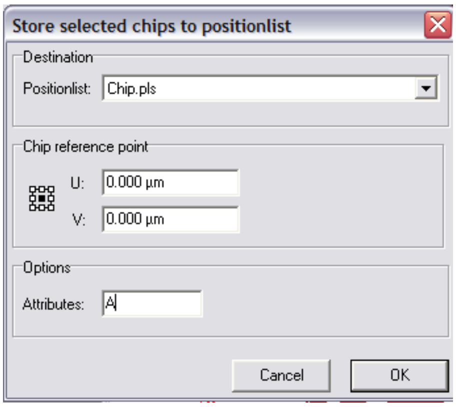


Figure 16-6: Store selected chips to a Positionlist

- A new dialog will open, in which you need to define the positions to which you wish to drive within the chips that you have selected e.g. the center or one of the edges.

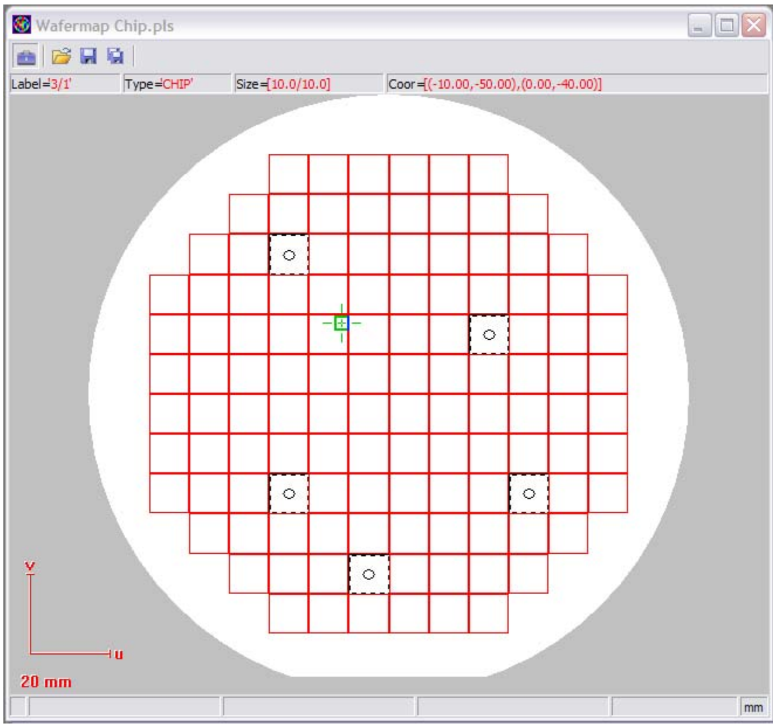


Figure 16-7: Defining the chip positions

It is recommended that you select the center position within the chips. If you select, for example, the left lower edge of the chip and the selected position is missed by only few nanometers, the stage position may be located in a different chip and the patterning would be carried out in the wrong chip.
Selecting the chip center as the position to which to drive makes your selection more secure.

- It is possible to select a different GDSII design structure for the patterning of each chip.
- You can select specific chips for a certain **GDSII design** structure and assign the same **Attribute A** to all of them. A different GDSII design structure can be chosen for another series of chips, which can be assigned **Attribute B** and so on. The chips are now marked with dots to illustrate that the stage will drive to them to carry out the selected task when the Positionlist is executed.
- At the moment we are dealing with a pure driving Positionlist. When this Positionlist is executed, one chip position after the other will be addressed.

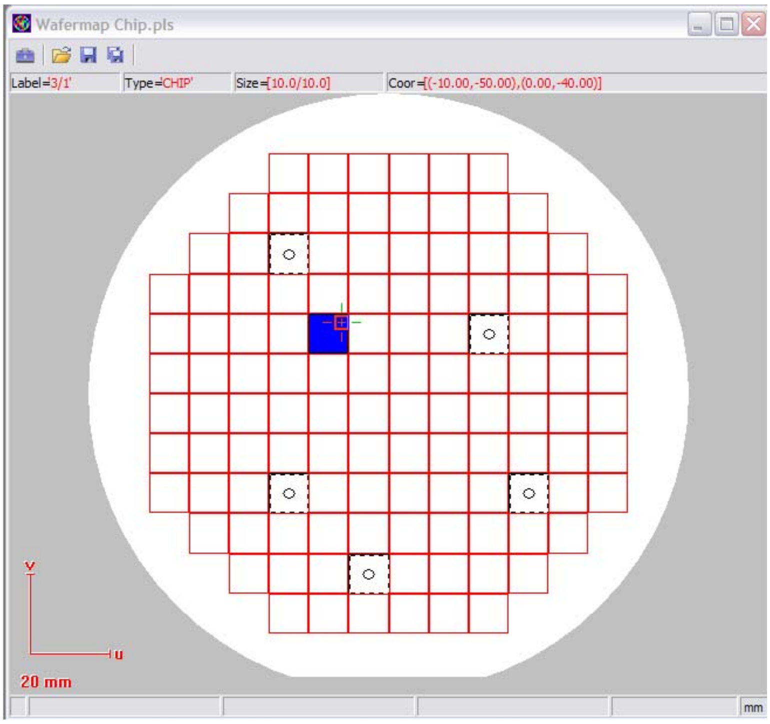
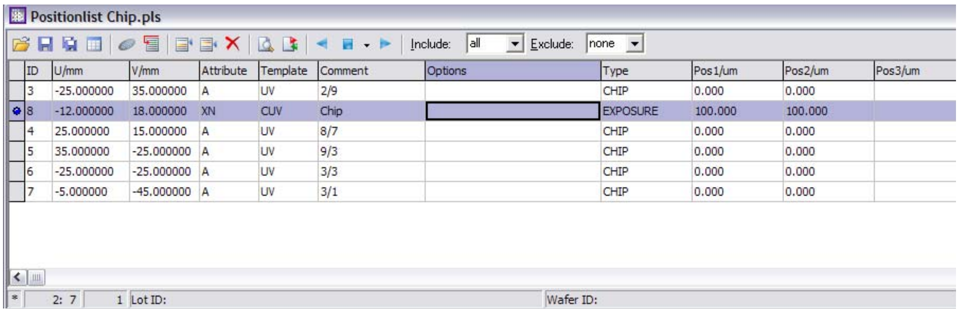


Figure 16-8: Inserting the chip positions into the Positionlist



ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type	Pos1/um	Pos2/um	Pos3/um
3	-25.000000	35.000000	A	UV	2/9		CHIP	0.000	0.000	
8	-12.000000	18.000000	XN	CUV	Chip		EXPOSURE	100.000	100.000	
4	25.000000	15.000000	A	UV	8/7		CHIP	0.000	0.000	
5	35.000000	-25.000000	A	UV	9/3		CHIP	0.000	0.000	
6	-25.000000	-25.000000	A	UV	3/3		CHIP	0.000	0.000	
7	-5.000000	-45.000000	A	UV	3/1		CHIP	0.000	0.000	

Figure 16-9: Positionlist with Drive commands and a local exposure

- The next step is to assign the patterning within each single chip, by dragging and dropping the GDSII design into the Positionlist. The GDSII design patterning task should now appear in the row beneath the drive command to each chip. The Positionlist will therefore now alternate as follows: Drive to chip 1, carry out the patterning on chip 1, drive to chip 2, carry out the patterning on chip 2 and so on.
- Prior to executing the Positionlist, it is of vital importance to change the entry in the **Template** column in the Positionlist. Add a capital '**C**', as the patterning will be performed in the local coordinate system, referenced to the center of the chip.

Use the capital '**C**' if you want to reference to the center of the chip. Use a small '**c**' if you wish to reference to the lower left corner of the chip.

When the capital '**C**' is used, then you need to enter negative U and V values to drive to the lower left corner of the chip.

When the small '**c**' is used, all UV values will be positive, as the UV origin, to which they are referenced, is the lower left corner.

The attribute **C** or **c** is an absolute chip drive command. For example, when a small c has been entered, the origin of the chip coordinate system is in the lower left corner and the chip drive command will drive to this position, when the corresponding row of the Positionlist has been executed. In the following row of the Positionlist, the GDSII design with its own design coordinate system will be applied to the current position and executed.

When a capital **C** is used, then the chip coordinate origin 0,0 will be set to the chip center and the GDSII design will be referenced to this origin.

Alternatively, the capital C can be used and in order to reference the GDSII structure to the lower left corner, an offset can be entered.

▶ The UV coordinates, which are relative, now need to be adapted. For example, if you enter '0', and a capital **C** has been entered, then the patterning would be placed at the origin 0,0, which is the chip center. If you enter -2,-2, then this offset would be applied for placement of the GDSII structure.

If 0,0 has been entered for the UV coordinates, as well as a capital **C**, then the center of the GDSII structure is equal to the center of the chip, no offset is applied.

Using the same principle, other tasks such as mark recognition to carry out a **Writefield Alignment**, or a **Patterning** task etc, can be inserted after the drive command in the Positionlist via the drag and drop function, thus enabling an automated procedure within a chip.

For example, the execution of a Positionlist can carry out the following automated procedure: Drive to a chip, carry out writefield alignment, including deskew and origin correction, using a 3-Points adjustment, then perform a patterning procedure, then drive to the next chip and repeat the same tasks again.

All the above can be carried out as an automated procedure. Various GDSII design structures can be applied to different chips. Any scripts, macros etc can be inserted into this automated procedure via drag and drop into the Positionlist.

The user can create one very long, complex macro, incorporating all the required actions, which will then only be inserted into one row of the Positionlist. Alternatively, a number of smaller, single macros, one row after the other as a group, can be inserted into different rows in the Positionlist and then executed.

As long as these single macros are grouped together after the drive command and each of the macros has either a small c or a capital C assigned in the **Attribute** column of the Positionlist, the stage will not be moved from its current position.

Whenever a task without a **c** or **C** in the **Template** column is executed, the stage will move to the next chip layout. When the **C** is included, the local chip coordinates are applied. Without the C in the template, the global coordinates will be used.

C Software Reference

1 Wafermap window

For the generation of customized wafermap layouts, choose the command **File➡Waferlayout**, as described below.

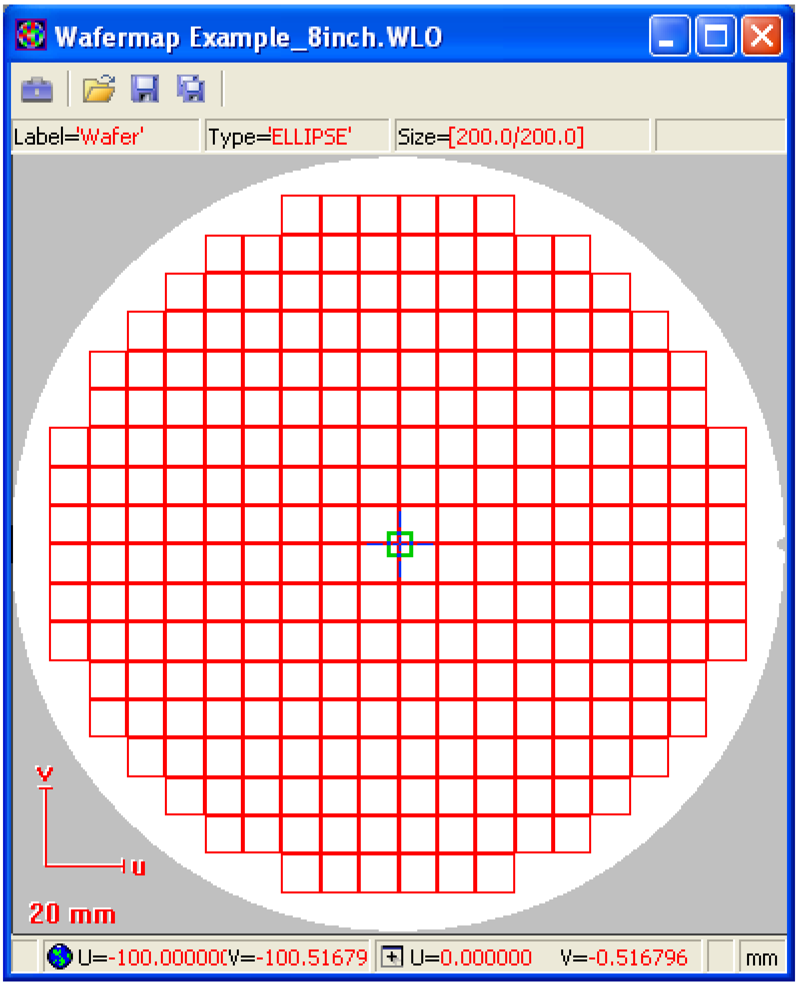


Figure 16-10: Example of a Wafermap

Use one of the following methods to open a Wafermap window:

- Choose the command **File➡New Wafermap**. In this case, a default wafermap is displayed. This map can be modified using the command **File➡Waferlayout** or replaced by an existing wafermap using the command **File➡Open**.
- Choose the command **File➡Open Wafermap**. In this case an existing wafermap will be loaded and displayed. This wafermap can also be modified or replaced by another one.

If you would like to assign a different wafermap layout to a Positionlist, please follow these steps:

Open and activate a Positionlist.

Use the **Open wafermap** button within the Positionlist to open the default assigned wafermap

Use the Open wafermap button within the assigned wafermap to open a different wafermap (with a different layout).

Save the assigned Positionlist.

The elements of the **Wafermap** window are as follows:

Control element	Function
Title Bar	The title bar shows the name of the loaded wafermap.
Scale	The scale serves to indicate graphically the size of your sample.
Limits	As an overlay, the software limits can be graphically indicated. Use the command **Extras**➡**Module status**➡**Wafermap**➡**Show limits** to enable this feature.
Tool bar	If a wafermap window is activated, additional buttons, menus and menu items are available within the menu bar, which will be explained in detail in the following sections.

1.1 Wafermap tool bar

Figure 16-11: Wafermap tool bar

Control element	Function
	To open the **Toolbox**
	To load an existing wafermap layout. This command replaces the currently active layout with the newly selected one.
	To save the wafermap

Control element	Function
	To save the wafermap to another file

1.2 Wafermap Toolbox subdialogs

The following sections explain the Wafermap Toolbox.

1.2.1 Toolbox

The Toolbox consists of the segment **Wafermap** tools. The toolbox is dynamic, changing according to the windows which are open.

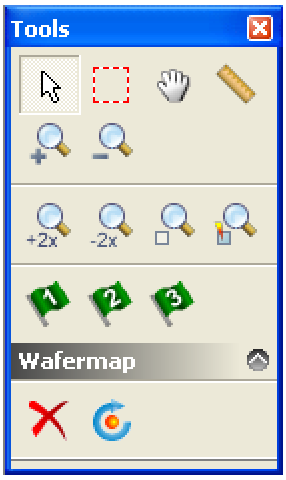
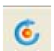


Figure 16-12: **Wafermap Toolbox**

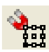
1.2.2 Wafermap subset

Control element	Function
	To delete selected chips
	To define the new Faraday position in the Waferlayout

1.3 Main tool bar Wafermap buttons

Whenever a wafermap is selected, the following additional buttons are displayed in the main tool bar.

Figure 16-13: Wafermap buttons in the main tool bar

Control element	Function
	To open the toolbox
	To perform Preadjustment, (⇒ *Performing an adjustment routine* on page 16-7)
	To define chip colors, (⇒ *Chip colors subdialog* on page 16-39)
	To define defect markers
	To open the chipmap
	To display chip text. To edit the chip text please refer to the **Waferlayout** section
	To show defect markers
	To select the type of Catch mode, (⇒ *Options menu* on page 16-37).
	To define the Catch offset
	To open associated CAD design, if a CAD design has been defined within the waferlayout.

2 Chipmap window

First activate a **Wafermap** window, then use one of the following to open a **Chipmap** window:

- Choose the command **File➡Open Chipmap** and select the chip of interest within the new dialog box.
- Double click on the chip of interest.

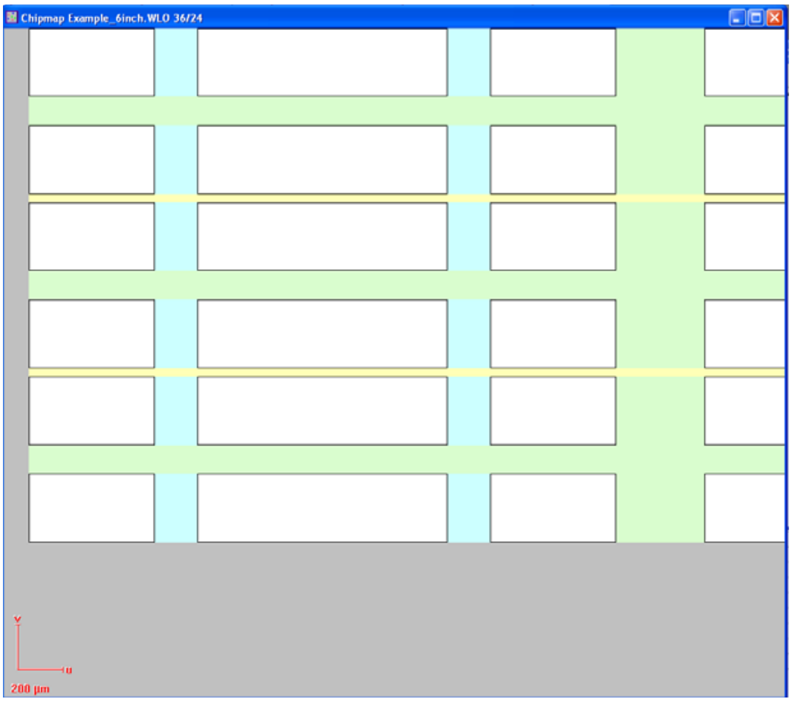


Figure 16-14: Example Chipmap with memory bitfailmap in the background.

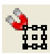
The elements of a chipmap window are:

Control element	Function
Title bar	Shows the name of the related wafermap and the number of the chip within this wafermap

2.1 Main tool bar Chipmap buttons

Whenever a chipmap is selected, the following additional buttons are displayed in the main tool bar.

Figure 16-15:Chipmap buttons in the main tool bar

Control element	Function
	To open the toolbox
	To zoom out of the chip level
	To zoom into the chip level
	To define the defect markers
	To set the type of Catch mode (⇒ *Options menu* on page 16-37).
	To define the Catch offset

3 Main menu

The main menu consists of the following.

3.1 File menu

The following menu items are available:

3.1.1 File menu - Wafermap

File menu item	Function
Open	To load an existing wafermap layout. This command replaces the currently activated layout with the newly selected one.
Save	To save the activated wafermap layout under its current name

File menu item	Function
Save as	To save the activated wafermap layout under a freely selectable name
Link Positionlist	To open a **Positionlist** and link it to the activated wafer-map. This relationship between wafermap and Positionlist is necessary to make use of the defect overlay feature, which is not used in standard lithography applications. This is just a temporary link, unlike the wafermap assign-ment. To assign a wafermap, the Positionlist has to be opened first, followed by the steps in the Task, (⇒ *Setting up automated wafer patterning* on page 16-9).
Open chipmap	Choose this command or double click on a chip within the **Wafermap** to open the related **Chipmap** window.
Waferlayout	Opens a dialog for editing the currently opened wafermap, according to your application. A dialog is preset with parameters taken from the active wafermap window. Edit-ing is described in the following section in the section **Defining waferlayouts**.
New Wafermap	To open a default wafermap
Open Wafermap	To open an existing wafermap

3.1.2 Edit Waferlayout subdialog - Wafer tab

Call up this dialog by selecting **Waferlayout** in the **File** menu.

A schematic of the waferlayout, showing the parameters in a drawing, can be viewed in Part A, **Overview of Wafermap**, (⇒ *Functional Description* on page 16-2).

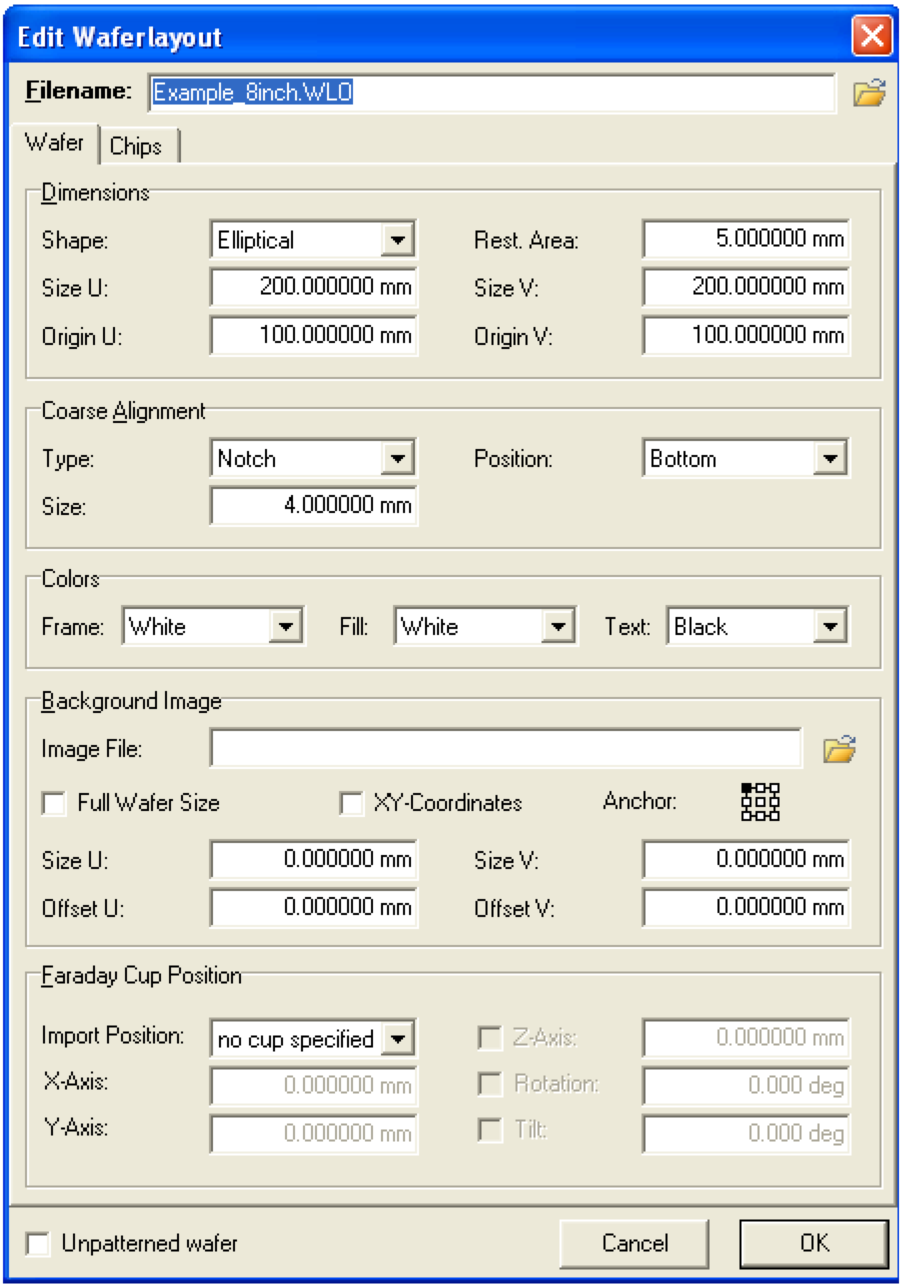


Figure 16-16: Dialog for editing a Wafermap -**Wafer** tab

Control element	Function
Filename	Enter a suitable name for your wafermap layout file. The extension <*.wlo> will be added automatically. The file will be stored in the subdirectory **WAFER** of the user's directory.
Unpatterned wafer	Check this checkbox if you wish to work with an unpatterned wafer. If an unpatterned wafer is selected, the **Chips** tab will no longer be available.

On the **Wafer** tab, the following controls can be used.

Control element	Function
Dimensions	The following dimensions can be selected.
Shape	Select an **elliptical** or a **rectangular** layout shape from this dropdown list box.
Rest. Area	Enter the width of the restriction area, i.e. the thickness of a margin restricting the chip area. Chips which are partly or completely located inside the restriction area are not drawn in the wafermap layout, but they may be counted in the chip numbering sequence.
Size U, V	Enter the outer diameter of your wafer in U and V directions.
Origin U,V	Enter the origin of the UV (sample) coordinate system, i.e. the distance from the UV origin to the lower left corner of a square enclosing the wafer layout. Typical values are either **0,0**, i.e. UV origin is lower left, or **Radius, Radius,** i.e. UV origin is wafer center. Set the origin according to the conventions you are using for the general sample (UV) coordinate system.
Coarse Alignment	The following alignment options can be selected.
Type	Use this dropdown list box to select one of the following orientation markers: **Major Flat, Square** or **Notch.**
Position	Use this dropdown list to select one of the following locations of the orientation marker: **Top, Right, Bottom** or **Left**.
Size	Enter the size of the orientation marker.

Control element	Function
Colors	The following colors can be selected.
Frame	Select the color of the wafer border.
Fill	Select the fill color.
Text	Select the text color.
Background Image	The background image can selected.
Image File	To select an optional background image of the sample holder, including mounts and clamps. The background images are stored in **<root system>➡Users➡All users➡Wafer.**
Full Wafer Size	Check this checkbox for the full wafer size. The background image will be stretched to the full wafermap.
X-Y Coordinates	Check this checkbox to set the coordinates to **XY** instead of UV. This is a useful feature when working with a sample holder background image.
Anchor	Choose an anchor by selecting a small black anchor in the drawing.
Size U,V or X,Y	Enter the size of the image. Select **Full wafer size** to stretch the image to the full Wafermap.
Offset U,V or X,Y	Enter the position of the image in the wafermap. Choose one of the small black quadrants to define the anchor point.
Faraday Cup Position	The Faraday Cup Position can be selected.
Import Position	Select the position for the Faraday cup from the drop-down list. The options are the cup positions specified, or current position. Select the **Pipette** (Read) button to read in the current coordinates.
X, Y, Z - Axis	The values can be either entered or read in by the **Pipette** (Read) button.
Unpatterned wafer	Check the checkbox if you wish to work with an unpatterned wafer. If an unpatterned wafer is selected, the **Chips** tab will no longer be available.

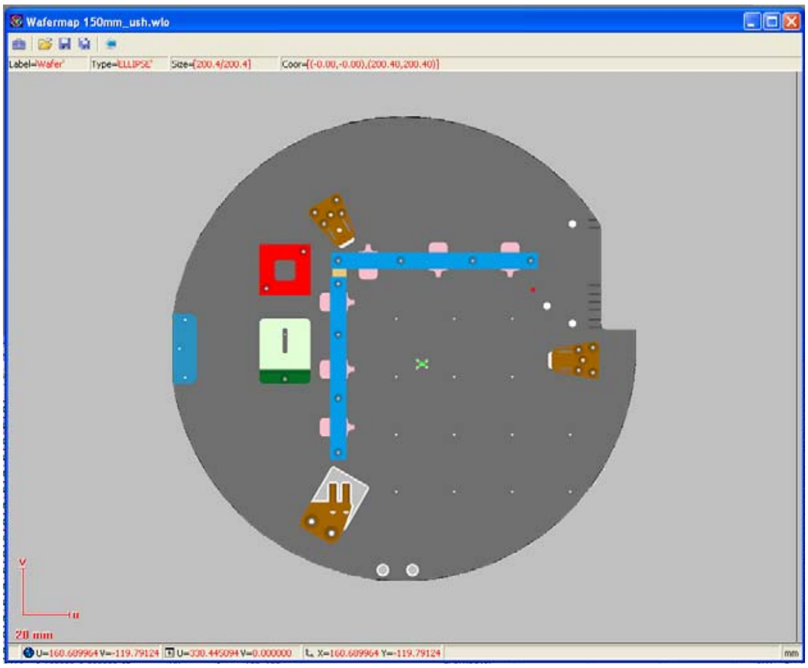


Figure 16-17: Universal Sample Holder background image applied

The user may define the position for the Faraday cup on the holder. Positions can be imported, from a list of positions in the **Stage Control** dialog. Alternatively, the user can read in the current position of the Faraday cup via the **Pipette** (Read) button. The user can also enter the exact position. Clicking on the  button in the Wafermap tool bar will define the new Faraday position in the Waferlayout.

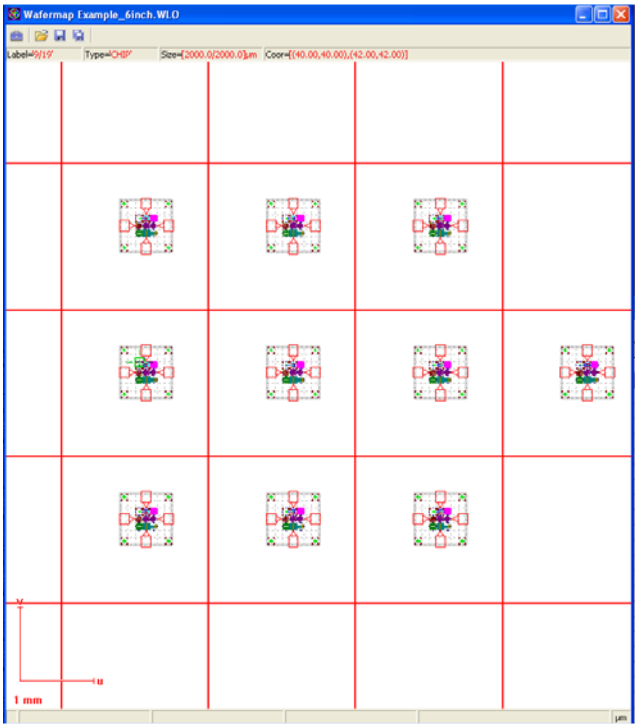


Figure 16-18: Wafermap example with CAD design overlay

3.1.3 Edit Waferlayout subdialog- Chips tab

A schematic of the waferlayout, showing the parameters in a drawing, can be viewed in Part A, **Overview of Wafermap**, (⇒ *Functional Description* on page 16-2).

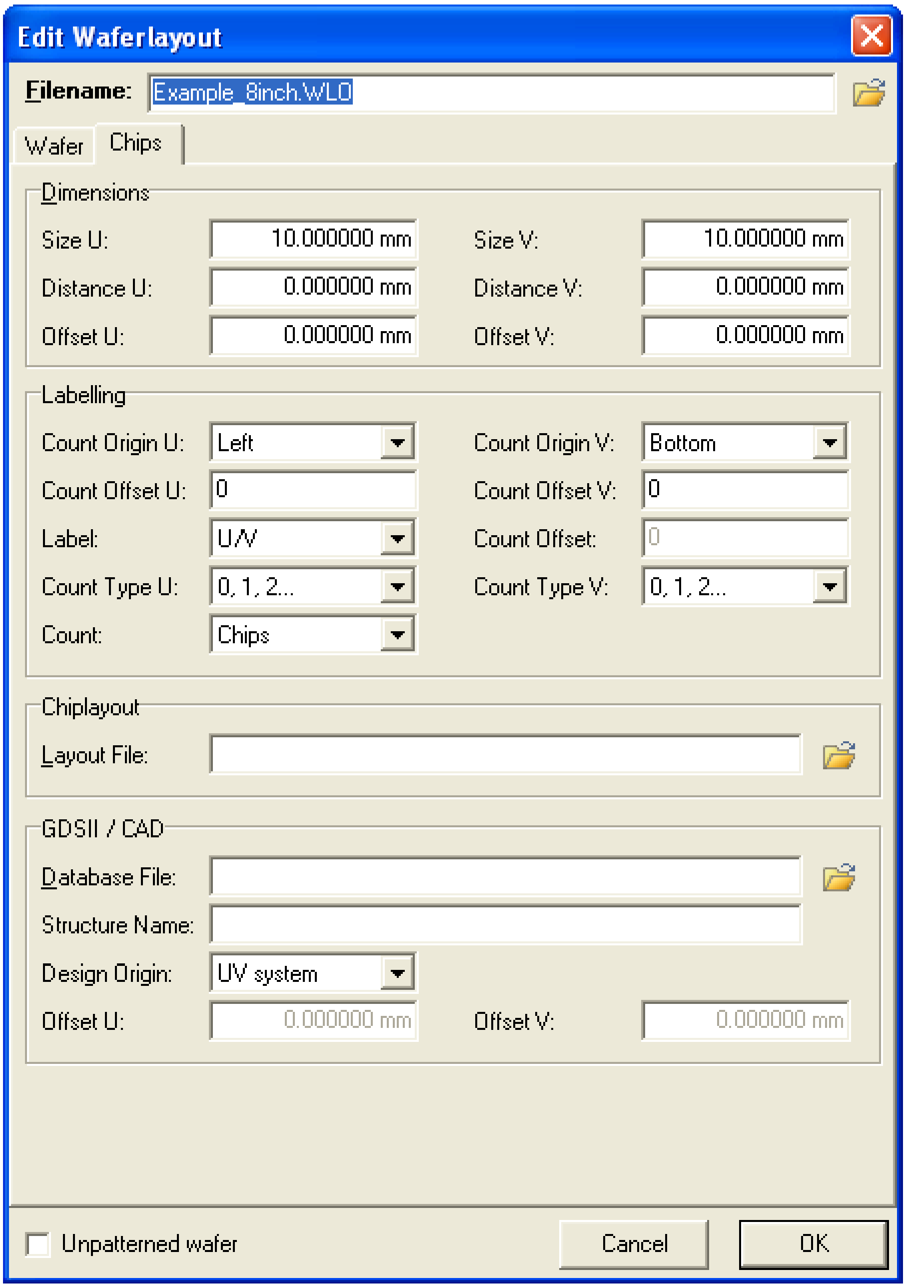


Figure 16-19: Wafermap -**Chips** tab

On the **Chips** tab, the following parameters are available.

Control element	Function
Dimensions	The following dimensions can be selected.
Size U, V	Enter the chip size with U = width and V = height.

Control element	**Function**
Distance U, V	Enter the horizontal (U) distance and the vertical (V) distance between two adjacent chips.
Offset U, V	Enter the distance between the lower left corner of the chip and the UV origin of the wafer.
Labelling	The following Labelling options can be selected.
Count origin U, V	Use these dropdown list boxes to select the starting point of the chip numbering scheme, i.e. to determine the chip counting direction. To increase chip numbers from left to right use **Left** and vice versa. To increase chip numbers from bottom to top choose **Bottom** and vice versa.
Count offset U, V	Enter the start number for the chip numbering scheme in the U direction and in the V direction. These text boxes are accessible for the numbering methods U, V and V, U only.
Label	Use this dropdown list to select the manner of chip numbering. **U/V** first matrix coordinate U, **V/U** first matrix coordinate V, Number single number.
Count offset	Enter the start number for the chip numbering scheme. This start number is valid for both matrix directions (U and V). This text box is only accessible for the numbering method **Number**.
Count Type U, V	Use this dropdown list to select between different alphanumerical numbering methods.
Count	Use this dropdown list to select the method of chip counting. Select either **Chips** or **Matrix**.
Chiplayout	Use this text box to assign an existing chip layout file to the wafermap layout. The specified chip layout file must be available in the subdirectory **CHIP** within the user's directory. It is used to overlay a substructure on the chipmap. If no chip layout is available or required, leave this text box free.
GDSII / CAD	GDSII or CAD can be selected.
Database File	Choose the GDSII database which is associated with the wafer and overlaid in each chip.
Structure Name	Choose the structure within the GDSII database.

Control element	Function
Design Origin	Use this dropdown list to select the entry point for the overlay.
Offset U, V	If the origin requires an offset, use these fields to define it.

3.1.4 Edit Wafer Element

To modify the waferlayout online on a chip-wise basis, point to the chip of interest, press and hold the **Shift** key and click the **left mouse button**. This operation opens the **Edit Wafer Element** dialog for the selected chip, which can be used to modify chip representations or to delete the complete chips from the wafer.

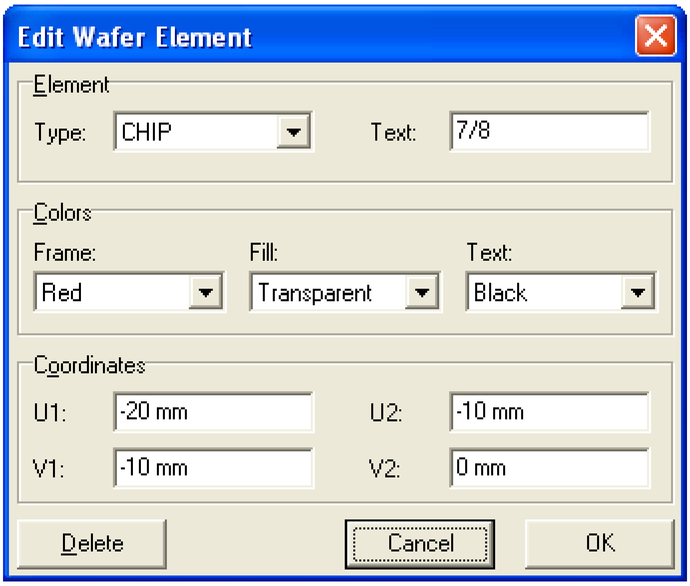


Figure 16-20: **Edit Wafer Element** dialog

Control element	Function
Element	Select the element **Type** from the dropdown list. Available types are **ELLIPSE, BOX, CHIP, TEXT**. The Text field is used only for the elements **CHIP** and **TEXT** but is ignored for **ELLIPSE** and **BOX**.
Type:	The following types can be selected.
ELLIPSE, BOX, CHIP, TEXT	Select the type of element that you want to create on the wafer from the dropdown list.
Text:	Select the chip position from the Text field

Control element	Function
Colors	Select **Frame** color, **Fill** color and **Text** color for the specified element from the dropdown lists. For the element type **TEXT**, fill color and text color are ignored, but the frame color is used instead. To enable chipwise color coding to be shown in the wafermap, go to **Extras**➡**Settings**➡**Wafermap**➡**Use default chip color** and select **color = OFF.**
Frame	Select a color for the Frame from the dropdown list.
Fill	Select a color for the Fill from the dropdown list.
Text	Select a color for the Text from the dropdown list.
Coordinates	The following coordinates can be selected.
U1, V2	Enter the coordinates of the lower left corner (**U1, V1**) of the specified element within the related text boxes.
U2, V2	Enter the coordinates of the upper right corner (**U2, V2**) of the specified element within the related text boxes.
Delete	To delete the specified Element completely from the wafer layout.

The element will be deleted immediately, without further query. An undo function is not implemented.

3.1.5 File menu - Chipmap

File menu item	Function
Open	Choose this command to load a chip layout file to be overlaid within the **Chipmap** window. Chip layout files (extension <*.clo>) are commonly used to define a Bitmap design.

3.2 Edit menu

The following menu items are available.

3.2.1 Edit menu - Wafermap

The following functions are available for the wafermap and chipmap.

Edit menu item	Function
Delete	To delete the selected chips.
Preadjustment	To set up and open the **Automatic Adjustment** dialog, showing information about the transformation parameters. Choose the **OK** button to perform the preadustment. These values are stored together with the wafermap file and are imported when the corresponding file is opened. Alternatively, you can select the button.
Unpatterned wafer adjustment	The adjustment procedure for an unpatterned or blank wafer requires a different approach, compared to that for other samples, as in this case there are no easily recognized regular structures, for which the sample coordinates are known. To overcome this disadvantage, the adjustment procedure determines the center of the wafer and its angular orientation by using its perimeter and its flat or notch. It requires no additional input of information such as precise wafer size or a set of known coordinates.
Adjustment	The adjustment marks can be set.
Set mark 1, 2, 3	To set the adjustment marks
Selection	The following selections can be chosen.
Select all	To select all
Clear selection	To clear the selection
Invert selection	To invert the selection
Store to Positionlist	To store the selection to the Positionlist
Load selection	To load a previously used selection
Save selection	To save the current selection
Delete selection	To delete the selection

Edit menu item	Function
Measure	To measure a distance, (⇒ *Edit menu* on page 4-19).

3.2.2 Automatic Adjustment subdialog

Call up this dialog by selecting **Preadjustment...** in the **Edit menu.** Alternatively, you can click the button in the main tool bar.

Automatic adjustments are special tools used in the semiconductor industry. It is essential that the wafer is placed inside the wafer holder every time at exactly the same position. If this is achieved, the **Automatic Adjustment** can be carried out, enabling a completely automatic adjustment of the wafer, not requiring the other **UV Adjustment** procedures.

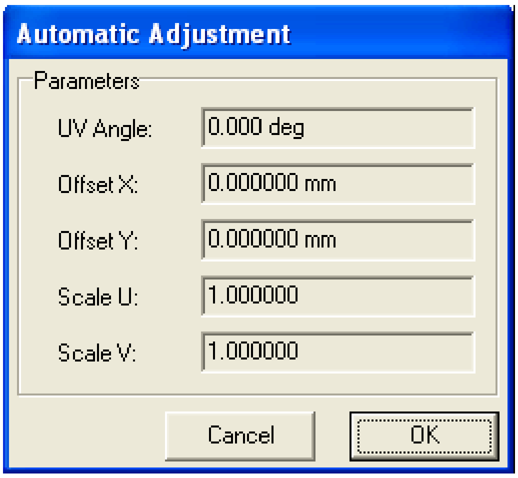


Figure 16-21: **Automatic Adjustment** dialog

Control element	Function
Parameters	The following parameters can be selected.
UV Angle	The UV angle will be calculated automatically using the flat angle XY.
Offset X, Y	Calculated X and Y offset are displayed. These values are defined in the **Setup Preadjustment** dialog, (⇒ *Setup Preadjustment subdialog* on page 16-40).

Control element	Function
Scale U, V	Scale factors for U and V axes are displayed. These values are defined in the **Setup Preadjustment** dialog, (⇒ *Setup Preadjustment subdialog* on page 16-40).

3.2.3 Waferadjust subdialog

Call up this dialog by selecting **Unpatterned wafer adjustment** in the **Edit** menu.

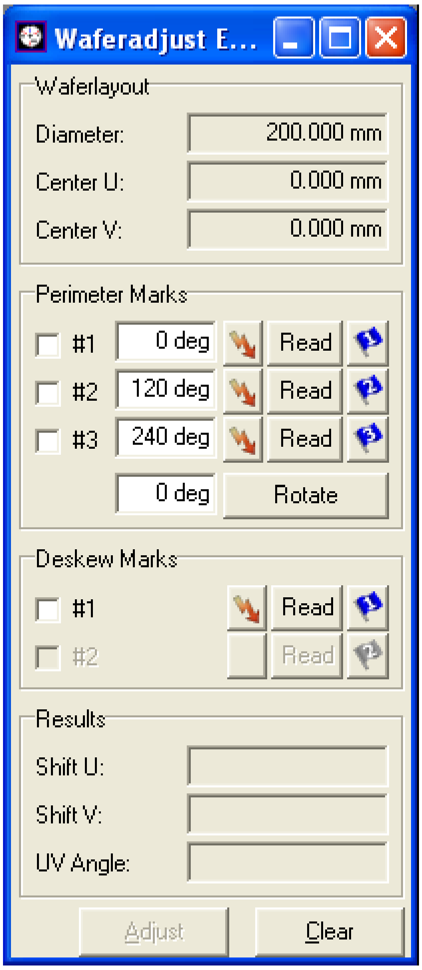


Figure 16-22: **Waferadjust** dialog

The corresponding dialog offers a toolbox and calculator for the adjustment routine. In parallel, there is an overlay displayed on the wafermap which indicates the selected sites within the adjustment procedure.

Control element	Function
Waferlayout	This section displays information already available to the program from the import of the wafermap, such as wafer diameter and wafer center in UV coordinates.
Diameter	The diameter of the wafer is displayed.
Center UV	The UV center is typically set to **Diameter**/2.
Perimeter Marks	This section is the interactive part which also allows configuration of the marks to be used for adjustment. Three perimeter marks must be set and used for determining the exact center of the wafer. The ideal locations of the marks are displayed in parallel on the wafermap as **P#1** to **P#3**. Each of these marks can be assigned with a dedicated angular position or all marks together can be rotated stepwise on the wafermap, if the default settings are not suitable for the instrument or application. Use **Go to** followed by the **Read** buttons (or the **blue flags** for reading coordinate data from UV windows) to read the current stage position when at the corresponding mark at the perimeter.
#1, #2, #3	The Perimeter Marks 1,2,3 must be set.
	The **Flash** button will drive the stage to the selected position.
Read	To read in the current coordinates
	The flags for perimeter marks 1,2,3 can be set. Using the blue flags enables the reading of the XY position in a UV window.
Rotate	To rotate the perimeter marks by a manually entered angle
Deskew Marks	This section allows measurement of the angular orientation of a blank wafer by aligning to the wafer notch or flat. From the information in the wafermap, the program automatically determines whether the wafer has a flat or a notch. The procedure differs in these two cases. In the case of a flat, two marks are used, which are along the flat of the wafer. In the case of a notch, only one mark is used. For both cases, only the angular orientation is calculated. Use the **Goto** buttons, followed by the **Read** buttons (or the **blue flags** for reading coordinate data from UV windows) to read the current stage position when at the corresponding mark at a flat or notch.

Control element	Function
Results	The results of the adjustment procedure become active after choosing the **Adjust** button. The button can only be applied if a complete set of angular and lateral wafer adjustment steps has been performed. The resulting UV offset value of the wafer center and the calculated rotation are displayed. The individual adjustment steps can be repeated and changed independently.
Shift U, V	The Shift in U and V will be displayed.
UV Angle	The UV Angle will be displayed.
Clear	To reset the coordinate transformation back to the identity transformation.

3.2.4 Edit menu - Chipmap

Edit menu item	Function
Cell navigator	This command is only available if a chip layout file was selected. Choose this command or use the hot keys **Ctrl + n** to open the memory cell navigator dialog box. After finishing navigation work, use the **Esc** key to close this dialog box. The cell navigator allows entry of drive commands digitally at the bit line and word line level, similar to the operation of the **Stage Control**.

The following commands are available for the chipmap cell navigation.

Table 16-2: Chipmap cell navigation commands

Commands	Functionality	Example	
<n>B	Relative drive command over <n> bit lines.	-1B	Moves 1 bit line down.
<n>b	Absolute drive command to bit line <n>.	120b	Moves to bit line number 120.
<n>W	Relative drive command over <n> word lines.	4W	Moves 4 word lines up.

Table 16-2: Chipmap cell navigation commands

Commands	Functionality	Example	
<n>w	Absolute drive command to word line <n>.	8191w	Moves to word line number 8191.

The small button on the right hand side of the text box allows the repeat of the last command.

3.3 View menu

The following menu items are available.

3.3.1 View menu - Wafermap

The following functions are specific to wafer and chiplayout.

View menu item	Function
Chip text	Choose this command or button to display the chip labeling. Alternatively, you can select the button.
Defect marker	Choose this command or button to display the defect positions. Alternatively, you can select the button.
Open waferlayout design	To open the associated GDSII design in GDSII viewer. This enables the GDSII overlay with a wafermap so that the user is able to view the GDSII design inside the chips in the wafermaps. To define the maximum number of chips showing the GSDII design, go to **Extras**➡**Settings**➡**Wafermap**➡**Max. GDSII overlays** and enter the maximum number. Please refer to the **Edit Waferlayout** dialog description of how to assign GDSII designs to the wafermap, (⇒ *Overview of Wafermap* on page 16-2).

3.3.2 View menu - Chipmap

View menu item	Function
Chip layout level	To increase or to decrease the chip layout level. The buttons are within the main tool bar, (⇒ *Main tool bar Chipmap buttons* on page 16-20).

3.4 Options menu

The following functions are specific to wafer and chiplayout. The other functions within the **Options** menu are explained in the chapter **UV Windows**, (⇒ *Options menu* on page 4-21).

Options menu item	Function
Catch mode	The catch mode defines how a location within a chip is addressed, for example when a chip is the target for a drag and drop action or driving to position via **Ctrl** + **right mouse** click. Three different modes exist: **None, Offset** and **Position**. Alternatively, you can select the button.
None	If the catch status is **None**, the exact pointer position is used for UV addressing.
Offset	If the catch status is **Offset**, the **Chip Catch Offset** setting with the chip is used. Choose the command **Chip Catch Offset** to define this offset.
Position	If the catch status is **Position**, the existing position located nearest to the mouse pointer is used. The Catch mode status is displayed also for the buttons **None**, and **Offset**.
Catch offset	The buttons can also be used to toggle between the modes. The **Chip Catch Offset** can also be defined using the button. Alternatively, you can select the button.
Chip colors...	To define the chip colors, (⇒ *Chip colors subdialog* on page 16-39). Alternatively, you can select the button.
Defect marker	The marker positions can be defined with an associated Positionlist, (⇒ *Tool bar* on page 9-12).
Setup preadjustment	The **Flat angle**, center of wafer X and Y as well as **Scale UV** can be defined, (⇒ *Performing an adjustment routine* on page 16-7). Alternatively, you can select the button.
Same aspect ratio	Check this checkbox to keep the aspect ratio constant.
Cursor grid	Define a cursor grid from the dropdown list.

3.4.1 Catch offset subdialog

Call up this dialog by selecting **Catch offset...** in the **Options** menu.
Alternatively, you can select the button.

The catch mode defines how a location within a chip is addressed, for example when a chip is the target of a drag and drop action. Three different modes exist: **None**, **Offset** and **Position**. If the catch status is **None**, the exact pointer position is used for UV addressing. If the catch status is **Offset**, the **Chip Catch Offset** setting with the chip is used.

The command **Chip Catch Offset** is used to define this offset. If the catch status is set to **Position**, the existing position of the associated Positionlist located nearest to the mouse pointer is used.

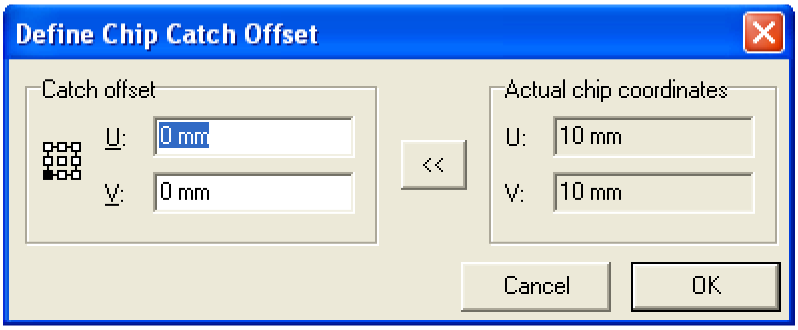


Figure 16-23: **Define Chip Catch Offset** dialog

Control element	Function
Catch offset	The Catch Offset can be defined.
U, V	Enter the U and V offsets in mm or define a black square in the diagram.
Current chip coordinates	The current chip coordinates can be defined.
U, V	The current chip coordinates are shown and can be transferred to the **Catch offset** via the arrow button.

3.4.2 Chip colors subdialog

Call up this dialog by selecting **Chip colors...** in the **Options** menu.
Alternatively, you can select the button.

Some chips containing one or more positions from an associated Positionlist can be color coded and thus give an overview of which kinds of defects are in which chips.

The button next to the chip can be used to modify the assignment of the chip color coding to the defect attributes within the wafermap layout in the related Positionlist.

A **Positionlist** must be opened and linked to the **Wafermap**, otherwise this command is not available.

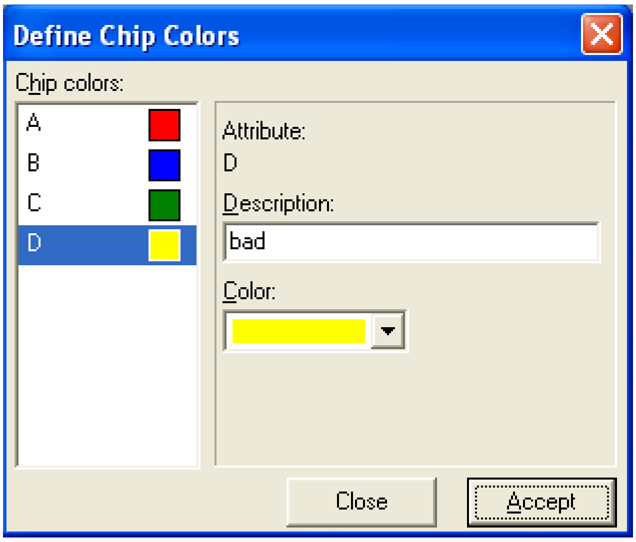


Figure 16-24: **Define Chip Colors** dialog

Control element	Function
Chip colors	The **Chip colors** list box shows the current assignment for all attributes of the associated Positionlist.
Attribute	The related attribute is displayed.
Description	Here you may enter a description of the attribute.
Color	The current assignment for all attributes of the associated Positionlist is shown. Select the current color assignment to be modified within that list box. Then select the new color to be assigned and choose **Accept.** In addition, you may enter a description of the attribute.

3.4.3 Setup Preadjustment subdialog

Call up this dialog by selecting **Setup Preadjustment...** in the **Options** menu.

Alternatively, you can select the button.

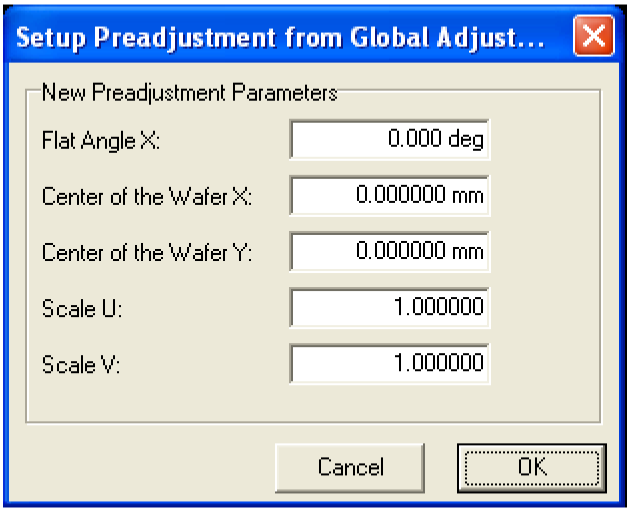


Figure 16-25: **Setup Preadjustment** dialog

Control element	Function
New Preadjustment Parameters	This dialog enables the definition of the parameter settings for the Wafer Preadjustment. These preadjustment parameters are normally taken from a KLA file. The most important parameter is **Flat Angle X**.
Flat Angle X	Angle between the major flat of the wafer and the X axis of the stage.
Center of the Wafer X,Y	Define the Center of the Wafer in X and Y.
Scale U,V	Define the Scale in U and V.

4 Mouse and keyboard commands

The table below gives a summary of all available mouse commands within an active wafermap window.

Table 16-3: Mouse and keyboard commands

Action	Key(s) + Mouse Button	Functionality
Single click	**Shift + left**	Opens the **Edit Wafer Element** dialog
Single click	**Ctrl + right**	Drives the stage to the selected position

Table 16-3: Mouse and keyboard commands

Action	Key(s) + Mouse Button	Functionality
Double click	**none + left**	Opens the related chipmap window
Wheel mouse	**none**	Zoom in and out of a wafermap

The table below gives a summary of all available mouse commands within an active chipmap window.

Table 16-4: Mouse and keyboard commands

Action	Key(s) + Mouse Button	Functionality
Single click	**Control + right**	Drives the stage to the selected position.
Double click	**none + left**	Opens the **Edit Position** dialog box for the defect located near to the selected position.

5 Interaction with other modules

A summary of drop operations involving a wafermap is listed below.

Table 16-5: Interaction with other modules

Source	Object	Target	Action
GDSII Database	Structure	Wafermap with associated Positionlist	GDSII structure is placed in a wafermap and stored in a Positionlist.
Positionlist	Position	Wafermap with associated Positionlist	Redefines the location of an object.

Table 16-5: Interaction with other modules

Source	Object	Target	Action
Scan Manager	Scan Definition	Wafermap with associated Positionlist	Scan task is placed in a wafermap and stored in a Positionlist.
Automation	Script or Macro	Wafermap with associated Positionlist	Macro file is placed in a wafermap and stored in a Positionlist.

Chapter 17
Height Control (option)

This chapter describes how to control the height of a sample surface via the **Height Sensing** software module in combination with a Laser Height Sensing tool (LHS) in order to keep the electron beam in focus.

The **Height Sensing** module is an optional module of the Raith NANOSUITE software and only attainable, if a Laser Height Sensing tool is installed to the chamber.

This chapter contains the following topics and tasks:

- Brief description of the hardware component
(⇒ *Laser Height sensing tool (LHS)* on page 17-2),
- User interface of the **Height Sensing** software module
(⇒ *Height Sensing software module* on page 17-2),
- Function principle of the height control
(⇒ *Functional Description* on page 17-2),
- How to synchronize the working distances of the electron or ion optics and the NANOSUITE software as well as setting up height control datasets
(⇒ *Basic Tasks* on page 17-6),
- How to monitor and evaluate the CCD peak detection
(⇒ *Monitoring CCD peak detection* on page 17-7)
- How to control the sample height and correct the focus depending on the current loaded sample and application
(⇒ *Controlling and correcting the sample height* on page 17-8),
- How to prepare the piezos for the height control in RAITH150 Two systems(⇒ *Preparing sample holders and piezos* on page 17-20),
- How to calibrate the CCD camera and the piezos
(⇒ *Calibrating the CCD camera* on page 17-16),
- Description of all functions and control elements of the **Height Sensing** software module
(⇒ *Software Reference* on page 17-25),

A Functional Description

This chapter gives a short overview of the hardware and software equipment of the Laser Height Sensing tool as well as the principle and methods of height control.

1 Laser Height sensing tool (LHS)

The Laser Height Sensing tool is an optional component for the Raith lithography systems RAITH150 Two and eLINE. The control and operation is exclusively carried out by the **Height Sensing** software module. The LHS consists of the following components, which are installed to the chamber:

- 8 Bit CCD single line camera containing an orthogonal CCD line with 1024 pixels.
- Laser gun providing a light signal, which will be reflected from the sample and detected by the CCD camera.

2 Height Sensing software module

The **Height Sensing** software module consists of the following main dialogs:

- **Height Control** to choose and activate a proper height control method for your sample or application.
- **CCD Control** for monitoring and adjusting the CCD peak detection as well as for calibrating the CCD camera.
- **Piezo Control** dialog for controlling and calibrating the piezos in the RAITH150 Two system. The dialog is only attainable for a Raith trained system expert.

(⇒ Figure 17-1 on page 17-3) shows the three main dialogs of the **Height Sensing** module.

For a detailed description of all control elements of the dialogs refer to sections

– (⇒ *Height Control dialog* on page 17-25),

– (⇒ *Piezo Control dialog* **(***RAITH150 Two)* on page 17-38),

– (⇒ *CCD Control dialog* on page 17-31).

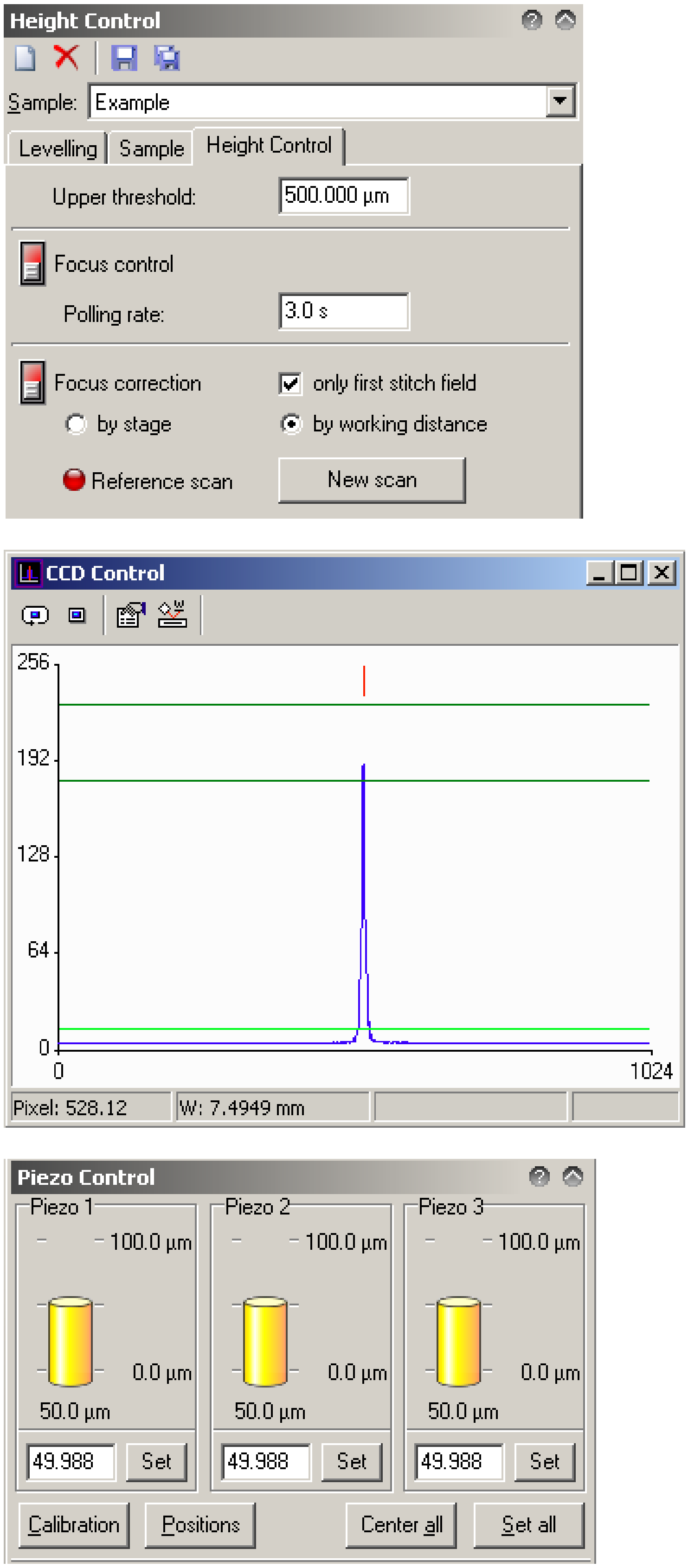


Figure 17-1: **Height Sensing** module main dialogs

3 Principle and methods of Height Control

In combination with a LHS the **Height Sensing** software module offers three methods to control the sample height and correct the focus. Generally, the height can be measured using the electron optic working distance or the LHS.

3.1 Height Control with the Height Sensing module

Height control and focus correction via the **Height Sensing** module can be performed as follows:

- Height control carried out exclusively by the LHS. The height control can be performed as:
  - a continuous focus correction during the operation of the system (Focus control mode) (⇒ *Using Focus Control mode* on page 17-8),
  - a focus correction during a patterning process or Positionlist scan (Focus correction mode) (⇒ *Using Focus Control mode* on page 17-8).
- Height control by triple-point sample leveling (⇒ *Triple-point leveling of the sample surface* on page 17-13).

  This kind of height control subjects to the topography of the sample surface. It is based on a plane function calculated from three points on the sample surface. It is suitable for flat samples like wafers or masks (⇒ *Triple-point leveling of the sample surface* on page 17-13).

  In RAITH150 Two systems additionally by fine leveling the piezos. The correction takes place in between the expanding range of the piezos of 100 µm. The fine leveling can be supported by using the CCD camera.

3.2 Functional principle of the LHS

The measuring principle of the LHS is based on an orthogonal CCD line in combination with a laser beam. The CCD line consists of 1024 pixels, i.e. the height of a sample surface is indicated as a pixel value. The pixel value corresponds to a working distance value in mm. To keep reliable data, the CCD camera must be calibrated respectively (⇒ *Calibrating the CCD camera* on page 17-16).

The LHS uses a triangulation method to determine the height of the sample: The laser beam is projected to the sample surface. The beam is reflected back to the CCD camera, focused through a receiver lens and projected onto the light receiving elements of the CCD line inside the camera. The LHS detects the peak value of the laser light distribution for each pixel and determines the precise sample position. As the position of the sample changes relative to the camera, the position of the reflected beam changes on the CCD line.

The position of the CCD peak and the rest of the detected laser light distribution will be displayed in the **CCD Control** dialog for monitoring purposes (⇒ *Monitoring CCD peak detection* on page 17-7).

3.3 RAITH150 Two specification

To compensate machining tolerances the RAITH150 Two is equipped with three piezos for fine-leveling all sample holders, especially the electrostatic chucks and mask holders. The piezos are mounted on the stage at three supporting points and provide the only connection between the stage and the sample holder.

Each piezo has an elongation range of 100 µm and can perform a fine leveling of the sample holders in between this range. To keep a reliable performance, the piezos must be pre-levelled and calibrated periodically (⇒ *Preparing sample holders and piezos* on page 17-20).

B Tasks

This chapter describes the basic and most important tasks for controlling the height via the LHS.

1 Basic Tasks

First, basic tasks will be explained.

1.1 Adjusting focus and working distance (Adjust W)

As a precondition for the reliable use of the height control functions of the LHS, the working distance measured by the electron or ion optics (focus) and the working distance calculated by the NANOSUITE software must be synchronized (**Adjust W** procedure).

Proceed as follows to perform an **Adjust W** procedure:

- Load a sample.
- Focus to any point on the sample.
- Change to the **Adjustments** module via control bar .
- Open the **Adjust UVW (Global)** dialog and change to the **Adjust W** tab.
- Read the current working distance of the electron or ion optics with a click on .

The system will prompt the user to adjust the column focus.

- Adjust the focus via live image acquisition of the sample surface and click on **OK** to confirm the prompt message.
- In the **Adjust UVW (Global)** dialog, click on **Adjust** to set the W axis to the value of the measured working distance.

The working distance and NANOSUITE software are synchronized and you can activate the height control via the LHS.

The W axis coordinates are used for the CCD calibration procedure and therefore must correspond to the real working distance value. So it is important to perform a W adjustment before using the LHS.

1.2 Setting up Height Control datasets

A height control dataset serves to store and recall all height control settings for a sample or special application.

- Change to the **Height Control** dialog.

- Click on to set up a new height control dataset for the current sample or application.

  The **New Sample** dialog will be opened.

- Type in a name for the dataset by using a significant name for your sample or application.

- Click on **OK** to set up the dataset.

  The height control dataset is stored with the current settings of the **Height Control** dialog. For use and editing, it can be selected of the **Sample:** combo box. Any change of the settings can be added to the dataset by clicking on .

2 Monitoring CCD peak detection

For the use of the LHS it is presumed that the CCD peak detection works reliable. Therefore, it is possible to monitor the peak detection. In case of a misalignment, the CCD camera must be calibrated.

To monitor the peak detection, the laser light distribution can be read out periodically and will be displayed in the **CCD Control** dialog graphically. By default, the **CCD Control** dialog will be displayed as minimized window in the desktop area of the NANOSUITE application. The CCD peak will be displayed as a red mark (⇒ Figure 17-2 on page 17-8).

Proceed as follows to monitor and evaluate the CCD peak detection:

- Maximize the **CCD Control** dialog.

- Click on to activate the continuous CCD polling.

  The polling will be performed at pre-defined intervals – e.g. every 3.0 s. You can define the polling interval via **Height Control** dialog ➡ **Height Control** tab ➡ **Polling rate**.

  The CCD peak detection will be displayed as a frequency function of the laser light distribution:

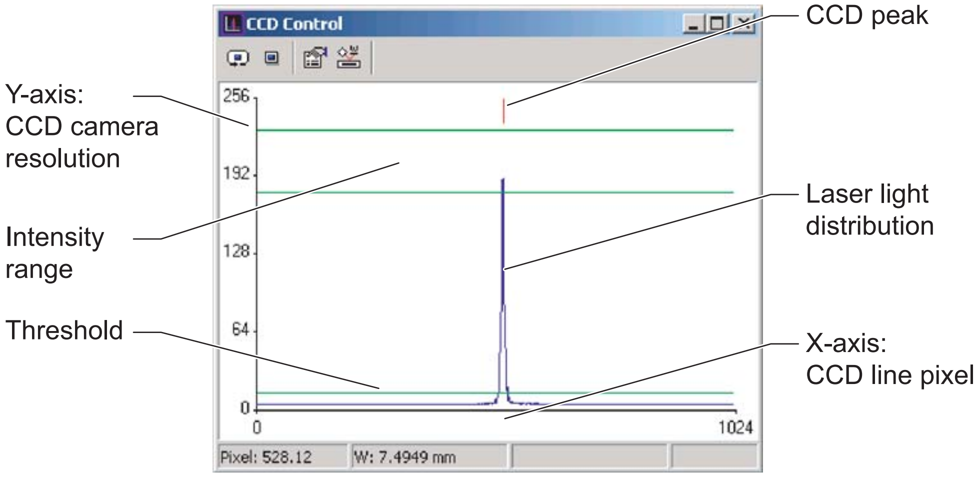


Figure 17-2: **CCD Control** dialog showing a correct laser line distribution and CCD peak

The CCD peak is a calculated value and represents the current position of the sample on the X-axis. The frequency distribution of the laser light distribution indicates all reflected light signals of the laser beam. It should be as thin as possible. If there is a broader scattering around the center axis, the CCD camera has to be calibrated.

In case of broader scattering or any significant deviation, consult the system expert to adjust the settings of the CCD camera.

---

3 Controlling and correcting the sample height

The following sections will explain how the sample height can be controlled and corrected.

3.1 Using Focus Control mode

In the **Focus Control** mode, the working distance will be corrected continuously during the operation of the system.

The focus control will be carried out by the LHS and is suitable for sample inspection purposes and manual image acquisition.

When using the **Focus Control** mode the working distance has to be in the range of W = 9 mm to W = 11 mm.

Proceed as follows to perform a continuous focus control:

- Load a sample and perform an **Adjust W** procedure (⇒ *Adjusting focus and working distance (Adjust W)* on page 17-6).

▶ Change to the **Height Control** dialog and select a height control dataset of the **Sample:** combo box or set up a new one (⇒ *Setting up Height Control datasets* on page 17-6).

▶ Open the **Height Control** tab:

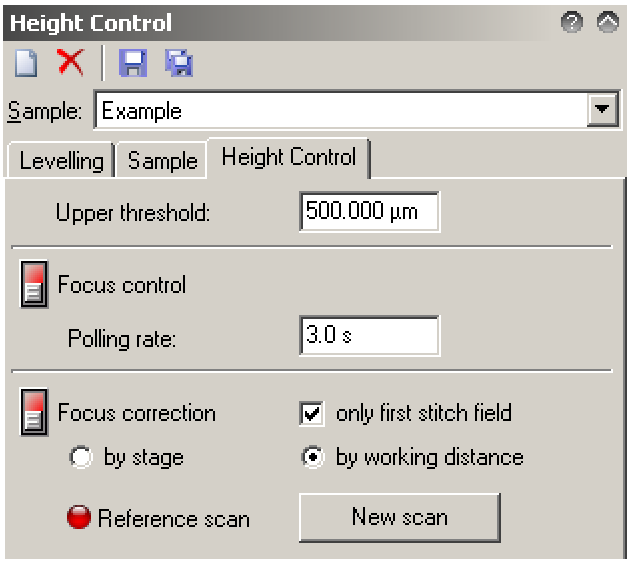


Figure 17-3: **Height Control** dialog – **Height Control** tab

▶ In the **Upper threshold** input field, enter an upper threshold for the working distance. Up to this value, the correction of the focus will be performed.

▶ Focus the beam anywhere on your sample. Take care the working distance corresponds to the CCD calibration range (⇒ *CCD Calibration Range subdialog* on page 17-35).

▶ Maximize the **CCD Control** dialog and check the CCD peak detection.

▶ Change back to **Height Control** dialog ➡ **Height Control** tab.

▶ In the **polling rate** input field enter an interval to which the peak detection is to be read from the CCD line.

A selected polling rate of e.g. 3.0 s results in a correction of the focus after every 3.0 s. The minimum time interval and increment is about 1.0 s.

▶ Click on **Focus control**.

From now the focus will be corrected continuously during any operation like stage movement via the joystick. During a patterning process, the **Focus Control** mode will be disabled automatically.

3.2 Using Focus Correction mode

In the **Focus Correction** mode, the working distance will be corrected during a patterning process or Positionlist scan after the stage was moved from one position or one stitch field to another one.

At a single-field patterning this correction will be executed after driving to the destination directly before starting the patterning. At a multi-field patterning this correction will be executed in front of every stitch field.

The focus correction will be carried out by the LHS and is suitable for applications with long patterning times.

When using the **Focus Correction** mode the working distance has to be in the range of W = 9 mm to W = 11 mm.

3.2.1 Using Focus Correction mode during Patterning

Proceed as follows to perform a correction of the focus during a patterning process:

- Load a sample and perform an **Adjust W** procedure (⇒ *Adjusting focus and working distance (Adjust W)* on page 17-6).
- Change to the **Height Control** dialog and select a height control dataset of the **Sample:** combo box or set up a new one (⇒ *Setting up Height Control datasets* on page 17-6).
- Open the **Height Control** tab:

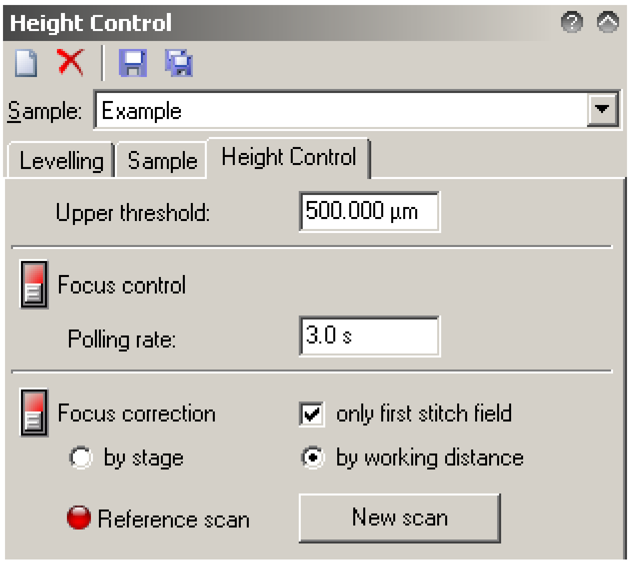


Figure 17-4: **Height Control** dialog – **Height Control** tab

- In the **Upper threshold** input field, enter an upper threshold for the working distance. Up to this value, the correction of the focus will be performed.

- Select how the correction is to be executed:
  - **by stage**: The system will adjust the Z-level of the stage.
  - **by working distance**: The system will adjust the working distance of the electron or ion optics.
- Drive close to a characteristic position for patterning on your sample.
- Maximize the **CCD Control** dialog and check the CCD peak detection. If necessary, activate the **Automatic peak control** and adjust the **Threshold** to cut of the ground noise of the CCD peak detection (⇒ *CCD Control Parameters subdialog* on page 17-32).
- Change back to **Height Control** dialog ➡ **Height Control** tab and click on **Focus correction**.

  The **Focus Correction** mode is activated.
- Focus the electron beam at the desired height. Take care not to expose any position where a patterning will be placed later on.
- Click on **New Scan** to take a reference scan of the current position.

  The corresponding LED illuminates green to indicate an active reference scan.
- Start the patterning process.

  The focus will be corrected during the patterning process.

3.2.2 Using Focus correction during Positionlist scan

Proceed as follows to perform a correction of the focus during a Positionlist scan:

- Load a sample and perform an **Adjust W** procedure (⇒ *Adjusting focus and working distance (Adjust W)* on page 17-6).
- Enable the height control function via **Extras** ➡ **Settings** ➡ **Drive Scan** ➡ **Enable Height Control**.
- Generate a new Positionlist or open an existing one via the main menu bar **File** ➡ **New positionlist**/**Open positionlist...**
- Change to the **Height Control** dialog ➡ **Height Control** tab.

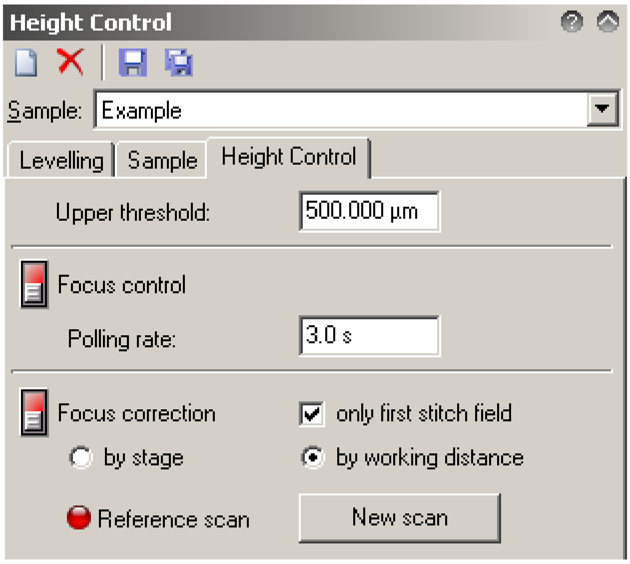


Figure 17-5: **Height Control** dialog – **Height Control** tab

- In the **Upper threshold** input field, enter an upper threshold for the working distance. Up to this value, the correction of the focus will be performed.
- Select, how the focus correction is to be executed:
  - **by stage**: The system will adjust the Z-level of the stage.
  - **by working distance**: The system will adjust the working distance of the electron or ion optics.
- Click on **Focus correction**.

  The **Focus Correction** mode is activated.
- Focus the electron beam at the desired height. Take care not to expose any position where a patterning will be placed later on.
- Click on **New Scan** to take a reference scan of the current position.

  The corresponding LED illuminates green to indicate an active reference scan.
- Start the Positionlist scan.

  The focus will be corrected during the positionlist scan.

3.3 Triple-point leveling of the sample surface

For flat surfaces like a mask or a wafer the working distance may vary with the XY-coordinates of the stage due to a non parallelism between the surface of the sample and the linear XY-bearings. The triple-point levelling procedure offers the possibility to make the sample surface parallel to the XY-movement. For this, a plane function will be calculated from three defined points on the sample surface.

This procedure is only useful in case of real flat surfaces like a mask or a high quality wafer on an electrostatic chuck. Otherwise the calculation of the sample surface can differ significantly from the actual sample surface.

3.3.1 Triple-point leveling without piezos

Proceed as follows to level the sample surface by defining three free points:

▶ Change to the **Height Control** dialog and select a height control dataset of the **Sample:** combo box or set up a new one (⇒ *Setting up Height Control datasets* on page 17-6).

▶ Open the **Sample** tab:

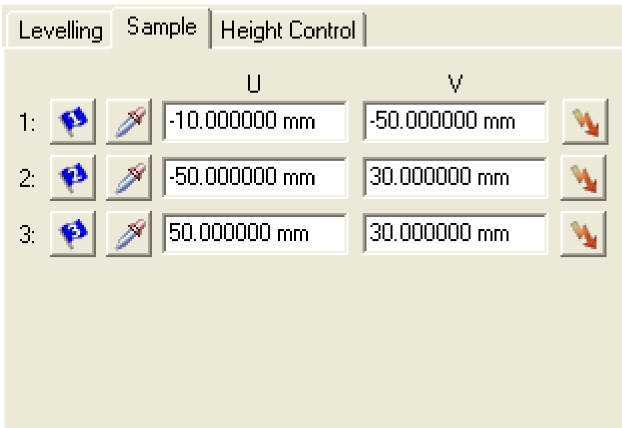


Figure 17-6: **Height Control** dialog – **Sample** tab

▶ Define the position of the three leveling points by doing one of the following:

- Drag and Drop one blue flag in any UV-window, e.g. an image, GDSII viewer or editor, or
- drive to the position and read the actual UV coordinates by clicking on , or
- enter the UV-coordinates in the corresponding input fields.

The higher the distance between the three leveling points, the higher the accuracy of levelling of the sample surface.
Furthermore it is recommended to choose positions without high topography to guarantee a high quality of the peak detection of the CCD camera.

▶ Change to the **Levelling** tab:

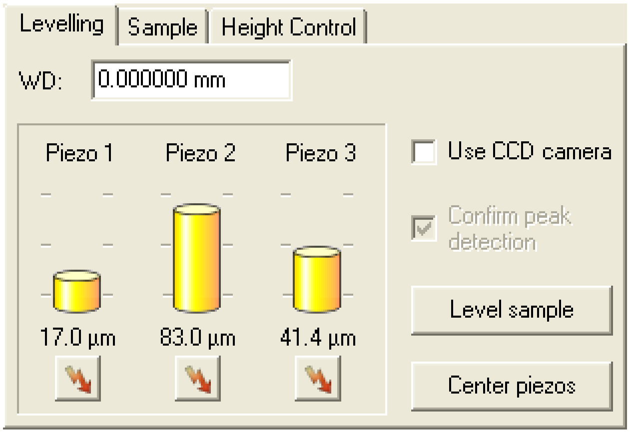


Figure 17-7: **Height Control** dialog – **Levelling** tab

▶ In the **WD:** input field, enter the working distance you want to use after the leveling procedure is finished. The stage will be moved on its Z-axis correspondingly.

If no working distance is defined, the Z-axis will not be moved after the leveling procedure.

▶ Click on [save icon] to store the working distance in the selected height control dataset.

▶ Enable **Use CCD camera** to get the height information of the stage from the LHS.

Otherwise, the system will ask to focus the microscope in order to read out the working distance instead of getting the height information from the LHS.

If it is not sure whether the CCD peak detection will work reliable at the three leveling points, enable **Confirm peak detection**. If it is safe, leave it disabled and the leveling procedure will run without any further user interaction.

▶ Click on **Level sample**.

The triple-point leveling procedure will be performed.

3.3.2 Triple-point leveling using piezos (RAITH150 Two)

When leveling the sample surface using the piezos, it is recommended to choose the leveling points above the piezos, in order to get the highest accuracy.

- Change to the **Height Control** dialog and select a height control dataset of the **Sample:** combo box or set up a new one (⇒ *Setting up Height Control datasets* on page 17-6).
- Open the **Levelling** tab:

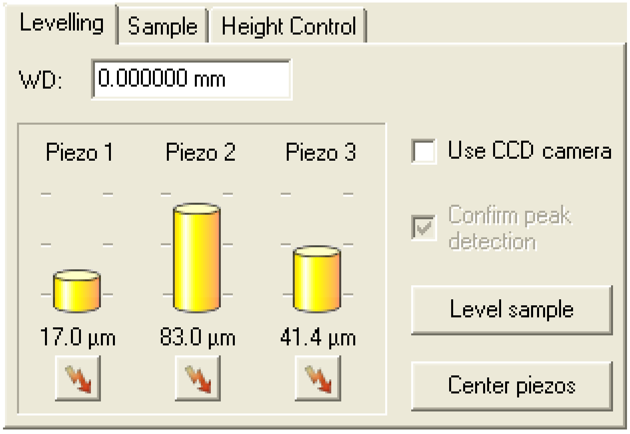


Figure 17-8: **Height Control** dialog – **Levelling** tab

- In the **WD:** input filed, enter a working distance you want to use after the leveling procedure is finished. The stage will be moved on its Z-axis correspondingly.
- For every piezo, click on to drive the stage to the corresponding piezo position.

  The value of the current piezo elongations will be displayed under the three column symbols respectively.
- Click on **Center piezos**.

  The piezos will be set to the middle of their elongation range of 50 µm.

Set the piezos always to the middle of their elongation range. From this position is enough space for correction in both directions – upwards and downwards.

- Click on **Level sample**.

  The piezos will be leveled.

4 Calibrating the CCD camera

The calibration of the CCD camera has to be performed exclusively by the local system expert.

As a precondition for the calibration of the CCD camera, the working distance measured by the electron or ion optics (focus) and the working distance calculated by the NANOSUITE software must be synchronized (**Adjust W** procedure).
Otherwise, there will be differences in the height measuring and the calibration procedure can fail.

The CCD calibration procedure can be used to recalibrate the interpretation of the CCD peak position shift. This means a redefinition of the relationship between the peak position shift and the corresponding height changing of the stage.

During the calibration procedure, the detected CCD peak position in pixel will be adjusted to the height of the stage in mm.

STEP 1: Making preparations

- Load in a flat and reflective sample mounted somewhere on the stage.

The height variation of the sample surface must be less than 10 µm when driving 5 mm in any XY-direction.

- Focus on the sample and perform an **Adjust W** procedure (⇒ *Adjusting focus and working distance (Adjust W)* on page 17-6).

STEP 2: Adjusting the peak detection

- Maximize the **CCD Control** dialog in the desktop area of the NANOSUITE application.
- Click on to activate the continuous CCD polling.
- Click on .

The **CCD Control Parameters** dialog will be opened:

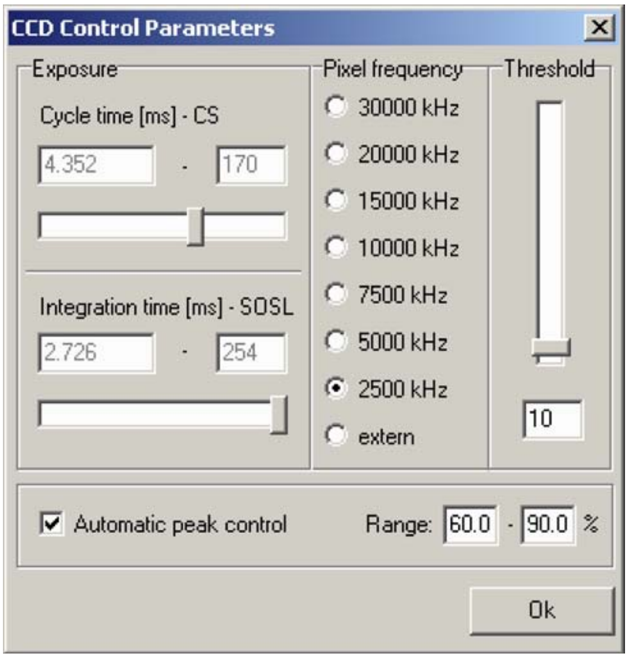


Figure 17-9: **CCD Control Parameters** dialog

- Enable **Automatic peak control** to activate a CCD peak detection with default settings.
- Specify the intensity **Range**.

  Useful limits are from 60% up to 90%. The hardware settings like pixel frequency, cycle and integration time will be adjusted automatically in order to get a peak with an optimized intensity between the selected limits.

  The limits are indicated as a dark green line within the CCD Control window.

- Adjust the Threshold to cut off the background noise of the signal. Only the distribution above this threshold will be used for the peak detection.

  The threshold is indicated as a light green line within the **CCD Control** dialog.

STEP 3: Calibrating the camera

- In the **CCD Control** dialog click on .

  The **CCD Calibration** dialog will be opened.

- Click on **New range**.

  The **CCD calibration range** subdialog will be opened:

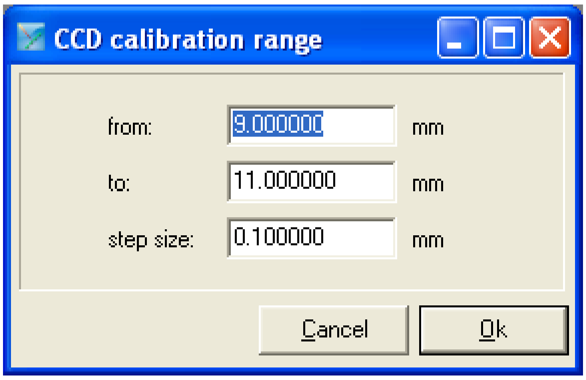


Figure 17-10: **CCD calibration range** subdialog.

▶ Define the working distance range and the increment of calibration points:

- Set the working distance **from** 9 mm **to** 11 mm.
- Enter a **step size** as increment of the calibration points.

  The relationship of the peak position and real height of the surface is a linear function. Therefore a higher number of calibration points increases the accuracy of the linear fit. A step size of at least 100 µm is recommended.

▶ Click on **OK** to store the settings.

In the **CCD Calibration** dialog, the number of calibration points and the corresponding working distances will be displayed:

▶ Click on **Start**.

The calibration procedure will be performed fully automatically. The stage will drive to the lower W limit of the calibration range. The stage will be moved down successively, by using the selected increment as step size. At each level the height of the surface will be detected by the LHS.

The detected peak will be displayed in the **CCD Control** dialog as a red mark. The corresponding pixel and working distance values are displayed in the status bar simultaneously.

After every calibration point, the system asks to check and accept the detected peak:

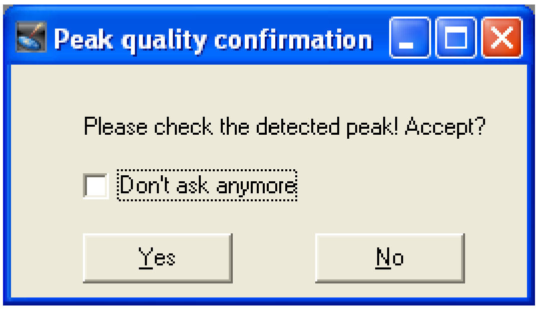


Figure 17-11: **Peak quality confirmation** dialog

- In the **CCD Control** dialog, check the peak detection (⇒ *Monitoring CCD peak detection* on page 17-7).
- Click on **OK** to confirm.

The **Peak quality confirmation** dialog will be displayed after every calibration point. Enable **Don‘t ask anymore**, if the quality of the peak detection is reliable.

The calibration procedure will be proceeded. It is finished, if all W values are assigned to a corresponding pixel value:

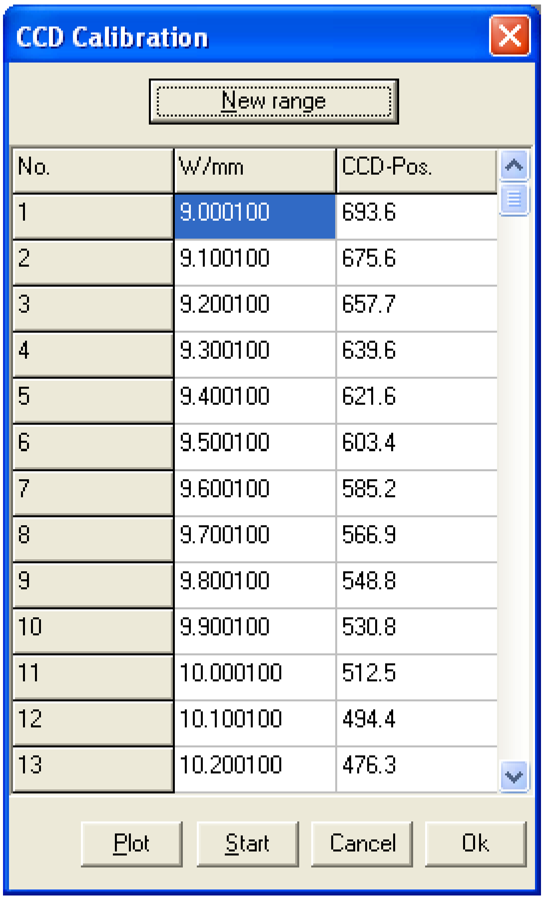


Figure 17-12: **CCD Calibration** subdialog – calibration ready

▶ Click on **Plot** to check the calibration result.

The **CCD calibration data** subdialog will be opened, showing the detected peak position as a function of the working distance.

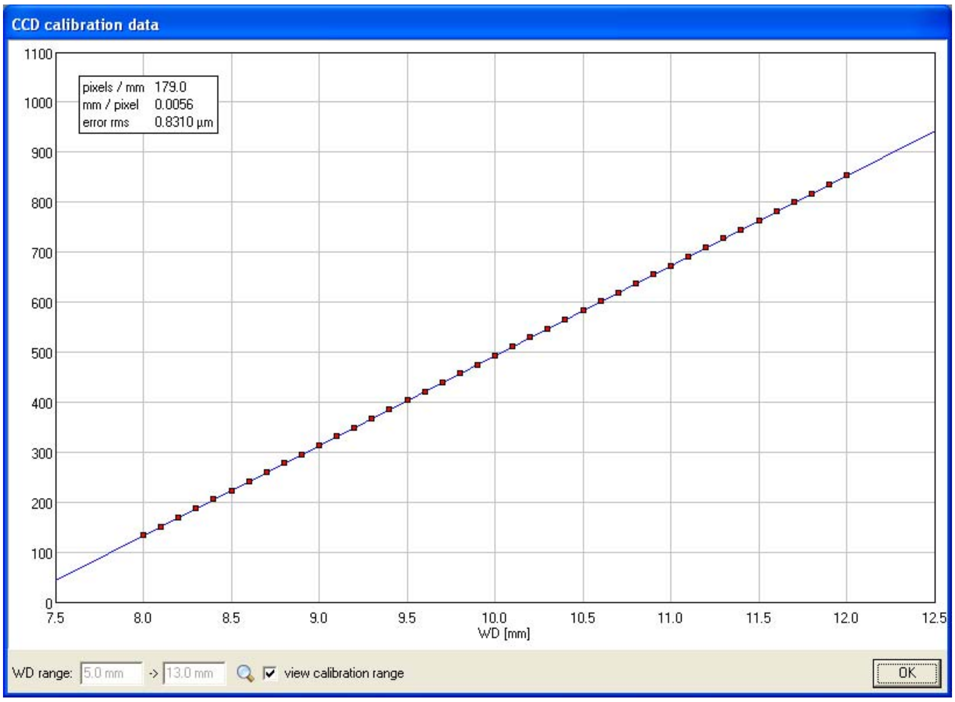


Figure 17-13: **CCD Calibration data** subdialog – calibration result

▶ Check the graph: If there are significant deviations from a linear function or any outliers repeat the calibration procedure.

▶ Click on **OK** to close the subdialog.

▶ In the **CCD Control** dialog, click on **OK** to store the calibration data.

5 Preparing sample holders and piezos

The following sections explain the preparation of the sample holders.

5.1 Pre-leveling the RAITH150 Two sample holders

The RAITH150 Two sample holders are equipped with three kinematic mounts to place them on the three piezos of the stage. To bring the sample holders into the piezo elongation range of 100 µm, the kinematic mounts have to be pre-leveled manually. The pre-leveling is necessary at the first time of use a new sample holder or in case of misalignment.

How to pre-level the sample holders refer to the RAITH150 Two operation manual.

5.2 Calibrating the piezos

– This calibration procedure has to be performed exclusively by a Raith trained system expert.
– A repetition is recommended every six months or in case of misalignment.

The piezo calibration procedure can be used to recalibrate the piezos, which means a redefinition of the relationship between the control voltage and the resulting piezo elongation. You can perform the calibration procedure by focusing manually via the electron or ion optics or by using the automatic focus control mode of the LHS. In the last case, the piezo movement will be detected by the CCD camera.

In the following the calibration using the automatic focus control is described.

Using the LHS, the calibration procedure should only be carried out with valid calibration data for the CCD camera (⇒ *Calibrating the CCD camera* on page 17-16).
Further the peak detection should have been checked.

STEP 1: Checking current piezo positions

▶ Maximize the **CCD Control** dialog and check the CCD peak detection. If necessary, calibrate the CCD camera (⇒ *Calibrating the CCD camera* on page 17-16).

▶ Change to the **Piezo Control** dialog:

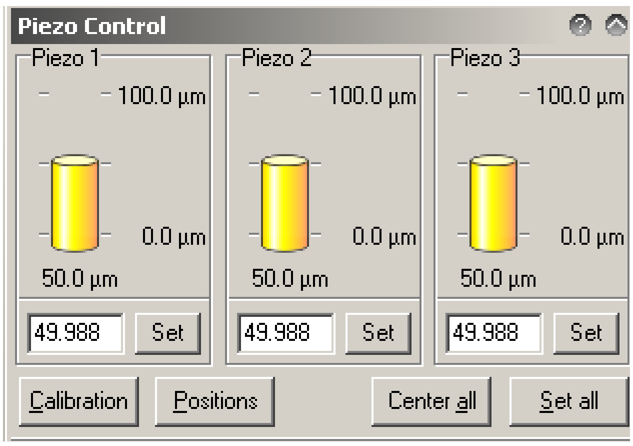


Figure 17-14: **Piezo Control** dialog

▶ Click on **Positions** to check, if the correct positions of the piezos are stored.

▶ The **Piezo positions** subdialog will be opened:

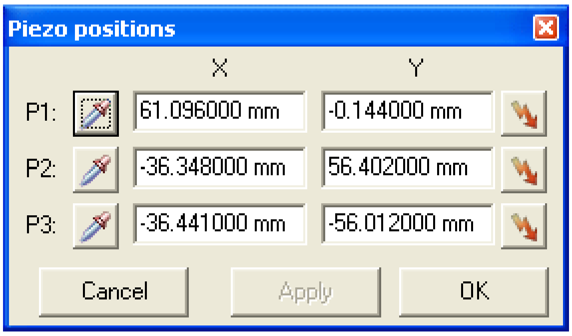


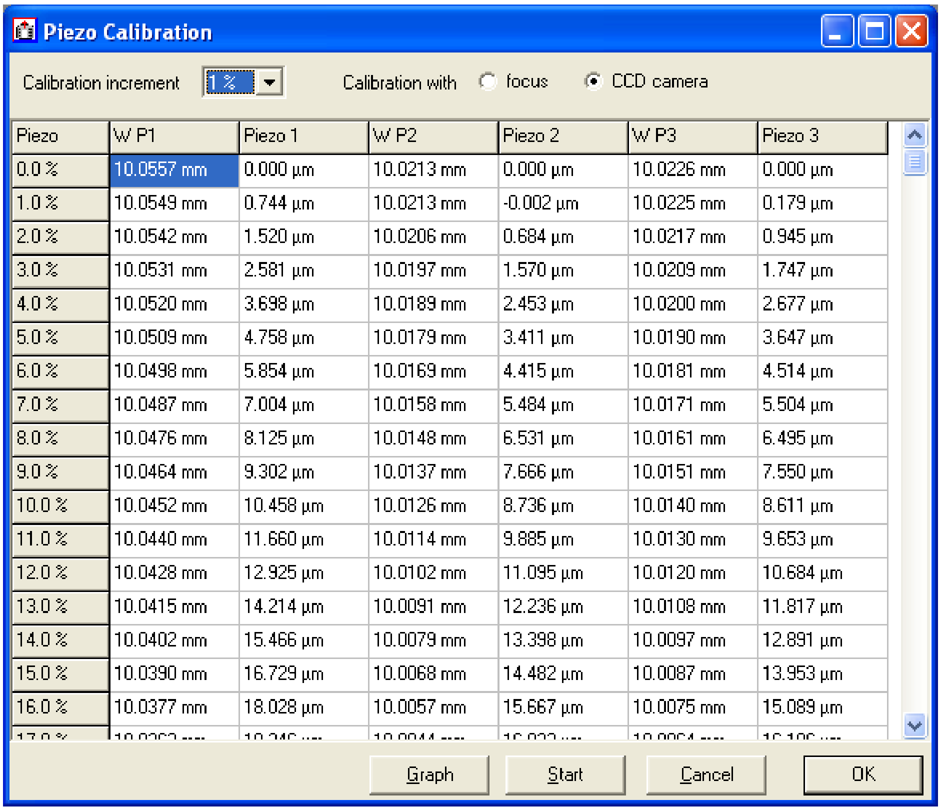
Figure 17-15: **Piezo positions** subdialog

- Click on to drive the stage to the corresponding positions.
- Click on to read the current positions and check the XY coordinates.
- Click on **OK** to close the dialog and save the settings.

STEP 2: Performing the calibration procedure

- In the **Piezo control** dialog, click on **Calibration**.

The **Piezo Calibration** dialog will be opened and the values of the last calibration will be displayed:



Piezo	W P1	Piezo 1	W P2	Piezo 2	W P3	Piezo 3
0.0 %	10.0557 mm	0.000 µm	10.0213 mm	0.000 µm	10.0226 mm	0.000 µm
1.0 %	10.0549 mm	0.744 µm	10.0213 mm	-0.002 µm	10.0225 mm	0.179 µm
2.0 %	10.0542 mm	1.520 µm	10.0206 mm	0.684 µm	10.0217 mm	0.945 µm
3.0 %	10.0531 mm	2.581 µm	10.0197 mm	1.570 µm	10.0209 mm	1.747 µm
4.0 %	10.0520 mm	3.698 µm	10.0189 mm	2.453 µm	10.0200 mm	2.677 µm
5.0 %	10.0509 mm	4.758 µm	10.0179 mm	3.411 µm	10.0190 mm	3.647 µm
6.0 %	10.0498 mm	5.854 µm	10.0169 mm	4.415 µm	10.0181 mm	4.514 µm
7.0 %	10.0487 mm	7.004 µm	10.0158 mm	5.484 µm	10.0171 mm	5.504 µm
8.0 %	10.0476 mm	8.125 µm	10.0148 mm	6.531 µm	10.0161 mm	6.495 µm
9.0 %	10.0464 mm	9.302 µm	10.0137 mm	7.666 µm	10.0151 mm	7.550 µm
10.0 %	10.0452 mm	10.458 µm	10.0126 mm	8.736 µm	10.0140 mm	8.611 µm
11.0 %	10.0440 mm	11.660 µm	10.0114 mm	9.885 µm	10.0130 mm	9.653 µm
12.0 %	10.0428 mm	12.925 µm	10.0102 mm	11.095 µm	10.0120 mm	10.684 µm
13.0 %	10.0415 mm	14.214 µm	10.0091 mm	12.236 µm	10.0108 mm	11.817 µm
14.0 %	10.0402 mm	15.466 µm	10.0079 mm	13.398 µm	10.0097 mm	12.891 µm
15.0 %	10.0390 mm	16.729 µm	10.0068 mm	14.482 µm	10.0087 mm	13.953 µm
16.0 %	10.0377 mm	18.028 µm	10.0057 mm	15.667 µm	10.0075 mm	15.089 µm

Figure 17-16: **Piezo Calibration** subdialog, showing last calibration values

- Enable **CCD camera** to detect the piezo elongation by the automated focus control of the LHS.
- Define the **Calibration increment**.

  The relationship of the piezo control voltage and the piezo elongation is not linear. Therefore a higher number of calibration points increases the accuracy of this calibration. At least an increment in the range of 10% is recommended.

  After defining the increment, the former values will be deleted.

- Click on **Start**.

  The calibration procedure will be performed fully automatically for each piezo one after the other. The current operating status will be displayed on the left hand side of the command bar. At each calibration level, the piezo expansion will be detected by the CCD camera. The system will prompt to check the CCD peak detection, respectively.

- Enable **Don't ask anymore** and confirm with **OK**.

  The procedure will be carried on and the working distances at each calibration level will be displayed in the columns **W1** to **W3**. When the detection of the working distances is finished, the corresponding effective piezo expansion in µm will be calculated and displayed in the columns **Piezo 1** to **Piezo 3**.

  If the calibration procedure is successfully finished, the message **Calibration OK** will be displayed next to the command bar of the **Piezo Calibration** dialog.

STEP 3: Checking the calibration result

- Click on **Graph** to check the result.

  The piezo elongation as a function of the piezo control voltage will be displayed:

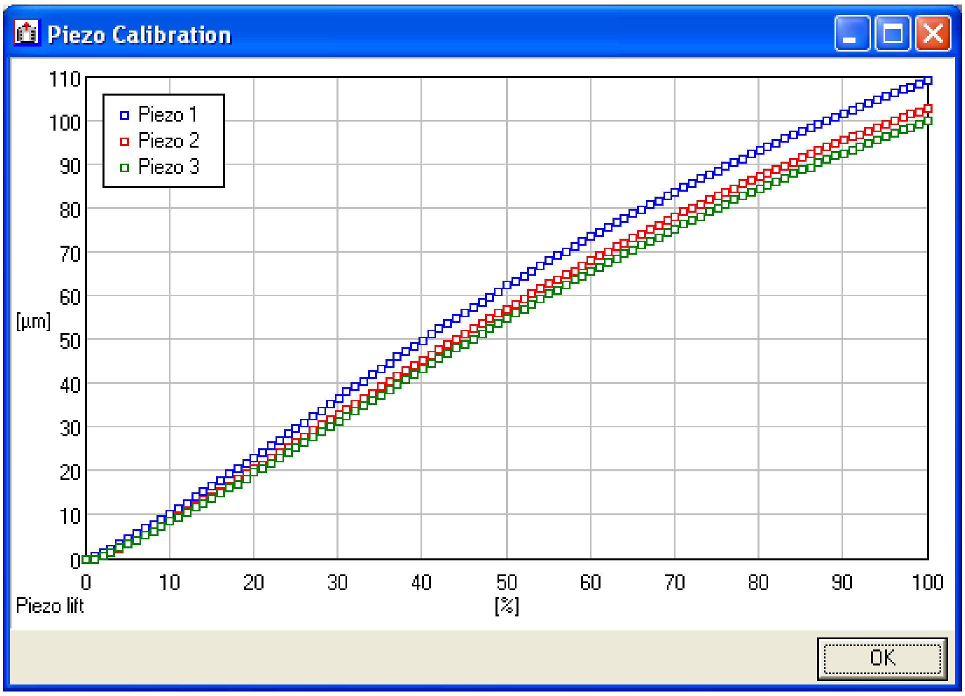


Figure 17-17: **Piezo Calibration** dialog, showing the calibration result as graph

The maximum elongation of each piezo should be in the range of 105 µm +/- 10%. Observe the smoothness of the curves. If there are significant jumps or peaks repeat the calibration procedure.

- ▶ Click on **OK** to close the graph.
- ▶ In the **Piezo Calibration** dialog, click on **OK** to store the calibration data.

C Software Reference

1 Height Control dialog

In the **Height Control** dialog, you can choose an appropriate method for the height control of your sample.

You call up the dialog via control bar .
The **Height Control** dialog will be displayed right to the control bar. On the control bar **Height Sensing** will be displayed.

How to activate and use one of the height control methods refer to (⇒ *Controlling and correcting the sample height* on page 17-8)

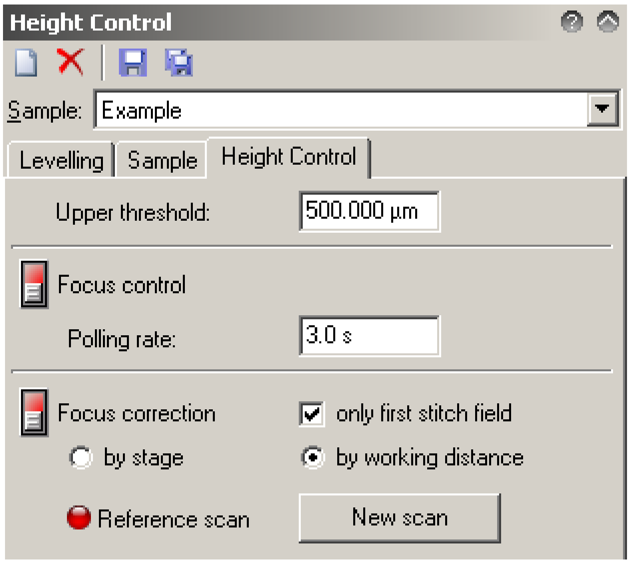


Figure 17-18: **Height Control** dialog

The dialog consists of two subsets

- Tool bar to set up and manage height control datasets for a sample.
- Tab subset with three tabs to:
  - activate and define one of the height control methods carried out exclusively by the LHS (⇒ *Height Control dialog – Height Control tab* on page 17-26),
  - define the UV coordinates for the triple-point sample leveling (⇒ *Height Control dialog – Sample tab* on page 17-28),
  - control and center the elongation of the piezos and start the triple-point sample leveling (⇒ *Height Control dialog – Leveling tab* on page 17-29).

1.1 Height Control tool bar

Figure 17-19: **Height Control** dialog – tool bar

The control elements have the following functions:

Control element	Function
	Sets up a new height control dataset for the current sample or application. All height control settings in the **Height Control** dialog can be stored to this dataset.
	Deletes a dataset selected in the **Sample:** combo box.
	Saves the current height control settings in the selected dataset.
	Saves the current height control settings under a different name.
Sample:	To select an existing height control dataset.

1.2 Height Control dialog – Height Control tab

The **Height Control** tab serves to activate and define the **Focus Control** or **Focus Correction** mode.

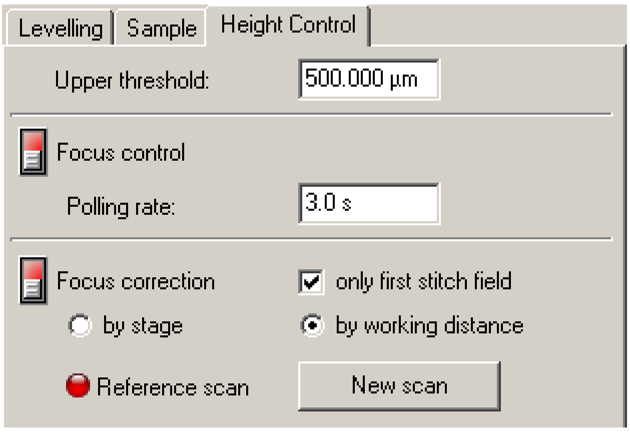


Figure 17-20: **Height Control** dialog – **Height Control** tab

The control elements have the following functions:

Control element	Function
Upper threshold:	To define a working distance threshold up to which the focus control and focus correction are to be executed.
Focus control	To activate the continuous **Focus Control** mode. In this mode, the working distance will be corrected continuously during the operation of the system – e.g. stage movement via the joystick. During a patterning process, the mode will be disabled automatically. Green indicates the activated state.
Polling rate:	To define, how often the CCD peak detection of the CCD camera will be read and the focus corrected.
Focus correction	To activate the stage movement triggered **Focus Correction** mode. In this mode, the working distance will be corrected during a patterning process or a Positionlist scan. Green indicates the activated state.
only first stitch field	To execute the correction of the focus only at the beginning of the patterning. **Note:** By default, the correction will be executed at any stitch field.
by stage	To perform the correction by moving the stage on its Z axis.
by working distance	To perform the correction by working distance adjustment.
Reference scan	Indicates that a new reference scan is executed. The LED illuminates green, if a reference scan is currently executed.
New scan	To perform a reference CCD line scan that is needed for proper correction of the focus during the patterning or positionlist scan. The reference scan serves to determine a reference working distance as basis for the focus correction (⇒ *Using Focus Correction mode* on page 10).

1.3 Height Control dialog – Sample tab

The **Sample** tab serves to define the UV coordinates of three leveling points on the sample as basis for the triple-point sample leveling method.

How to perform a triple-point sample leveling refer to (⇒ *Triple-point leveling of the sample surface* on page 17-13)

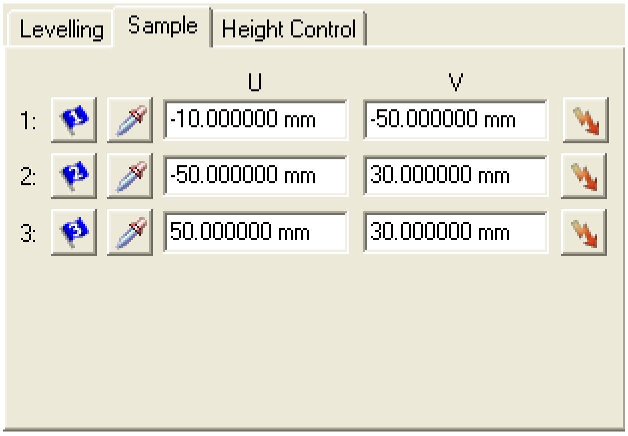


Figure 17-21: **Height Control** dialog – **Sample** tab

The control elements have the following functions:

Control element	Function
1: to **3:**	Indicates the number of one leveling point.
 to 	Adds a leveling point to any UV window, e.g. an image, GDSII viewer or editor. The corresponding UV values will be read into the adjacent UV input fields. **Recommendation:** The greater the distance between the three leveling points, the higher the accuracy of leveling the sample surface. Furthermore it is recommended to choose positions without high topography to guarantee a high quality of the peak detection of the CCD camera.
	Reads the current UV position of the stage for a selected leveling point.
UV columns	– Displays the UV coordinates of a leveling point defined via the blue flags or the pipette. – To define the UV coordinates manually.
	Drives the stage to a defined leveling point.

1.4 Height Control dialog – Leveling tab

The **Levelling** tab serves to control and center the elongation of the piezos and start the triple-point sample leveling.

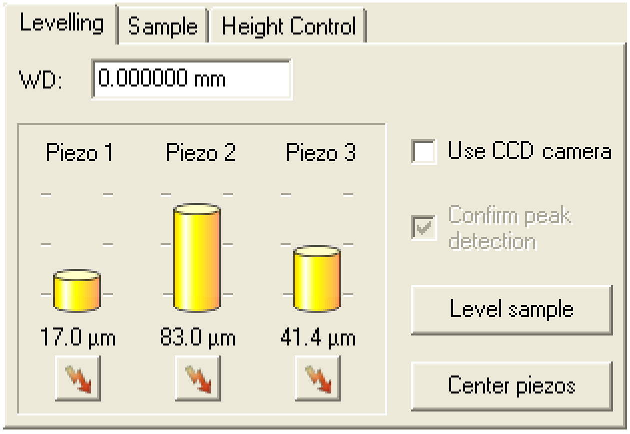


Figure 17-22: **Height Control** dialog – **Levelling** tab

The control elements have the following functions:

Control element	Function
WD:	To enter a working distance in mm, to which the stage has to be moved after the leveling procedure is finished. After finishing the leveling procedure the stage will be automatically adjust the Z axis. **Note:** If no working distance is defined, the stage will not be moved after the leveling procedures.
Piezo 1 to Piezo 3 columns	Shows the physical elongation of the corresponding piezo µm.
	Drives the stage to the position of the corresponding piezo.
Use CCD camera	To execute the height control during the leveling procedures via the CCD camera of the Laser Height Control. **Note:** If **Use CCD camera** is disabled, you will be asked to read out the working distance instead of getting the height information from the Laser Height Sensing.
Confirm peak detection	To activate a prompt message to confirm the CCD peak detection before a leveling procedure. **Note:** Enable **Confirm peak detection**, if the CCD peak detection may not work reliable. Leave it disabled, if it is reliable. The leveling procedure will be executed without any further user interaction.

Control element	Function
Level sample	Starts the triple-point sample leveling procedure.
Center piezos	Sets all piezos to a physical elongation of 50 µm. This position is a good starting point for the first leveling procedure. All piezos have the same space for elongation in both directions – upwards and downwards..

2 CCD Control dialog

In the **CCD Control** dialog you can monitor and adjust the CCD peak detection as well as calibrate the camera.

The **CCD Control** dialog will be displayed as window in the desktop area of the NANOSUITE software.

How to monitor the peak detection and calibrate the CCD camera refer to
(⇒ *Monitoring CCD peak detection* on page 17-7)
(⇒ *Calibrating the CCD camera* on page 17-16)

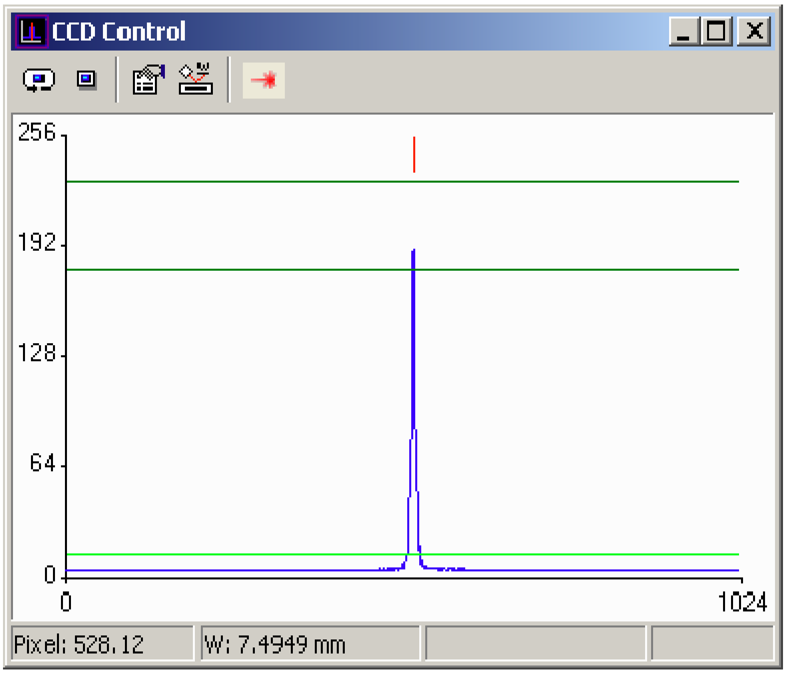


Figure 17-23: **CCD Control** dialog

The dialog consists of three subsets:

- Tool bar to control the peak detection and calibrate the CCD camera (⇒ *CCD Control dialog – tool bar* on page 17-32).
- Peak detection window to monitor and check the CCD peak detection (⇒ *Monitoring CCD peak detection* on page 17-7).
- Status bar with the pixel value of the peak and the corresponding working distance in mm.

2.1 CCD Control dialog – tool bar

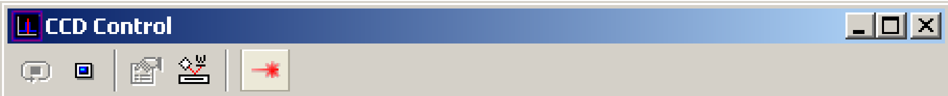
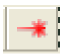


Figure 17-24: CCD Control dialog – Tool bar

The commands have the following functions:

Command	Function
	Activates continuous CCD polling for monitoring the CCD peak detection.
	Displays the current CCD peak detection in the CCD peak detection window.
	Calls up the **CCD Control Parameters** subdialog to make settings for the peak detection (⇒ *CCD Control Parameters subdialog* on page 32).
	Calls up the **CCD Calibration** subdialog to define the calibration range (⇒ *CCD Calibration subdialog* on page 34).
	To switch the laser beam on or off. **Note:** If the button is enabled, the laser beam will be switched on permanently. If the button is disabled, the laser beam will only be switched on at every defined polling interval (⇒ *Height Control dialog – Height Control tab* on page 26). On the one hand, this protects the surface of sensitive samples. On the other hand, the laser beam polling can affect the quality of the electron beam and lead to an inaccurate patterning process – e.g. during image scans.

2.2 CCD Control Parameters subdialog

In the **CCD Control Parameters** dialog, you can define settings for the CCD peak detection. The parameters affect the frequency distribution of the laser light distribution in the **CCD Control** dialog.

You call up the subdialog via **CCD Control** dialog .

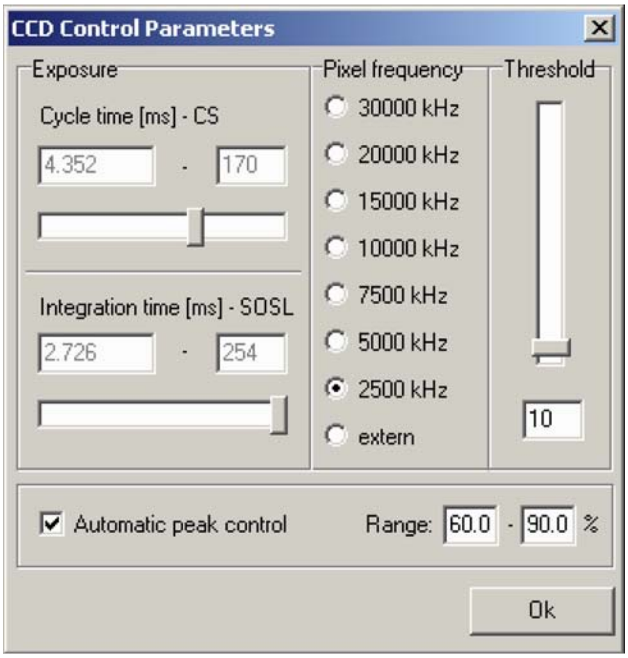


Figure 17-25: **CCD Control Parameters** dialog

Patterning and Pixel frequency subsets contain hardware settings of the LHS. The parameters are only to be set by a Raith trained system expert. We recommend to activate the **Automatic peak control** function to adjust the hardware settings with default values.

The control elements have the following functions:

Control element	Function
Threshold	To select a threshold for the intensity of the CCD signal. The laser light distribution with an intensity below this threshold will not be used for the peak detection.
Automatic peak control	Activates a CCD peak detection with default settings. The hardware settings will be adjusted automatically in order to obtain a peak with an optimized intensity.
Range	To define an intensity range which will be used to detect the CCD peak. The limits will be displayed as dark green lines in the **CCD Control** dialog (⇒ Figure 17-2 on page 17-8).
OK	Saves the current settings and closes the dialog.

2.3 CCD Calibration subdialog

The **CCD Calibration** serves to calibrate the CCD camera and check the results.

You call up the subdialog via **CCD Control** dialog .

How to perform the calibration procedure of the CCD camera refer to (⇒ *Calibrating the CCD camera* on page 17-16).

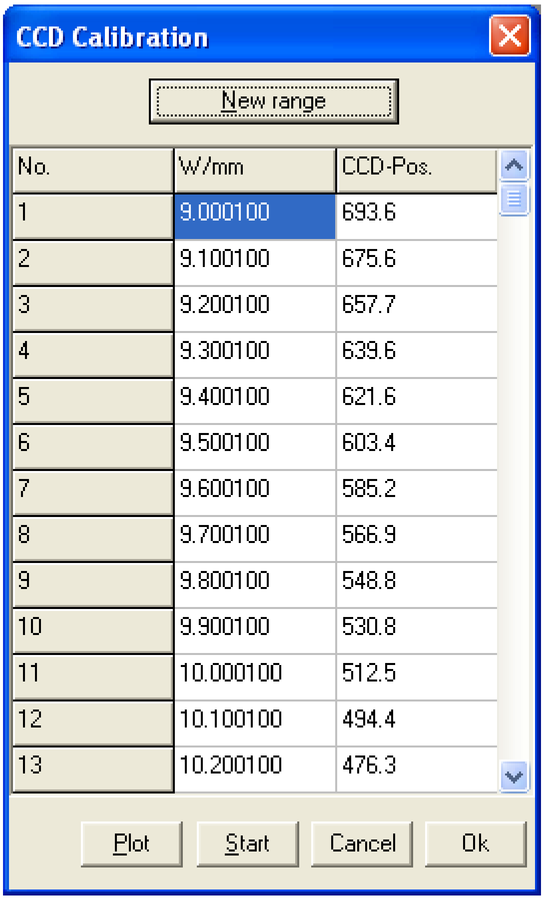


Figure 17-26: **CCD Calibration** subdialog

The control elements and columns have the following functions:

Control element/ Column	Function
New range	Calls up the **CCD calibration range** subdialog to determine the calibrating range (⇒ *CCD Calibration Range subdialog* on page 35).
No.	Shows the sequential number of the calibration step.
W/mm	Shows the working distance of each calibration step in mm.
CCD-Pos.	Shows the position of each calibrating point on the CCD sensor in px.
Plot	Calls up the CCD calibrating data subdialog to analyze the calibration result (⇒ *CCD Calibration Data subdialog* on page 36).
Start	Starts the calibration procedure. The procedure will run fully automatically.
Cancel	Cancels the calibration procedure at any time.
OK	Stores the calibration data and closes the dialog.

2.4 CCD Calibration Range subdialog

In the **CCD Calibration Range** subdialog, you can define the working distance range within which the calibration procedure is to be performed, as well as the increment of calibration steps. The procedure serves to calibrate the CCD peak detection against the real height of the sample surface.

Call up the subdialog via **CCD Control** dialog ➡  ➡ **New range**.

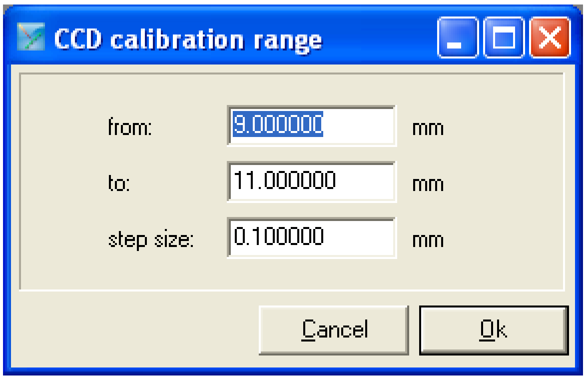


Figure 17-27: **CCD calibration range** dialog

The control elements have the following functions:

Control element	Function
from:	To enter the lower threshold of the working distance in mm.
to:	To enter the upper threshold of the working distance in mm.
step size:	To define the increment of steps at which the height of the sample surface will be detected by the LHS during the calibration procedure.

2.5 CCD Calibration Data subdialog

In the **CCD calibration data** dialog, you can analyze the result of the CCD calibrating procedure.

You call up the dialog via CCD Control dialog ➡ ➡ **Plot**.

How to perform the calibration procedure of the CCD camera, refer to (⇒ *Calibrating the CCD camera* on page 17-16)

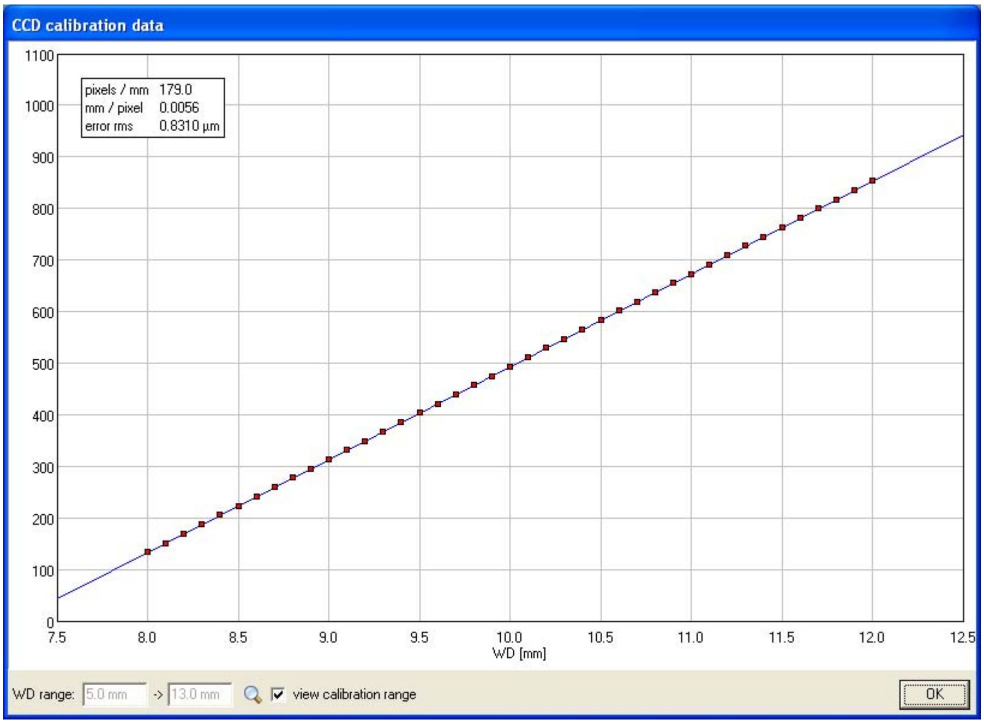


Figure 17-28: **CCD calibration data** subdialog

The graph represents the detected peak position as a function of the stage level. The dots correspond to the calibration points taken into account during the calibration procedure.

If there are significant deviations from a linear function or any peaks or outliers, repeat the calibration procedure.

The control elements have the following functions:

Control element	Function
WD range:	To enter the working distance range you wish to inspect.
	Shows the part of the graph corresponding to the defined working distance range.
view calibration range	Shows a graph over the calibration range. If the checkbox is disabled, a graph over the whole camera detecting range will be shown.
pixels/mm	Indicates the number of CCD pixels detected per mm working distance during the calibration procedure.
mm/pixel	Indicates how many mm of working distance per CCD pixel have been considered during the calibration.
error rms	Indicates the standard deviation from a linear function.

3 Piezo Control dialog (RAITH150 Two)

The dialog is only available to a Raith trained system expert.

The **Piezo Control** dialog serves to control and define the elongation of the three piezos as well as to calibrate the control voltage of the piezos against their physical elongation.

Call up the dialog by clicking on the icon in the control bar.
The **Piezo Control** dialog will be displayed to the right of the control bar.
On the control bar **Height Sensing** will be displayed.

How to perform the calibration procedure of the piezos refer to (⇒ *Calibrating the piezos* on page 17-21).

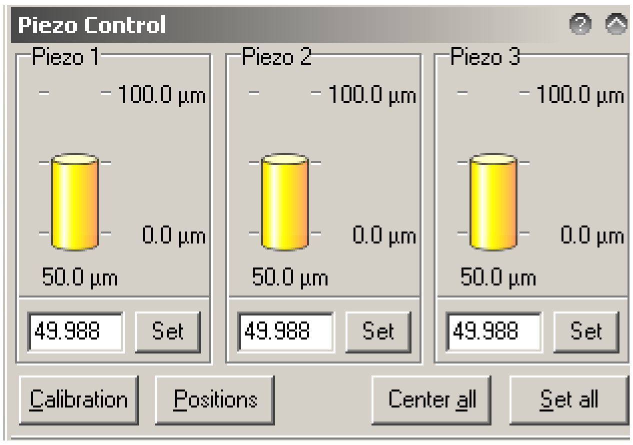


Figure 17-29: **Piezo Control** dialog

The control elements have the following functions:

Control element	Function
Piezo 1 to **Piezo 3** column	Shows the physical elongation of the corresponding piezo graphically and digitally in µm.
Input field	To enter the physical piezo elongation in µm.
Set	Adopts the entered value of the corresponding input field.
Calibration	Calls up the **Piezo Calibration** subdialog to calibrate the piezos (⇒ *Piezo Calibration subdialog* on page 40).

Control element	Function
	Calls up the **Piezo positions** subdialog to check and define the XY coordinates of the piezos (⇒ *Piezo positions subdialog* on page 39).
	Sets all piezos to a physical elongation of 50 µm. This value is half of the travel range of the piezos of 2047 digits. The maximum piezo travel range is about 4095 digits.
	Adopts the entered values of the three input fields all together.

3.1 Piezo positions subdialog

In the **Piezo positions** subdialog, you can check the current XY coordinates of the three piezos.

Call up the subdialog via **Piezo Control** dialog ➡ **Positions**.

How to check the current coordinates refer to (⇒ *Checking current piezo positions* on page 17-21).

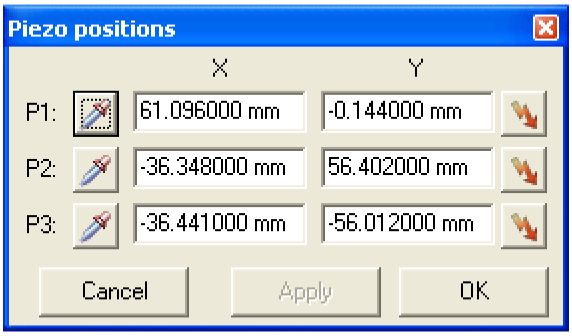


Figure 17-30: **Piezo Positions** subdialog

The control elements in the three rows have the following functions:

Control element	Function
P1: to **P3:**	Indicates the piezos 1 to 3.
	Reads the current XY coordinates of the corresponding piezo.
XY columns	To display and enter the XY coordinates of the piezos in mm.

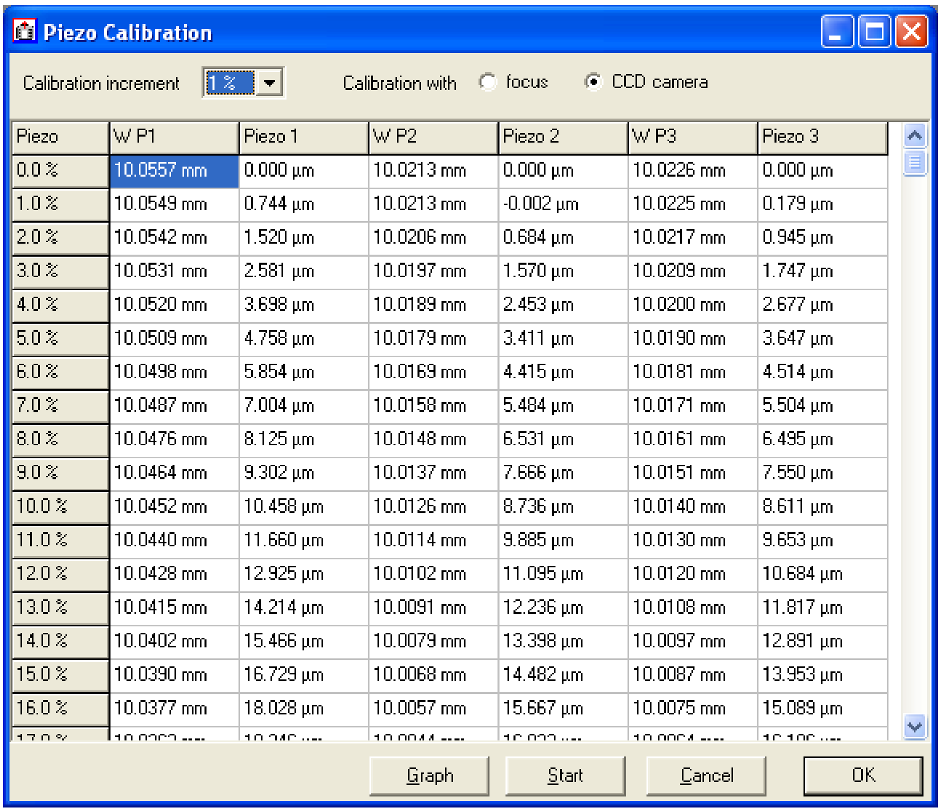
Control element	Function
	Drives the stage to the current XY coordinates of the corresponding piezo.
Cancel	Closes the dialog.
Apply	To apply new XY coordinates for the three piezos one after the other .
OK	Stores the current settings and closes the dialog.

3.2 Piezo Calibration subdialog

In the **Piezo Calibration** subdialog serves to calibrate the control voltage in % of the three piezos against their physical elongation in µm.

You call up the subdialog via **Piezo Control** dialog ➡ **Calibration**.

How to perform the calibration procedure of the piezos refer to (⇒ *Calibrating the piezos* on page 17-21).



Piezo	W P1	Piezo 1	W P2	Piezo 2	W P3	Piezo 3
0.0 %	10.0557 mm	0.000 µm	10.0213 mm	0.000 µm	10.0226 mm	0.000 µm
1.0 %	10.0549 mm	0.744 µm	10.0213 mm	-0.002 µm	10.0225 mm	0.179 µm
2.0 %	10.0542 mm	1.520 µm	10.0206 mm	0.684 µm	10.0217 mm	0.945 µm
3.0 %	10.0531 mm	2.581 µm	10.0197 mm	1.570 µm	10.0209 mm	1.747 µm
4.0 %	10.0520 mm	3.698 µm	10.0189 mm	2.453 µm	10.0200 mm	2.677 µm
5.0 %	10.0509 mm	4.758 µm	10.0179 mm	3.411 µm	10.0190 mm	3.647 µm
6.0 %	10.0498 mm	5.854 µm	10.0169 mm	4.415 µm	10.0181 mm	4.514 µm
7.0 %	10.0487 mm	7.004 µm	10.0158 mm	5.484 µm	10.0171 mm	5.504 µm
8.0 %	10.0476 mm	8.125 µm	10.0148 mm	6.531 µm	10.0161 mm	6.495 µm
9.0 %	10.0464 mm	9.302 µm	10.0137 mm	7.666 µm	10.0151 mm	7.550 µm
10.0 %	10.0452 mm	10.458 µm	10.0126 mm	8.736 µm	10.0140 mm	8.611 µm
11.0 %	10.0440 mm	11.660 µm	10.0114 mm	9.885 µm	10.0130 mm	9.653 µm
12.0 %	10.0428 mm	12.925 µm	10.0102 mm	11.095 µm	10.0120 mm	10.684 µm
13.0 %	10.0415 mm	14.214 µm	10.0091 mm	12.236 µm	10.0108 mm	11.817 µm
14.0 %	10.0402 mm	15.466 µm	10.0079 mm	13.398 µm	10.0097 mm	12.891 µm
15.0 %	10.0390 mm	16.729 µm	10.0068 mm	14.482 µm	10.0087 mm	13.953 µm
16.0 %	10.0377 mm	18.028 µm	10.0057 mm	15.667 µm	10.0075 mm	15.089 µm

Figure 17-31: **Piezo Calibration** subdialog

The subdialog consists of three subsets:

- Tool bar to define the calibration increment and the mode of the height control (⇒ *Piezo Calibration tool bar* on page 17-41),
- List subset to display the current calibrating data (⇒ *Piezo Calibration dialog – List subset* on page 17-42),
- Command bar to start the calibration procedure and check the results (⇒ *Piezo Calibration dialog – Command bar* on page 17-42).

3.2.1 Piezo Calibration tool bar

The control elements have the following functions:

Control element	Function
Calibration increment	To select the increment of calibration points at which the piezos are to be calibrated in % control voltage. **Note:** The relationship of the piezo control voltage and the piezo elongation is not linear. Therefore a higher number of calibration points increases the accuracy of this calibration. At least an increment in the range of 10% is recommended.
Calibration with focus	To activate the semiautomatic calibration mode. In this mode, the height control of the piezo elongation takes place via the focus control in the electron or ion optics under your control. At every calibration point the system will ask you to adjust the column focus. The Piezo Message subdialog will be displayed respectively (⇒ *Piezo Message subdialog* on page 42). **Note:** The semiautomatic mode may result in a higher accuracy of the calibrating data, but it takes longer.
Calibration with CCD camera	To activate the automatic height control mode of the LHS. In this mode the piezo elongation will be detected by the CCD camera and the corresponding working distances will be generated automatically. **Note:** Using the automatic height control presumes that the peak detection of the CCD camera is reliable. For this, check the CCD peak detection in the CCD Control dialog first and calibrate the CCD camera if necessary.

3.2.2 Piezo Calibration dialog – List subset

The column entries display the following data:

Column entry	Data
Piezo	Control voltage of the piezos in %.
W P1 to **W P2**	Working distances of the piezos detected by the CCD camera or set manually via the electron or ion optics.
Piezo 1 to **Piezo 3**	Physical elongation of the piezos ascertained by the calibration procedure in µm.

3.2.3 Piezo Calibration dialog – Command bar

The commands have the following functions:

Command	Function
Graph	Displays the calibration data as a graph to check the result. The graph shows the physical piezo elongation as a function of the piezo control voltage (⟹ *Checking the calibration result* on page 23).
Start	Starts the piezo calibration procedure.
Cancel	Cancels the piezo calibration procedure at any time.
OK	Stores the current calibrating data and closes the dialog.

3.3 Piezo Message subdialog

The **Piezo Message** subdialog will be displayed, if the piezo calibration procedure is performed by focusing manually. After every calibration point the system prompts to adjust the column focus.

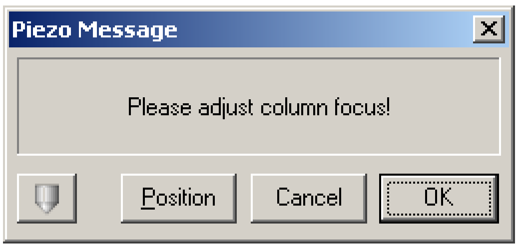
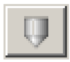


Figure 17-32: **Piezo Message** subdialog

The control elements have the following functions:

Control element	Function
	To switch the beam on or off.
Position	Drives the stage to the UV positions of the corresponding calibration points. **Note:** Generally, the stage will be driven to the UV positions of the calibration points automatically. Use this command, if you moved the stage with the joystick beforehand.
Cancel	Cancels the piezo calibration procedure at any time.
OK	Stores the current calibrating data and closes the dialog.

Chapter 18
Load Lock (option)

This chapter describes how to exchange sample holders and samples via the **Load Lock** software module in combination with a Load Lock.

The **Load Lock** module is an optional module of the NANOSUITE software and only available if the lithography system is equipped with a load lock.
The structure and the commands of the **Load Lock** module differ depending on the NANOSUITE software. In the following, the **Load Lock** module is described which comes with the RAITH150 Two system.

This chapter contains the following topics and tasks:

- Short overview of the hardware and software equipment
  (⇒ *Functional Description* on page 18-2),
- How to load and unload a sample holder
  (⇒ *Locking sample holders* on page 18-4),
- How to charge and discharge an electrostatic chuck
  (⇒ *Charging/discharging electrostatic chuck* on page 18-11),
- Measures of troubleshooting in cases of malfunctions
  (⇒ *Troubleshooting* on page 18-12),
- Description of all functions and control elements of the **Load Lock** software module
  (⇒ *Software Reference* on page 18-13).

A Functional Description

This chapter gives a short overview of the basic functions of the load lock hardware and software equipment.

1 Hardware equipment

The RAITH150 Two comes with a load lock to load and unload samples carried on a sample holder. The locking will be performed via automatic Load and Unload procedures. The hardware is fully controlled via two software modules to initiate the Load and Unload procedures, to set up the system, and to aid the user in case of an error.

The load lock can be controlled via:

- the **Load Lock** software module from the PC,
- the touch panel **System Control Panel** placed above the load lock on the system housing.

In the following, only the control via the **Load Lock** software module is described.
For further information about the load lock hardware equipment and the control via the **System Control Panel** refer to the RAITH150 Two system manual.

---

2 Load Lock software module

The control of the load lock via the PC is carried out by the **Load Lock** dialog:

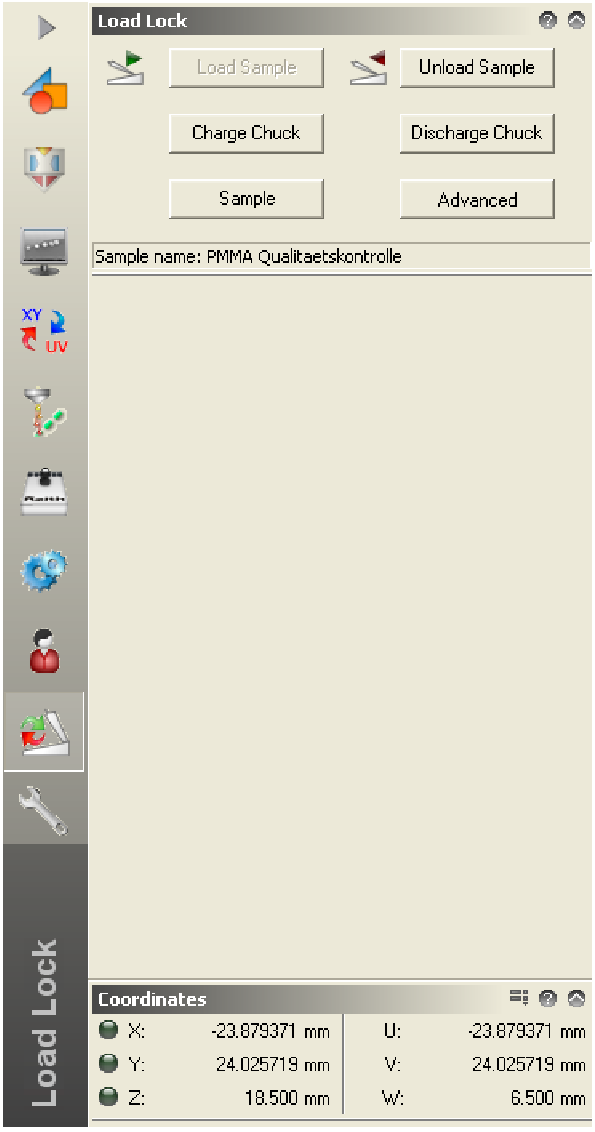


Figure 18-1: **Load Lock** module – **Load Lock** dialog

For a detailed description of all control elements and subdialogs of the dialog refer to section (⇒ *Load Lock dialog* on page 18-13).

B Tasks

This section describes how to:

- load and unload a sample holder (⇒ *Load Lock software module* on page 18-3).
- charge and discharge a chuck (⇒ *Charging/discharging electrostatic chuck* on page 18-11).
- perform troubleshooting in cases of malfunctions (⇒ *Troubleshooting* on page 18-12).

1 Locking sample holders

The locking of a sample holder takes place via the automated Load and Unload procedures. The user initiates the procedures with the **Load Sample** and **Unload Sample** commands. There are two ways to run the commands:

- using the **Load Lock** dialog on the PC monitor
- using the **Status** window on the **System Control Panel**.

A Load or Unload procedure is only executable when the robot arm of the load lock is situated in its pre-defined target position inside the load lock or inside the chamber.
If the robot arm is at a different position, the commands **Load Sample** and **Unload Sample** will be inactive. In such cases, the system has to be placed in a safe condition first (⇒ *Troubleshooting* on page 18-12).

1.1 Loading the sample holder

STEP 1: Making preparations

▶ Fix a sample on the sample holder.

▶ Insert the sample holder into the load lock.

How to fix samples and insert the sample holder refer to the RAITH150 Two system manual.

STEP 2: Performing the Load procedure

▶ Initiate the Load procedure by doing one of the following:

- on the PC monitor, click on **Load Sample** in the **Load Lock** dialog or
- on the **System Control Panel**, touch **Load Sample**.

By default, the following prompt message will be displayed on the monitor:

Figure 18-2: Load procedure – Prompt message start

▶ On the monitor, click on **OK** to start the load procedure.

To facilitate the control of the load lock via the **System Control Panel**, you can disable this prompt message: Set the **Confirm Load** variable to **OFF** via **Extras** ➡ **Settings** ➡ **LoadLockSPS**.
Otherwise you have to confirm the prompt message on the monitor every time before the Load procedure will be executed.

The load procedure will be started and the **Load Sample** dialog will be opened:

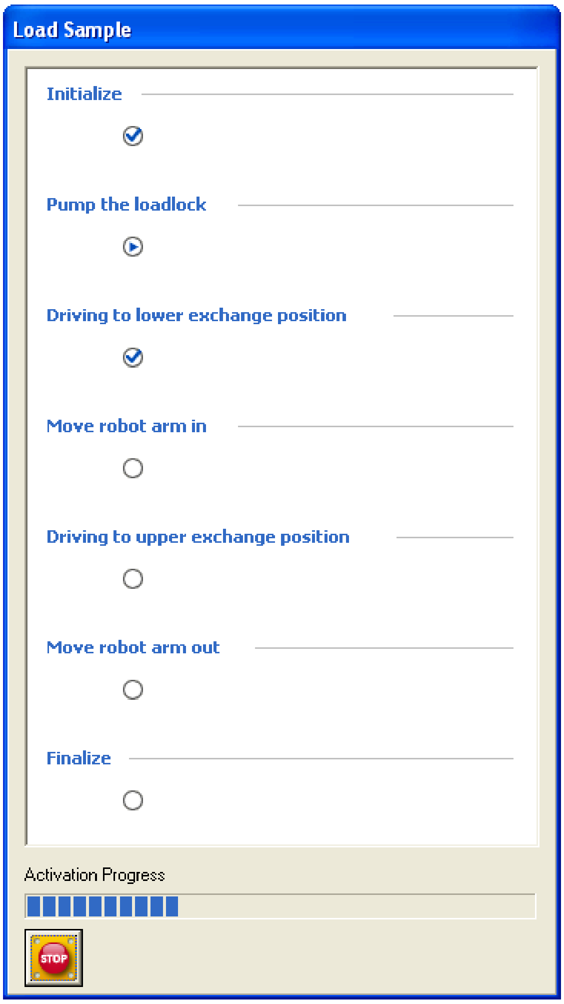


Figure 18-3: Load procedure – **Load Sample** dialog

The Load procedure will be carried out step-by-step and the progress bar will indicate the **Activation Progress**.

The ⦿ button indicates the currently active step, the ✓ button indicates a completed step.

You can stop the Load procedure at any time by pressing the **STOP** button. The sample holder will be moved back into the load lock.

Once the Load procedure is completed, the **Load Sample** dialog will be closed and the system will ask for a sample name:

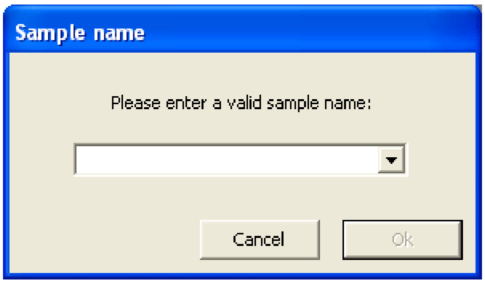


Figure 18-4: Load procedure – **Sample name** dialog

▶ Enter a unique sample name and confirm with **OK**.

All successive events recorded on the system will be saved under the sample name in a protocol. These events can be viewed in the Raith Protocol Tool.

The dialog will be closed and the system will ask for the position of the stage:

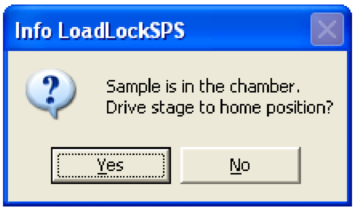


Figure 18-5: Load procedure – Prompt message home position

The home position is the starting position, prior to moving the sample into the required working position.

▶ Do one of the following:

- Confirm the prompt message with **Yes,** if the stage should be driven to the home position.
- Click on **No**, if you wish to keep the current position of the sample.

The system will ask to reset the coordinate system of your sample:

Figure 18-6: Load procedure – Prompt message coordinate system

The **Coordinates** windows displays the UVW coordinates of the currently loaded sample:

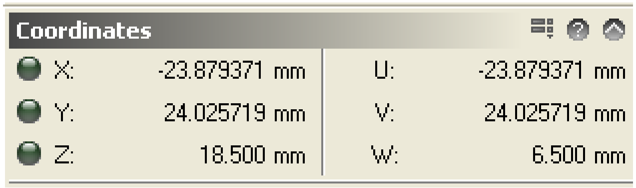


Figure 18-7: Load procedure – **Coordinates** window

The W value of 10 mm is the pre-defined working distance for the RAITH150 Two.

The value is based on the sample coordinate system you have calibrated for the sample loaded previously. Always reset the sample coordinate system when you are loading a new sample and calibrate it again. Reset it, even when you are loading a sample you have used before, but if you have loaded a different one in between.

How to calibrate the sample coordinate system refer, (⇒ *Coordinate systems and transformations* on page 12-3).

The system will prompt to reactivate the last used column parameter set:

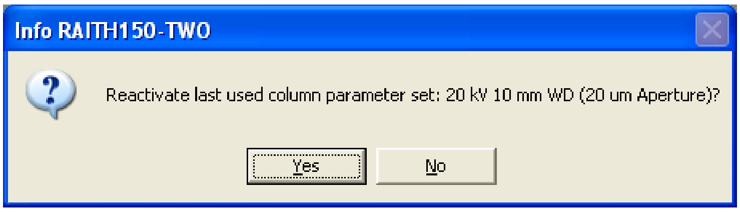


Figure 18-8: Load procedure – Prompt message coordinate system

- Click on **Yes**.

The **Load** procedure is now complete and you can move your sample into the working position.

How to position your sample refer (⇒ *Loading the sample holder* on page 18-4).

1.2 Unloading the sample holder

STEP 1: Performing the Unload procedure

- ▶ Initiate the Unload procedure by performing one of the following:
  - on the PC monitor, click on **Unload Sample** in the **Load Lock** dialog

    or
  - on the **System Control Panel**, click **Unload Sample**.

  By default, the following prompt message will be displayed on the PC monitor:

Figure 18-9: Unload procedure – Prompt message start

- ▶ On the monitor, click on **OK** to confirm and start the Unload procedure.

To facilitate the control of the load lock via the **System Control Panel**, you can disable this prompt message:
Set the **Confirm Unload** variable to **OFF** via **Extras** ➡ **Settings** ➡ **Load-LockSPS**.
Otherwise you have to confirm the prompt message on the PC monitor every time before the Unload procedure will be executed.

The Unload procedure will be started and the **Unload Sample** dialog will be opened (⇒ Figure 18-10 on page 18-10).

The Unload procedure will be carried out step-by-step and the progress bar will indicate the **Activation Progress**:

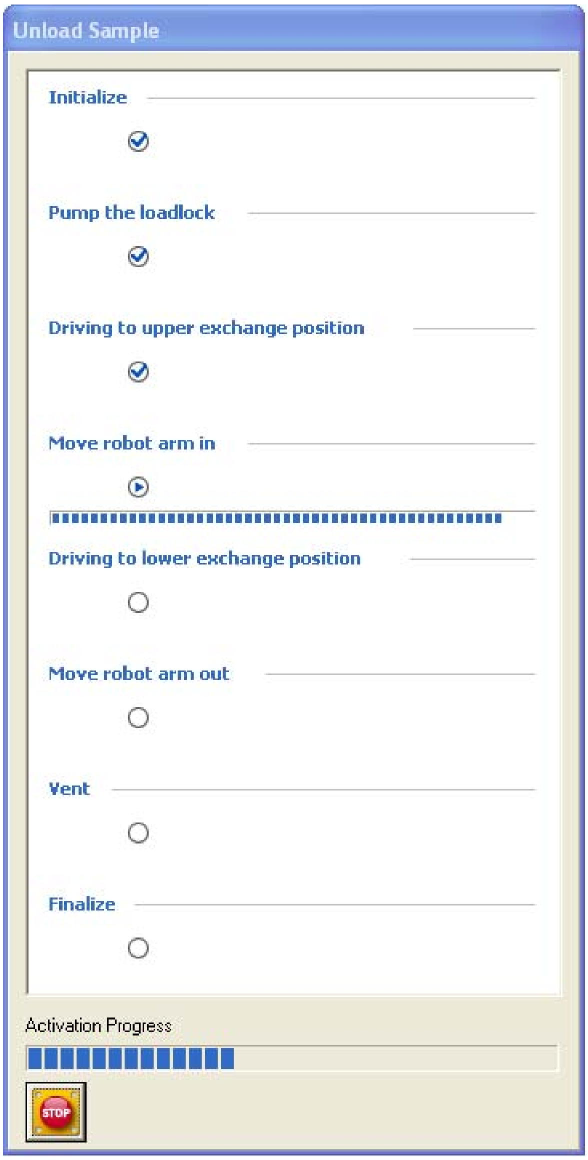


Figure 18-10: Unload procedure – **Unload Sample** dialog

The button indicates the currently active step, the button indicates a completed step.

You can stop the Unload procedure at any time by pressing the **STOP** button. The sample holder will be moved back into the chamber.

The Unload procedure is completed and you can remove the sample holder from the load lock.

NOTICE

The system vacuum can be degraded by dust or dirt from the load lock chamber. Remember to close the lid to the load lock chamber, once you have removed the sample holder.

2 Charging/discharging electrostatic chuck

For the RAITH150 Two different types of electrostatic chucks are available in order to handle wafers. The chucks can be charged or discharged while they are placed inside the load lock.

For general guidelines how to handle a chuck refer to the RAITH150 Two system manual.

STEP 1: Making preparations

- Prepare the chuck for use.
- Fix the wafer on the chuck.
- Insert the chuck into the load lock.

How to fix wafers and insert chucks refer to the RAITH150 Two system manual.

STEP 2: Charging the chuck

- On the PC monitor, change to the **Load Lock** module.
- In the **Load Lock** dialog, click on **Charge Chuck**.

The chuck will be charged.

STEP 3: Discharging the chuck

- If necessary, Unload the chuck (⇒ *Unloading the sample holder* on page 18-9).
- In the **Load Lock** dialog, click on **Discharge Chuck**.

The chuck will be discharged.

3 Troubleshooting

The sample holders are loaded and unloaded into the chamber via fully automated Load and Unload procedures. During these procedures, various mechanical switches are monitored and software checks are carried out to ensure that all functions are performed without errors.

During a Load or Unload procedure the system is in its most error-sensitive state. In the event of any kind of error indication, the system will initiate its own routine to deal with the errors and it will display appropriate prompt messages on the PC monitor. If this occurs, follow the instructions. The system will reset itself into a safe state.

The system is in a safe state when:

- The robot arm is in its target position inside the load lock,
- the VAT valve is closed,
- the valve between the backing pump and the load lock is closed so that the load lock cannot be pumped.

If the system's own troubleshooting procedure is not successful, a SYS-level user can manually complete a failed Load or Unload procedure, transferring the system to a safe state step by step. In order to do this, there are the **Single steps** commands in the **Advanced Load Lock** dialog available (⇒ *Single Steps window* on page 18-20).

NOTICE

System damage can occur when using inappropriate **Single steps** commands.

In **Single steps** mode, all safety interlocks are disabled. Only ever use the **Single steps** commands when instructed by Raith support.

C Software Reference

This section describes all functions and control elements of the **Load Lock** module.

1 Load Lock dialog

The **Load Lock** dialog serves to drive and control the load lock.

You call up the dialog via control bar .
The dialog will be displayed right to the control bar. On the control bar **Load Lock** will be displayed.

How to load or unload a sample holder and charge or discharge a chuck refer to
(⇒ *Locking sample holders* on page 18-4),
(⇒ *Charging/discharging electrostatic chuck* on page 18-11).

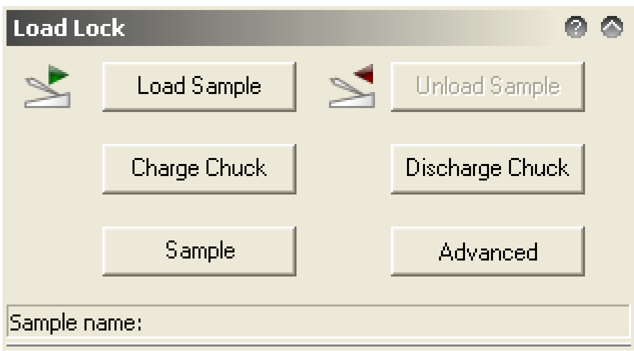


Figure 18-11: **Load Lock** dialog

The indicator and buttons have the following functions:

Indicator/Button	Function
Load Sample	Initiates the Load procedure.
Unload Sample	Initiates the Unload procedure.
Charge Chuck	Charges the electrostatic chuck.
Discharge Chuck	Discharges the electrostatic chuck.

Indicator/Button	Function
Sample	Opens the **Sample name** dialog to define a name for the sample. Under this name all successive events recorded on the system will be stored and saved in a protocol. **Note:** The events can be viewed via the Raith Protocol Tool. The sample name can be edited at any time. This may be done when, for example, several sample have been mounted on the universal sample holder.
Advanced	Opens the **Advanced Load Lock** dialog to monitor the operating status of the load lock and to make settings. The structure of the dialog is different depending if it is for USR-level or SYS-level user: (⇒ *Advanced Load Lock dialog (USR)* on page 14) (⇒ *Advanced Load Lock dialog (SYS)* on page 18)
Sample name:	Displays the name of the currently loaded sample.

1.1 Advanced Load Lock dialog (USR)

In the USR-level, the dialog serves to monitor the operating status of the load lock.

You call up the dialog via **Load Lock** dialog ➡ **Advanced** button.

The dialog is divided into the following windows:

- Error window to view error messages and procedures (⇒ *Error window* on page 18-15),
- Status window to monitor the load lock status (⇒ *Status window* on page 18-15).

Clicking on the corresponding button on the left hand side will toggle between the windows.

1.1.1 Error window

The **Error** window displays the latest error messages as well as the latest procedure executed.

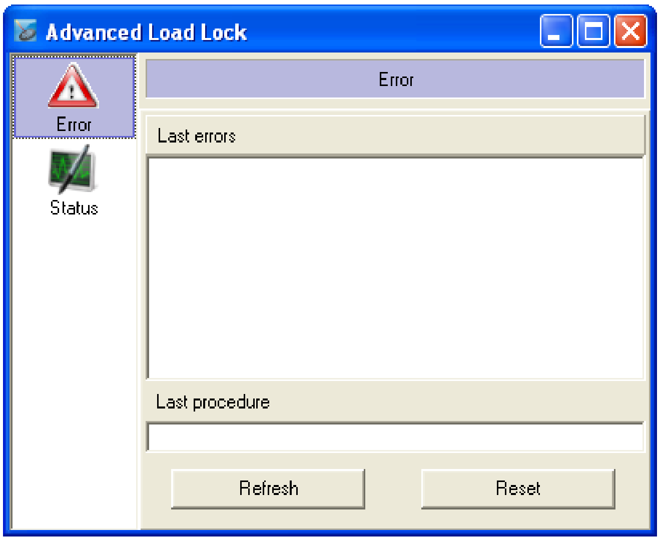


Figure 18-12: **Advanced Load Lock** dialog – **Error** window

The control elements have the following functions:

Control element	Function
Last errors	Shows a listing of the most recent error messages.
Last procedure	Shows the latest performed procedure.
Refresh	Refreshes the list of error messages.
Reset	Resets the latest error messages.

Not all error messages can be reset in the USR level. If individual error messages can not be reset, contact your local system expert.

1.1.2 Status window

The **Status** window enables you to call up information regarding

- the position of the robot arm,
- the type and position of the selected sample holder,
- the status of the valves.

The Status LEDs indicate the current status of the corresponding parameter.

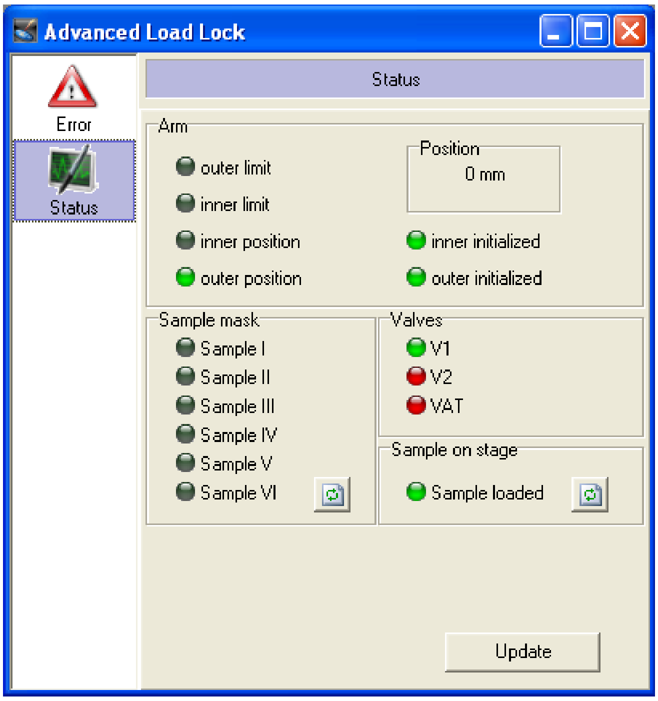
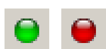


Figure 18-13: **Advanced Load Lock** dialog – **Status** window

The indicators and control elements have the following functions:

Indicator/ Control Element	Function
Arm	The **Arm** subset displays the current position of the robot arm. When the required position is reached, the LED illumination will change to green. The Status LED can indicate the following modes:
	Gray: Position has not been reached.
	Green: Position has been reached.
The robot arm can be situated in the following positions:	
outer limit	Robot arm is inside the load lock and the mechanical limit switch has been contacted.
inner limit	Robot arm is inside the chamber and the mechanical limit switch has been contacted.
inner position	Robot arm is at the target position inside the chamber.

Indicator/ Control Element	Function
outer position	Robot arm is at the target position inside the load lock.
Position	Shows the current position of the robot arm in millimeters.
outer initialized	Indicates if **outer position** is initialized or not. **Note:** It is not possible to load or unload samples if **outer position** is not initialized.
inner initialized	Indicates if **inner position** is initialized or not. **Note:** It is strongly recommended to initialize the **inner position** before first sample exchange otherwise it may happen that inner position can not be reached during the sample exchange procedure.
Sample mask	The **Sample mask** subset displays the type of the selected sample holder. The sample holder type is recognized and identified by the optical sample holder recognition of the load lock.
 Sample I-VI	The indicated sample holder type is inside the load lock (green) or not (gray).
	Refreshes the display.
Valves	The Valves subset indicates the status of the valves of the systems concerning the load lock.
 V1	Valve between the backing pump and turbo molecular pump is open (green) or closed (red).
 V2	Valve between backing pump and load lock is open (green) or closed (red).
 VAT	Valve between the load lock and the chamber is open (green) or closed (red).
Sample on stage	Indicates if the sample has reached its target position inside the chamber.
 Sample is loaded	Indicates if the sample is placed on the stage (green) or not (red).

Indicator/ Control Element	Function
	Refreshes the sample loaded state.
Update	To refresh the contents of status indicators.

1.2 Advanced Load Lock dialog (SYS)

In the SYS-level, the dialog serves to monitor the operating status of the load lock and to edit settings for the sample-stage loading position as well as the configuration of the load lock.

You call up the dialog via **Load Lock** dialog ➡ **Advanced** button.

The dialog consists of the following windows:

Window	Function
Error	– To display the latest error messages as well as the latest procedure performed (⇒ *Error window* on page 15). – To reset an error state.
Settings	To define the upper and lower loading position of the sample inside the chamber (⇒ *Settings window* on page 19).
Single steps	To perform some steps of a Load or Unload procedure manually in order to set the system into a safe state, under guidance of Raith support (⇒ *Single Steps window* on page 20).
Service	To edit the basic configuration of the load lock under guidance of Raith support (⇒ *Service window* on page 22).
Status	To monitor the operating status of the robot arm, sample holder and valves (⇒ *Status window* on page 15).

1.2.1 Settings window

NOTICE

Selecting inappropriate setting parameters can lead to system damage. Only ever change any **Settings** parameters when requested by Raith support.

In the Settings window you may set the X, Y and Z-coordinates in mm for the upper and lower loading positions of the sample.

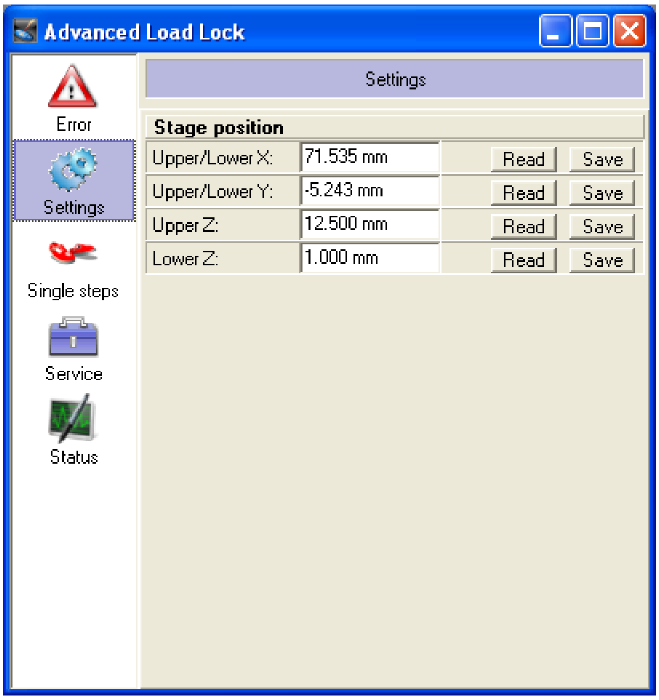


Figure 18-14: **Advanced Load Lock** dialog – **Settings** window

The control elements have the following functions:

Control element	Function
Upper/Lower X:	To set the X-coordinate for the upper and lower loading position.
Upper/Lower Y:	To set the Y-coordinate for the upper and lower loading position.
Upper Z:	To set the Z-coordinate for the upper loading position.
Lower Z:	Set the Z-coordinate for the lower loading position.
Read	To read in the current stage coordinate.

Control element	Function
Save	To save the coordinate entered.

1.2.2 Single Steps window

NOTICE

System damage can occur when using inappropriate **Single steps** commands.

In **Single steps** mode, all safety interlocks are disabled. Only ever use the **Single steps** commands when instructed by Raith support.

In the **Single steps** window, you can perform some of the steps of the automatic Load and Unload procedures manually, under guidance of Raith support.

The commands serve to set the system into a safe condition if a Load or Unload procedure failed and the system cannot be set into a safe condition via its own automatic error handling (⇒ *Troubleshooting* on page 18-12).

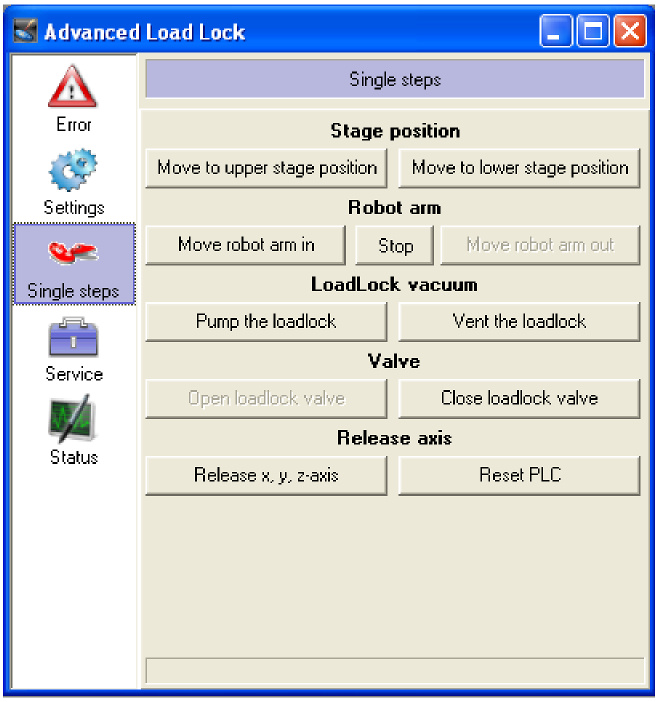


Figure 18-15: **Advanced Load Lock** dialog – **Single steps** window

The buttons have the following functions:

Button	Function
Move to upper stage position	Moves the stage into the predefined upper loading position.
Move to lower stage position	Moves the stage into the predefined lower loading position.
Move robot arm in	Moves the robot arm into its target position in the chamber. In order to do this, the stage must be positioned in its – upper loading position if the sample holder is in the load lock, – lower loading position if the sample holder is in the chamber. **Note:** This button is only active when the robot arm is inside the load lock and the VAT valve is open.
Stop	Stops any movement of the robot arm.
Move robot arm out	Moves the robot arm into its target position inside the load lock. **Note:** The button is only active when the robot arm is inside the chamber and the VAT valve is open.
Pump the load lock	Pumps down the load lock.
Vent the load lock	Vents the chamber
Open load lock valve	Opens the VAT valve. **Note:** The button is only active when the VAT valve is closed
Close load lock valve	Closes the VAT valve. **Note:** The button is only active when the VAT valve is open.
Release X, Y, Z-axis	Unlocks the X, Y and Z-axes of the stage after an system error. The axes of the sample holder will be automatically blocked during a Load or Unload procedure and are unlocked after the procedure is completed. Due to a system error it may be that the axes are still locked and they can be unlocked manually via the button **Release X, Y, Z-axis**.

Button	Function
Reset PLC	Resets the PLC (Programmable Logic Controller). After the reset, the PLC is transferred into a safe condition. **Note:** During this procedure the user will be asked a number of safety questions. Confirm those with **OK**. The reset might take several minutes.

1.2.3 Service window

NOTICE

You can seriously damage the system and the load lock by using commands from the **Service** window without the guidance of Raith support.

The settings inside the **Service** window control the basic default system configuration. Never change any settings in the **Service** window on your own by using your own values and settings. Always follow the instructions from Raith support.

In the **Service** window you can change some system settings on the instructions from Raith support, in order to transfer the system into a safe condition after a system error has occurred.

When you open the **Service** window, the **Load Lock Schematic** window will be displayed as well (⇒ *Load Lock Schematic window* on page 18-24).

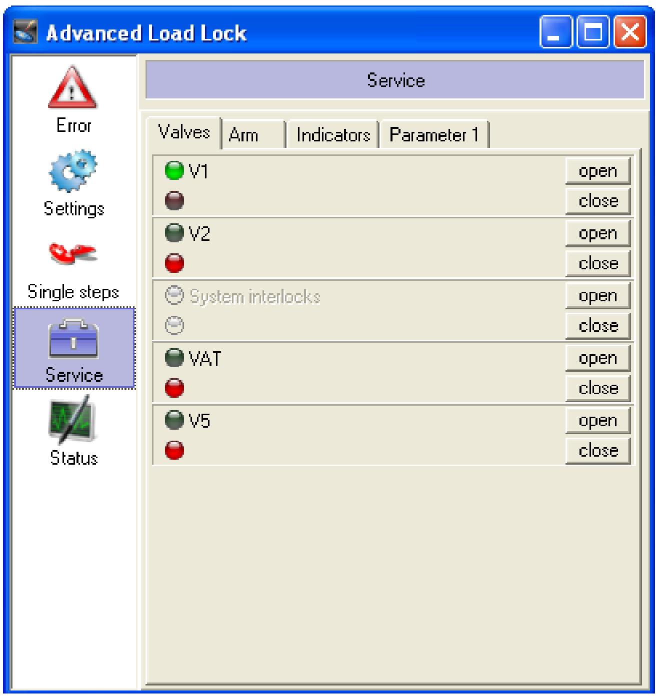


Figure 18-16: **Advanced Load Lock** dialog – **Service** window

The window consists of four tabs. The meaning of the individual commands and functions in the tabs will be explained to you by Raith support if a system error has occurred.

Load Lock Schematic window

The **Load Lock Schematic** window gives an overview of the basic structure of the RAITH150 Two as well as the position and function of the different valves and pumps. The valves are highlighted red.

The abbreviations of the valve and pumps are used in the **Advanced Load Lock** dialog (USR/SYS).

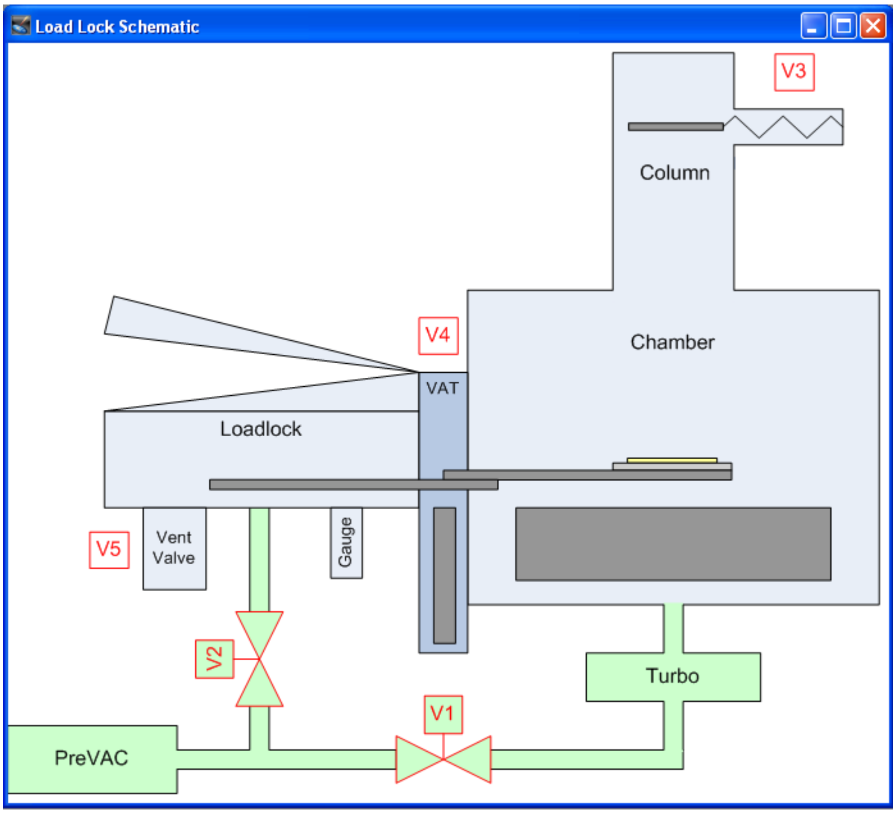


Figure 18-17: **Advanced Load Lock** dialog – **Load Lock Schematic** window

The abbreviations for the valves and pumps have the following meaning:

Valve/Pump	Function
PreVAC	Backing pump to create a pre-vacuum in the load lock.
Turbo	Turbo-molecular pump. Main pump to create the chamber vacuum.
V1	Valve between backing pump and turbo-molecular pump.
V2	Valve between backing pump and load lock.
V3	CCV valve. Valve between column and chamber.

Valve/Pump	Function
V4	VAT valve. Valve between load lock and chamber.
V5	Vent Valve. Valve between load lock turbo-pump and load lock.

Chapter 19
Service

This chapter describes how to parameterize the motor control and to initialize the coordinate system of the laser interferometer stage via the **Service** module.

The **Service** module is only available to authorized Raith service engineers.

All functions and settings of the **Service** module refer exclusively to Raith laser interferometer stages.

This chapter contains the following topics and tasks:

- Brief description of the purpose and dialogs of the **Service** module (⇒ *Service Software module* on page 19-2),
- How to parameterize the motor controls (⇒ *Parameterizing the Motor Control* on page 19-4),
- How to initialize the stage coordinate system (⇒ *Initializing the Stage Coordinate system* on page 19-4),
- Now to define the position of axis of rotation for tilt axis (⇒ *Tilt axis zero calibration* on page 19-7),
- Now to calibrate the tilt axis position with reference to the X, Y and W axis position (⇒ *Tilt axis calibration* on page 19-8),
- Now to calibrate the rotation axis position with reference to the X and Y position of a sample structure (⇒ *Rotation axis calibration* on page 19-10),
- Complete description of the dialogs, menus and control elements of the **Service** module (⇒ *Software Reference* on page 19-13).

A Functional Description

This chapter gives a brief description of the purpose and dialogs of the **Service** module.

1 Service Software module

The **Service** module serves to:

- initialize the coordinate system of the laser interferometer stage by defining the positions of the end switches and the home positions on the XYZRT axes,
- perform settings for the motor control on all axes.

The module consists of the following dialogs:

- **Motor Control** for the parameterization of the motor operation.
- **Initialize Coordinate System** to define the limit switches and home positions for all motorized axes of the stage.
- **RT Calibration** to perform the tilt and rotation axis calibration for RT module.

(⇒ Figure 19-1 on page 19-3) shows the both dialogs of the **Service** module.

For a detailed description of all control elements of the dialogs refer to:
(⇒ *Motor Control dialog* on page 19-13)
(⇒ *Initialize Coordinate System dialog* on page 19-19)

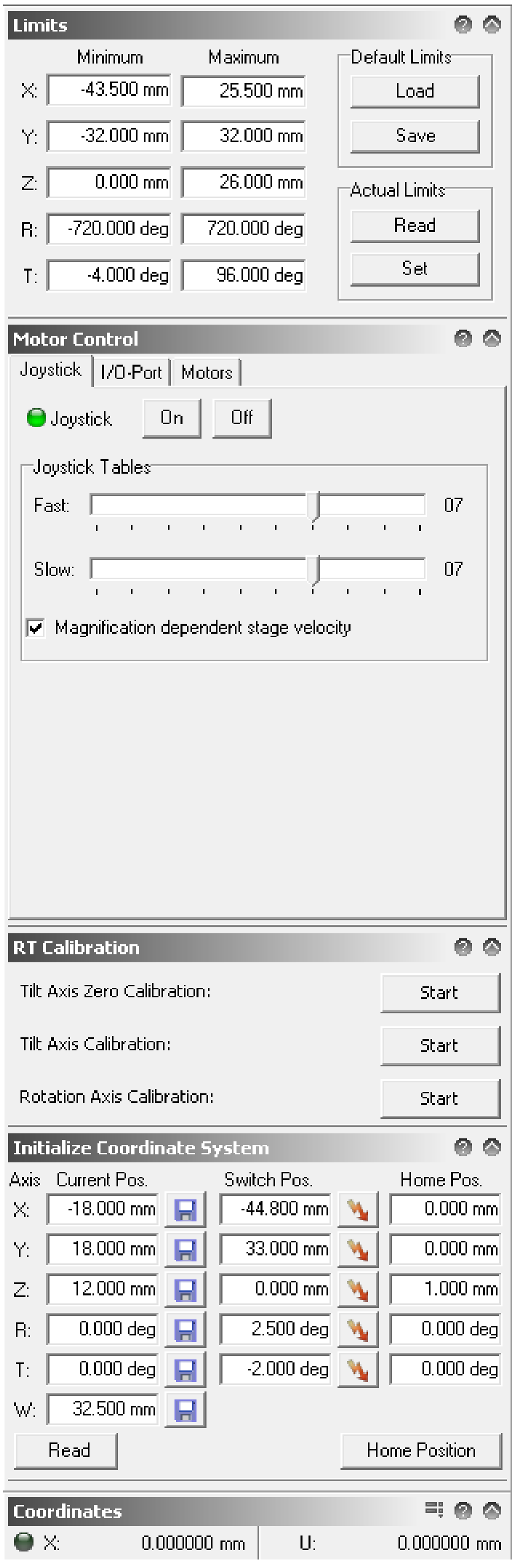


Figure 19-1: **Service** module dialogs

B Tasks

This chapter describes the following tasks:

- Parameterizing the motor control (⇒ *Parameterizing the Motor Control* on page 19-4).
- Initializing the stage coordinate system (⇒ *Initializing the Stage Coordinate system* on page 19-4).

1 Parameterizing the Motor Control

By default, the Raith lithography systems are equipped with DC tacho motors to drive the stage. The motors are directly mounted on drive spindles for the XYZ axes. On the XY axes, the motors are controlled via laser interferometer. Additionally, these axes are equipped with piezos for the fine-adjustment of the stage. In RAITH150 Two systems, the stage is equipped with three additional piezos for fine adjustment of the sample holders in the Z direction.

As an option, a Rotation-tilt (R/T) module can be used in some systems, equipped with its own motor control or using the same motor control that controls the XYZ axis. The R/T module is a removable substage with mechanical arrangements for two more degrees of freedom in rotation and tilt to allow sample inspection.

The travel range of the stage depends on the lithography system and can be up to 150 mm (RAITH150 Two system).

▶ To parameterize the motor control, configure the relevant settings in the **Motor Control** dialog as described in section (⇒ *Motor Control dialog* on page 19-13).

2 Initializing the Stage Coordinate system

The initialization of the stage coordinate system has to be performed

- during the initial commissioning of the lithography system,
- after exchanging a motor control or
- in case of misalignment of one or more motorized axes.

The stage coordinate system is the basis for the users to adjust the sample coordinates of individual applications as well as to adjust the coordinate systems of optional system components – e.g. GIS, Nanomanipulators.

During the initialization process, you define the positions for the end switches for the motors as well as the home position on the different axes. End switches and the Home position are the basis for the user to perform a **Find Home Position** routine in cases where the coordinate system has been inadvertently lost. The **Find Home Position** routine can be performed via the **Stage Control** module (⇒ *Stage Control* on page 8-1).

The software limits on the XYZ axes, which are defined in the **Administration** module ➡ **Limits** dialog will be deactivated during the initialization.

Initialize the stage coordinate system as follows:

STEP 1: Defining end switch positions

▶ Ensure that no axis is locked:

If one or more axes are locked, the corresponding LED in the **Coordinates** window will illuminate red. In this case, unlock the axes as follows:

- Change to the **Stage Control** module ➡ **Stage Lock** tab.
- Deactivate the corresponding checkbox.

In the **Coordinates** window, the LEDs will be switched off.

▶ In the **Initialize Coordinate System** dialog, click on to drive the stage on the X axis.

The corresponding input field in the **Switch Pos.** column, which will be highlighted yellow, changes to and the stage drives. The current position of the stage will be displayed in the **Coordinates** window. The corresponding axis LED illuminates green:

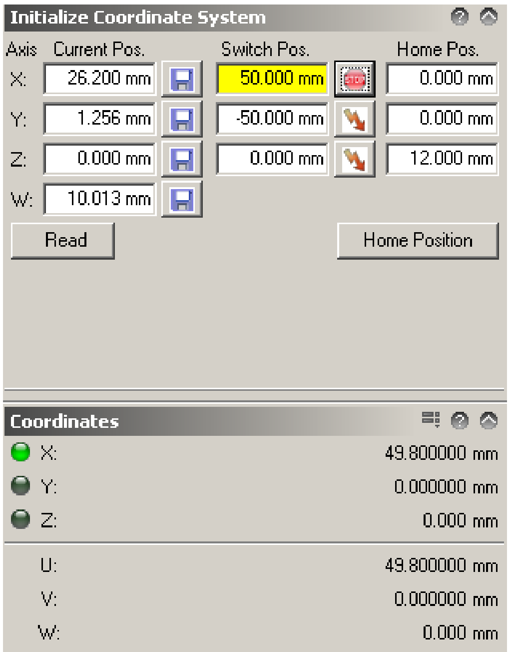


Figure 19-2: Defining end switch positions – stage driving

- Click on if the stage has reached the desired switch position on the X axis.
- Proceed as described for the Y and Z axes respectively.

- Alternatively, type in the end switch coordinates in one or more **Switch Pos.** input fields. Click on to drive the stage to the end switch respectively.
- If a R/T module is connected to the system, you have to define the end switches for the R and T axes as well. Both dialogs will, in addition, show the corresponding input fields and commands.

- Click on **Read** to read in the coordinate values of the current stage position.

  The coordinate values will be displayed in the corresponding input field in the **Current Pos.** column.

- Click on respectively, to save the coordinate value as an end switch position.

  The end switch position will be saved and is available for the **Find Home Position** routine in the **Stage Control** module.

STEP 2: Defining the Home position

- In the **Home Pos.** input fields, enter the coordinate values on the XYZ axes.

- Click on **Home Position**.

  All home positions will be saved and are available for the **Find Home Position** routine in the **Stage Control** module.

  The coordinate system of the stage is initialized.

3 Tilt axis zero calibration

STEP 1: Preparation

- Log in as user with SYS level rights.
- Ensue that R and T are initialized.
- Use a column setting which yields a small depth of focus, e.g. use a large aperture.

STEP 2: Calibration

- Go to the **RT Calibration** dialog and press **Start** button to open the **Tilt Axis Zero Calibration** Wizard.
  A start window will appear, showing the current calibration status and parameters.

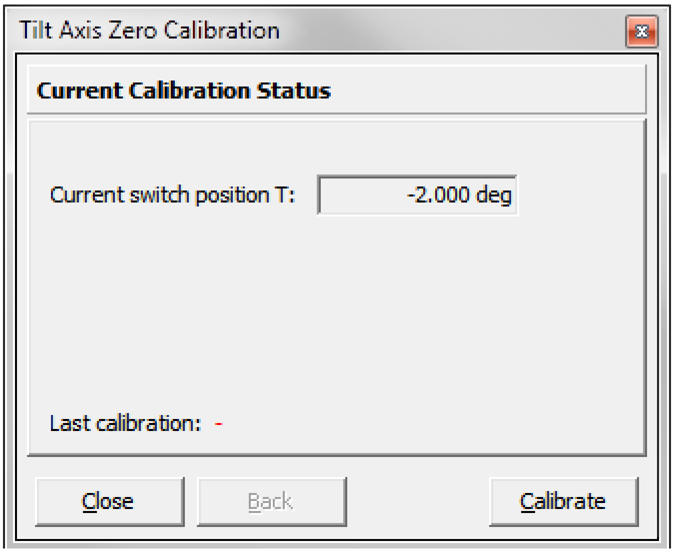


Figure 19-3: **Tilt Axis Zero Calibration** start window

- Press **Calibrate** button to start the calibration procedure.
- The next window will prompt you to drive the T axis to its current ZERO POSITION. Move the tilt axis to zero and press **Continue**.
- In the next steps you have to select three positions for working distance measurements. Choose the positions within an area of constant sample height and try to arrange them on a horizontal line perpendicular to the tilt

axis. You can also use the input field to drive the corresponding stage axis.

- When prompted, move to the first measurement position and press **Continue**.
- Focus on the sample surface, press **Read** and **Continue**.
- Repeat reading of working distances for other measurement positions.
- After the last measurement the new and old calibration results are presented. Press **Continue** to accept the new tilt position. The T position will be directly shifted with your acceptation.
- Close the Wizard dialog.

– If **Adjust switch position T** option is active, the acceptation will also shift the reference position for the **Find Home Position** procedure of the T axis. When unchecked, the last calibration will be ignored by the next coordinate system initialization.

---

4 Tilt axis calibration

STEP 1: Preparation

- Log in as user with SYS level rights.
- Ensue that R and T are initialized.
- Calibrate the **Tilt Axis Zero position** if not already done (this procedure is described in ⇒ *Tilt axis zero calibration* on page 19-7).
- Set the BEAM SHIFT in X and Y to zero. This can be made in **Fine Tune** dialog (see **Column Control**).
- Use a Column setting which yields to small depth of focus, e.g. use a large aperture.
- Adjust the W coordinate (chapter ⇒ *Adjust W tab* on page 12-18).

If you would like to use the eucentric drive mode during the wizard, it will move the axis based on the calibration parameters, which are currently activated. It is highly recommended to active the **Drive table** info display (see ⇒ *Eucentric tilt drive table* on page 8-9) for previous checking of the software intended movement.

STEP 2: Start the calibration

- Go to the **RT Calibration** dialog and press **Start** to open the **Tilt Axis Calibration** Wizard.

- A start window will appear and, showing the current calibration status and parameter.

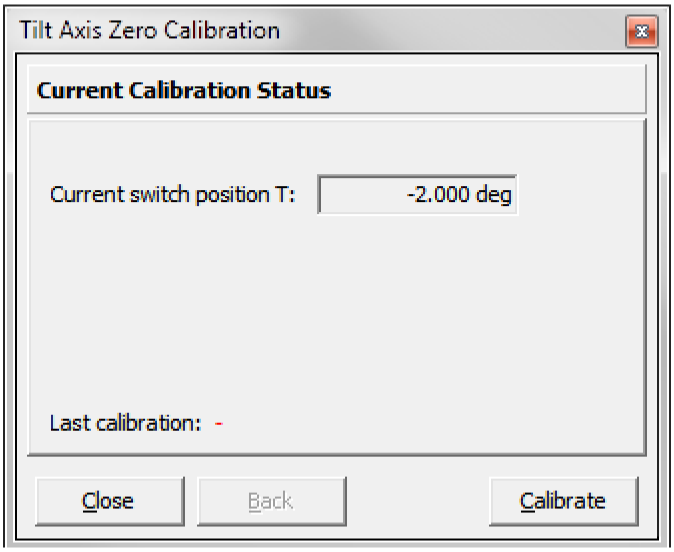


Figure 19-4: **Tilt Axis Calibration** start window

- Press **Calibrate** to start the calibration procedure.
- Move the axis to zero and press **Continue**.

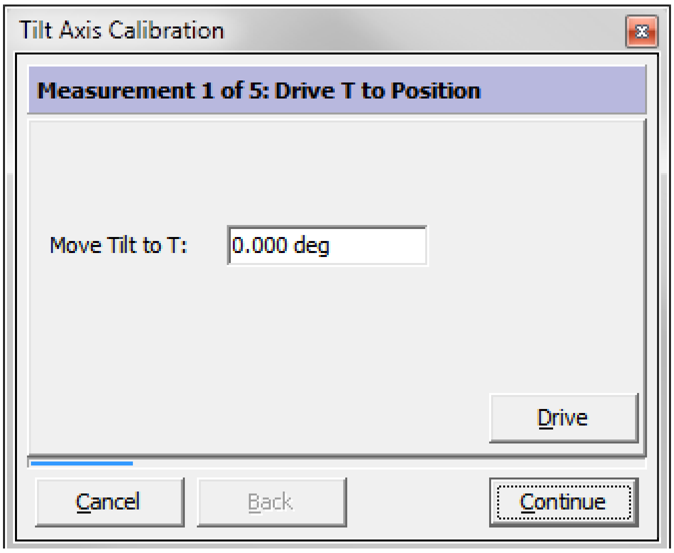


Figure 19-5: **Tilt Axis Calibration** first movement

- Focus on a characteristic structure and move it in the middle of the image. Press **Read** and **Continue** when finished.

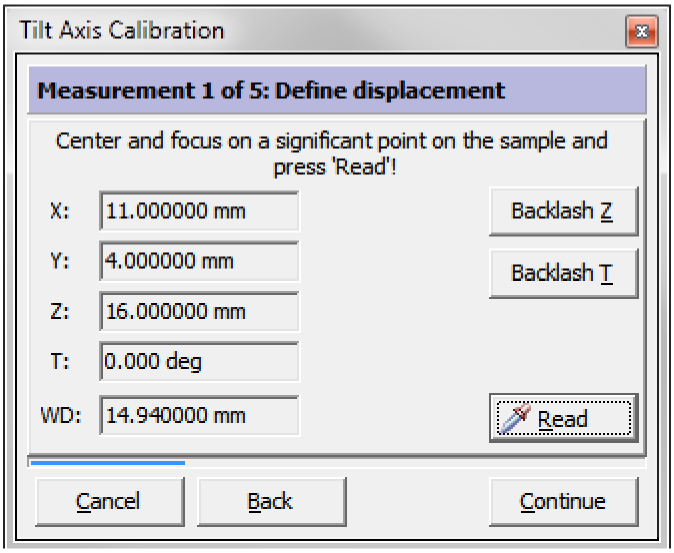


Figure 19-6: **Tilt Axis Calibration** first measurement

If you move the stage manually. e.g. by joystick, you can use the backlash button afterwards, to perform the backlash compensation at the current position.

- The T-axis has to be moved to the second measurement position. You can move it manually or eucentrically pressing the **Drive** button in wizard window. Press **Continue**.
- Center the structure in the image and refocus again. Press **Read** and **Continue** when finished.
- Repeat last two steps for other measurement positions.
- After the last measurement, new and old calibration results are presented. Press **Continue** to accept new results.
- Close the wizard dialog.

You have to readjust the W coordinate after the **tilt axis calibration**.

5 Rotation axis calibration

STEP 1: Preparation

- Log in as user with SYS level rights.
- Ensue that R and T are initialized.
- Set the BEAM SHIFT in X and Y to zero. This can be made in **Fine Tune** dialog (see **Column Control**).

STEP 2: Start the calibration

- Go to the **RT Calibration** dialog and press **Start** to open the **Rotation Axis Calibration** wizard.
  A start window will appear and, showing the current calibration status and parameters.

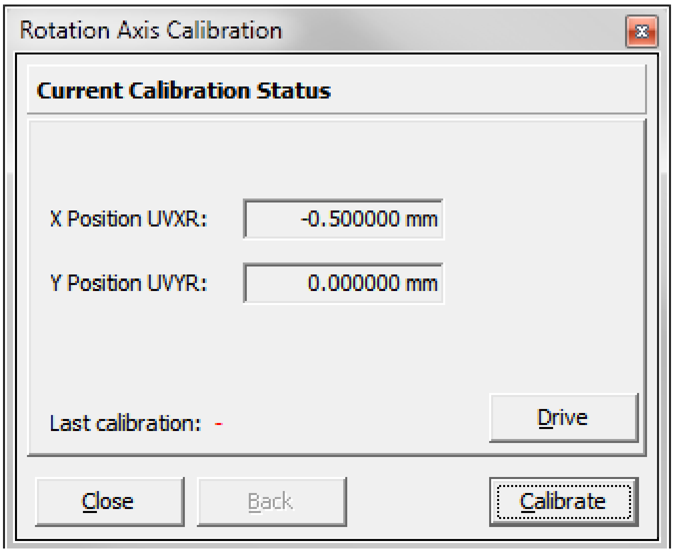


Figure 19-7: **Rotation Axis Calibration** start window

- Press **Calibrate** to start the calibration procedure.
- Depending on the current tilt axis position, you have to move to T to 0°. Press **Drive** to move T non-eucentrically. After that press **Continue**.
- Move the R-axis to zero and press **Continue**.

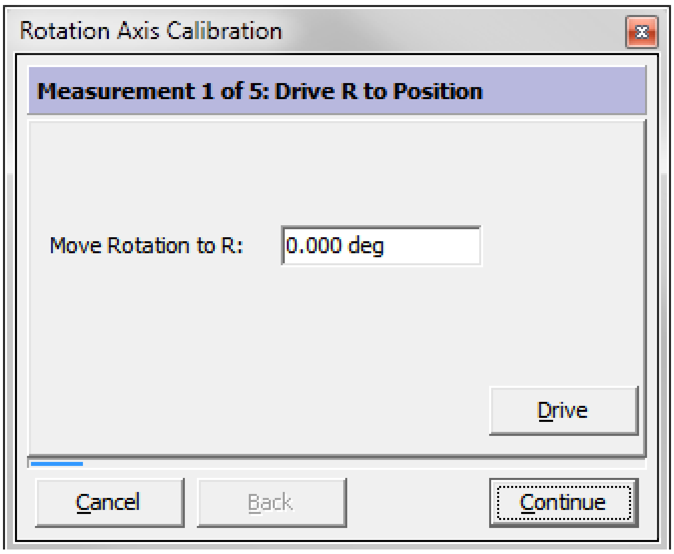


Figure 19-8: **Rotation Axis Calibration** first movement

- Move a characteristic feature in the middle of the image. Press **Read** and **Continue** when finished.

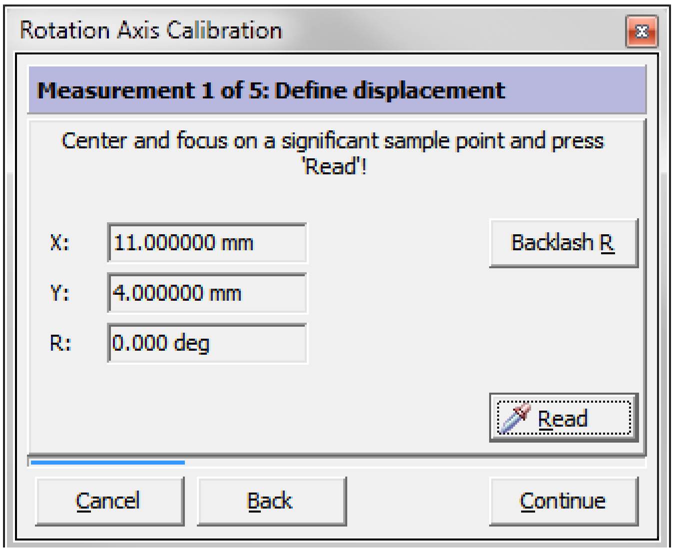


Figure 19-9: **Rotation Axis Calibration** first measurement

- The R-axis has to be moved to the second measurement position. You can move it manually or eucentrically pressing the **Drive** button in wizard window. Press **Continue**.

If you move the stage manually. e.g. by joystick, you can use the backlash button afterwards, to perform the backlash compensation at the current position.

- Center the structure in the image. Press **Read** and **Continue** when finished.
- Repeat last two steps for other measurement positions.
- After the last measurement, new and old calibration results are presented. Press **Continue** to accept new results.
- Close the wizard dialog.

C Software Reference

This section describes all functions and control elements of the Service module dialogs and subdialogs.

1 Motor Control dialog

The **Motor Control** dialog serves to parameterize the system motor control.

The dialog is divided into the following three tabs:

- **I/O-Port** tab to activate and wire the hardware components, which are controlled by the motors (⇒ *Motor Control dialog – I/O-Port tab* on page 19-13).
- **Motors** tab to set some parameters for the motor control (⇒ *Motor Control dialog – Motors tab* on page 19-14).
- **Joystick** tab to configure the stage velocity for stage movement via joystick (⇒ *Motor Control dialog – Joystick tab* on page 19-18).

1.1 Motor Control dialog – I/O-Port tab

The **I/O-Port** tab serves to view the assignment and the status of the input ports and to activate and deactivate the output ports. The corresponding motor bits can be named individually in the corresponding VDB file (**raith_app.db** file, section **Motion**).

The assignment of the I/O ports depends on the system configuration and the optional hardware components installed. The following screen shows the default assignment of RAITH150 systems.

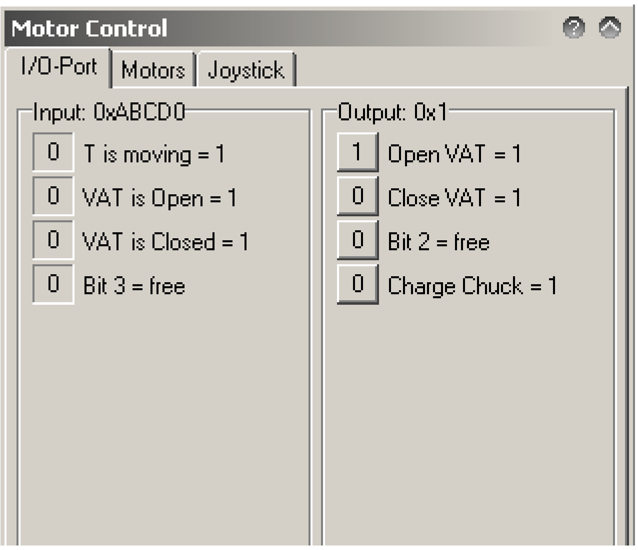


Figure 19-10: **Motor Control** dialog – **I/O-Port** tab

The **I/O-Ports** tab consists of two subsets.

- **Input** subset to display the assignment and status of the input ports.
- **Output** subset to activate and deactivate the output ports.

The motor bits and buttons of the input and output ports have the following meanings:

Motor bit/Button	Function
0 1	– Input port is deactivated and the function is locked. – Input port is activated.
0 1	To activate or deactivate the corresponding output port and enable the motor bit.

1.2 Motor Control dialog – Motors tab

The **Motors** tab serves to set motor parameters for each motor control on the different axes.

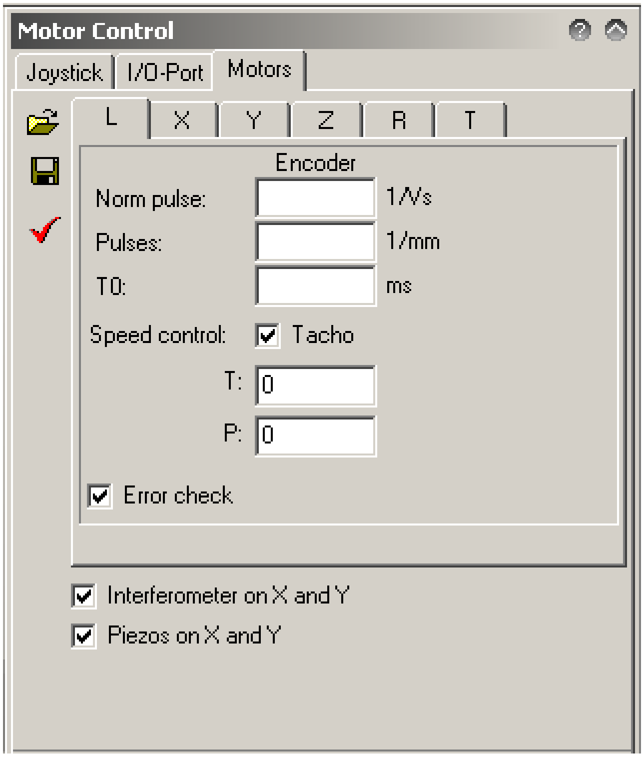


Figure 19-11: **Motor Control** dialog – **Motors** tab

The **Motors** tab is divided into the following subsets:

- **Tab** subset to set the parameters for the different motors:
  (⇒ *Motors tab – L, Z, R, T motor controls* on page 19-15)
  (⇒ *Motors tab – X, Y motor controls* on page 19-16)

- Tool bar and checkboxes which can be used in all tabs:
  (⇒ *Motors tab – Tool bar and checkboxes* on page 19-15)

1.2.1 Motors tab – Tool bar and checkboxes

The control elements have the following functions:

Control element	Function
	To load motor parameters from a file.
	To save the current settings in the currently loaded file.
	To send the motor parameters to the motor control. **Note:** A prompt asking will be opened to confirm the command.
Interferometer on X and Y	To enable the laser interferometer based position sensing in XY direction.
Piezos on X and Y	To enable the piezo position control in XY direction.

1.2.2 Motors tab – L, Z, R, T motor controls

The **L**, **Z**, **R** and **T** tabs serve to configure the encoder of the corresponding motor controls.

The **R** and **T** tabs are only attainable, if a R/T module is connected to the system.

The control elements have the following functions:

Control element	Function
Norm pulse:	To define the number of motor pulses per second normalized to an input voltage of 1 V.
Pulses:	To define the number of pulses per 1 mm travel range.
TO	To define a time constant in ms of the motor/piezo control.

Control element	Function
Speed control: Tacho	To enable/disable the speed control via tachometer. **Note:** Depending on the type of motor, the speed control will be performed as follows: – Motors with tachometer: Speed control via the analogue closed loop. – Motors without tachometer: Speed control via the firmware.
T:	To define the time constant in ms for the closed loop of a tacholess speed control.
P:	Closed loop gain for a tacholess speed control.
Error check	To enable/disable the error check function. **Note:** If **Error check** is enabled: The motor will be switched off, if the nominal voltage of 10 V will be overrun for more than 100 ms. Additionally, a corresponding error message will be displayed.

1.2.3 Motors tab – X, Y motor controls

The **X** and **Y** tabs serve to configure the motor controls for the X and Y axis. These motor controls are supported by laser interferometers and piezos. Therefore, all settings have to be configured for the encoder as well as for the laser interferometer.

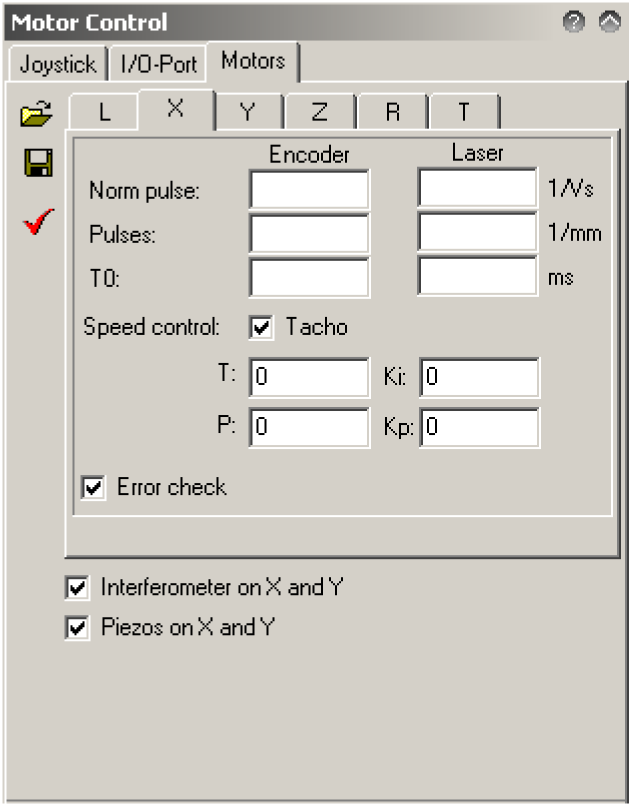


Figure 19-12: **Motor Control** dialog – **Motors** tab – **X** tab

In addition to the encoder parameters, the following parameters are to be set for the laser interferometer:

Parameter	Function
Ki:	To define the time constant in ms for the tacholess laser interferometer operating mode.
Kp:	Closed loop gain for the speed control in tacholess laser interferometer operating mode.

The encoder parameters are described in (⇒ *Motors tab – L, Z, R, T motor controls* on page 19-15).

1.3 Motor Control dialog – Joystick tab

In the **Joystick** tab you can activate the joystick and configure its velocity of movement on all axes.

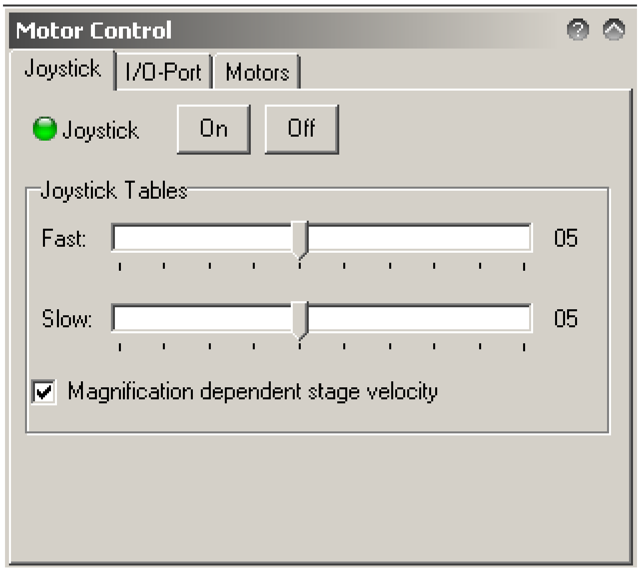


Figure 19-13: **Motor Control** dialog – **Joystick** tab

The control elements have the following functions:

Control element	Function
	Indicates that the joystick is enabled.
On, **Off**	To enable and disable the joystick.
Joystick Tables	Subset to define the joystick velocity:
Fast	To set the stage velocity stepwise for the **High Speed** mode (please refer to System basic documentation).
Slow	To set the stage velocity stepwise in the normal operating mode.
Magnification dependent stage velocity	To activate magnification-dependent stage movement.

2 Initialize Coordinate System dialog

In the **Initialize Coordinate System** dialog, you can initialize the stage coordinate system by defining the positions of the end switches and the home position on the X, Y and Z axes.

If a R/T module is connected to the system, the corresponding control elements for the R and T axes are displayed.

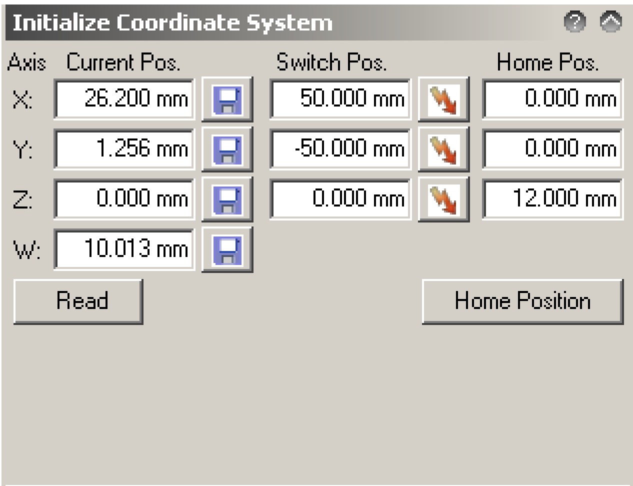


Figure 19-14: **Initialize Coordinate System** dialog

The control elements have the following functions:

Control element	Function
Axis X:, Y:, Z:, W:	Indicate the axis for which settings can be made in the adjacent input fields.
Current Pos. column	To display and define the current position of the stage – e.g. the current position of the motors on the XYZ axes.
W:	To display and define the working distance. **Note:** The working distance is the distance between the column and the stage or sample surface, determined in direction column to stage. Therefore, **W:** is calculated as the inverted value of **Z:**.
	To set the current position of the motors as end switch on the corresponding axis.
Read	To read in the current position of the stage from the **Coordinates** window. The coordinates will be displayed in the corresponding XYZW input fields.

Control element	Function
Switch Pos. column	To display and define the end switch position of the motors on the XYZ axis. **Note:** If the input field is highlighted yellow, the stage is currently moving.
	– Drives the stage on the corresponding axis. – Drives the stage to the end switch, which is defined in the corresponding **Switch Pos.** input field.
	Stops the stage on the corresponding axis.
Home Pos. column	To define the home position of the stage on the XYZ axes.
Home Position	To save the values in the **Home Pos.** input fields as home positions. **Note:** The home positions will be used for the **Find Home Position** routine, which can be performed in the **Stage Control** module (⟹ *Find Home Position* on page 8-23).

3 Tilt and rotation axis calibration dialog

The precision of the compensation movement depends on the accuracy of all involved stage axis and their calibration, as well as the sample height, determined by the W-Adjustment (⇒ *Adjust W tab* on page 12-18).

Three calibration procedures have to be performed to ensure best eucentric mode performance. They are accessible via the **RT Calibration** dialog.

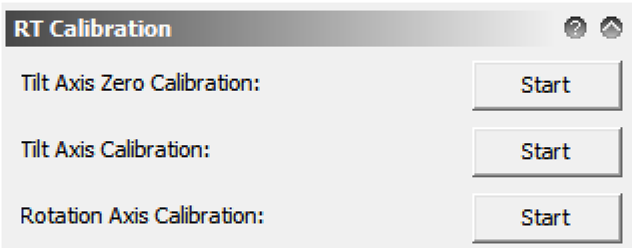


Figure 19-15: **RT Calibration** dialog

SYS level users can use the dialog to launch the calibration wizards.

The control elements have the following functions:

Control element	Function
Tilt Axis Zero Calibration	Calibration to determine the leveled tilt axis zero position. Redefines the current T position and can optionally shift the **Find Home Position** reference for enduring calibration.
Tilt Axis Calibration	Calibration of the tilt axis position by measuring the X or Y and Z position of a sample structure at five tilt angles.
Rotation Axis Calibration	Calibration of the rotation axis position by measuring the X and Y position of a sample structure at five rotation angles.

Chapter 20
Chipscanner (option)

The functionality of the Chipscanner is described here.

This chapter contains the following topics and tasks:

- Background information and reverse engineering applications
(⇒ *Overview* on page 20-2),
- The need for Chipscanning, the available techniques and the requirements which led to the development of Chipscanner are explained
(⇒ *The need for chipscanning* on page 20-2),
- Setting up **Create Preview** images for Chipscanner
(⇒ *Creating Preview images* on page 20-6),
- Other applications for Preview images,
(⇒ *Other applications for Preview Images* on page 20-8),
- Performing Chipscanning, explained step by step
(⇒ *Performing Chipscanning* on page 20-9),
- Detailed description of the **Scan Preview Image** dialog
(⇒ *Scan Image View dialog* on page 20-17).

A Functional Description

1 Overview

The Chipscanner enables reverse engineering applications.

Normally, a GDSII design structure is present and has to be exposed on a resist coated substrate. After development, a chip can be created, after some optional chemical processing steps. In reverse engineering, when a structure is already present on a sample, an image is created and the image is then applied to the GDSII design, a reverse engineering application.

Chipscanning is used to scan the surface of a chip using **Images**, initiated from the Positionlist. A chip or a structure will be already present on the sample surface, which will be acquired via an **Imagescan**. Generally, the structure will be larger than a writefield, so a sequence of imagescans will be required, to be stitched together afterwards. The first imagescan will be carried out and then the stage will be moved by the dimension of the writefield size and the next imagescan will be carried out and so on, until the whole structure has been scanned.

After the images have been scanned, there are several options:

- Using **Scan Image Preview**, the matrix of individual images, stitched together, can be viewed. The images are still individual files, with no combined image file created.
- Using **File➡Save as high resolution**, one single bitmap file can be created, summing up all pixels from all individual images. This bitmap file is for printing only. Special software may be required to open the file, because of its size.
- For **Chipscanning**, each individual image can be converted one by one to GDSII, followed by the creation of one single GDSII file.

2 The need for chipscanning

There are many ways to approach the task of reverse engineering of semiconductor devices. Almost all methods require a layer by layer deprocessing, so that circuit construction and material makeup can be brought into view for in-depth inspection.

One popular technique is to capture the exposed layer by collecting a series of micrographs and assembling them such that complete layers can be examined and evaluated. Since each magnified image covers only a small portion of the total device, the re-assembly step becomes one of the most critical tasks in this work.

Using chipscanning, the individual images are captured, the sample positioned and the images are then processed for conversion back to the original design layout. Only after the conversion of each individual image to GDSII, will one single GDSII file be created.

2.1 Thermal Field Emission optics

It is well known that design rules of modern devices can easily exceed the practical use of an optical light microscope for detailed examination. In most labs and semiconductor manufacturing facilities the use of electron optic systems is routine. The resolution required for examining imperfections on the surface and in cross sections of advanced devices can place excessive demands on many electron optic systems. The instrument of choice in today's facilities is the field emission system. These instruments can acquire an image with a resolution better than 2 nm. Another valuable feature of the Raith TFE system is its low voltage operation. In this mode, primary electrons will not penetrate very far into the surface or build up a charge on samples with low conductivity. The final imaging result from such an instrument is ideal for the inspection of large scale integrated circuits and memory arrays.

The size of a typical image at high resolution can change, but in all cases, no electron optic system can give a clear image of the fine details of an IC in one scan field. It therefore becomes necessary to look at only small portions of the device at one time. To examine the entire chip at high resolution, it becomes necessary to collect a series of high magnification images and build them together, forming a collage of micrographs that will ultimately be combined as a large scale mural of the device. The creation of each complete image and its relationship to its adjacent partner is a limiting step in the reverse engineering process. Using Chipscanning, each individual image can be converted to GDSII and then one single GDSII file can be created.

The normal size of a high resolution, high magnification image is in the range of 20 to 200 microns. Considering that a typical device is 10 mm x 10 mm in size, then 2,500 to 250,000 images must be collected to capture the entire chip, and this is only for one layer. There is no doubt that the stitching process is too tedious to be done manually, and a laser interferometer stage is required.

Accurate stitching can be handled in many different ways. One popular method is to allow each field to overlap its neighbor in order to avoid gaps. These overlaps can then be removed by advanced software routines. The more overlaps there are, the harder it becomes for the routines to recognize and remove them. Since gaps and overlaps in the field boundaries are a result of errors in stepping the sample under the final lens of the microscope, a better method is to improve the accuracy of the sample stage so that these errors are virtually eliminated.

2.2 Laser interferometer stage

There are many types of sample positioning stages on the market today. Most motorized stages can be controlled by computer, but the accuracy in their positioning can vary from several microns to about one tenth of a micron, depending on the technology used. Since most applications require no more than several microns in positioning accuracy, optical encoders or motor pulsing is an inexpensive and popular method for stage control. To reach the scale of sub-micron positioning accuracy, however, it becomes necessary to use a laser interferometer to read the current position of the mechanical stage.

Reading the stage position is not the only limiting factor for accurate control. Moving the stage to an exact position becomes limited by ordinary motor control, and thus in some cases a piezo-electric aspect is needed to bring the stage to its final destination.

It should be remembered that the ultimate task is to carefully acquire high resolution images that fit together with almost no stitching errors. This requires accurate stage control, as described above, but it also requires an accurate calibration and registration of each scan field.

2.3 The digital scan generator

By means of software routines and interactive instrumentation, it is possible to accurately synchronize the size and orientation of each scan field to the stepping period and direction of the stage motion. Software control of image scans can allow pre-selected resolution and pre-determined placement of each scan over the sample surface. Digitized images are easily handled and automatically managed through software, so that reorganization and stitching of the images does not need to be done manually.

The method above is at the heart of the Chipscanner. It is the integration of hardware components and a comprehensive software package to make a flexible and user-friendly system. The functionality of such a system encompasses many applications, not only reverse engineering.

2.4 Image to CAD conversion

The captured images of any layer can be converted from a standard <***.bmp**> or <***.tif**> format to GDSII, a file format generally used in the semiconductor industry. Once each individual image has been converted to GDSII, a single GDSII file will be created.

By selecting a large grid of pixels to superimpose on an image, the user can set intensity criteria that will convert the gray scale image to a binary array of black or white pixels. Further filtering and processing will eventually yield a pattern that closely resembles the circuit layout of the image in process. Organizing and storing each converted tile in the array of images to the GDSII format completes the conversion process. The converted images can be checked more carefully when overlaid on the original bitmap image. In many cases, it would be useful to return to a stage position for collecting a new image to replace the original one that may have an unwanted anomaly. To implement this, the Chipscanner system has a unique way of saving the coordinate information with each image.

In addition, the stage control allows navigation from any GDSII file, a technique generally called CAD navigation.

B Tasks

1 Creating Preview images

- Go to **Extras➡Settings➡Image➡Create Preview**. Ensure that the **Preview** is set to **On** and set the resolution for the image preview. If it is not set to On, **Create Preview** will not function. The default setting is **Off**, so it is important to change the setting. Define the width and height of the preview image. The preview image is an additional image with file name extension <***.prv**>. It is created together with the common <***.ssc**> Raith image file. The preview image file size can be equal to, or smaller than, the corresponding image file and is used in the **Scan Image View** module.

- The scan resolution can be set by the user, e.g. 512 x 512 pixels, using parameters **Preview Image Height** = 512 and **Preview Image Width** = 512, in the **Settings** dialog. Only when all of these parameters are set, should you proceed to the **Scan Manager**.

The resolution of **Preview Image** only affects the preview of the image. The intrinsic image resolution is defined in the **Scan Properties** dialog of the **Scan Manager**. Using a low resolution for Preview Image simply allows a quick overview.

Data evaluation is always performed on the basis of data stored in the corresponding <***.bmp**> file, which is created after each image is scanned.

Create preview is only for image viewing. The preview images are not used for any data processing.

Scan Image View will display the single images, stitched together in a matrix, but a single file will not be created.

The user also has the option to save a high resolution image via **File➡Save as high resolution**. This will create a very large bitmap file, which is the sum of all individual scanned images. This high resolution image is for printing. It can not be used for chipscanning.

For **Chipscanning**, each single image is converted to GDSII and the stitching will then take place, to create a single GDSII file.

- Define an image within the **Scan Manager**, for example an image with a size of 100 µm x 100 µm for a 100 µm x 100 µm writefield. Drag and drop the image into the Positionlist.

- Go to **Filter➡Matrix Copy** to define the overall area, e.g. 5 x 5 writefields, the size of each writefield needs to be identical, so that one image can be placed alongside the next image. The image should have roughly the same dimensions as the writefield. In this example, a step in the **Matrix copy** dialog of about 98 µm should be used, so that adjacent images are scanned with a slight overlap afterwards. This method ensures that the area is completely within the chip at the stitchfield borders.

- Execute the scanning of the full Positionlist by clicking on **Scan All**. The images will now be scanned and stored under a self-explanatory name.

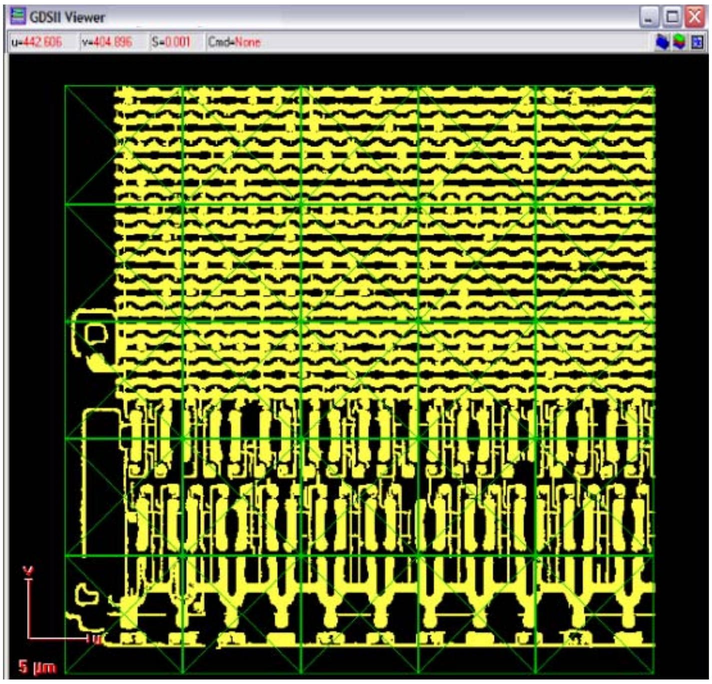


Figure 20-1: Image to GDSII result

- Once the scan is completed, go back to the Positionlist and apply the **Positionlist Filter Image to GDSII.** This filter will create GDSII structures based on images stored in the selected Positionlist. This process is called reverse engineering, as normally a GDSII design structure is applied to an image, but in reverse engineering an image is applied to the GDSII design.

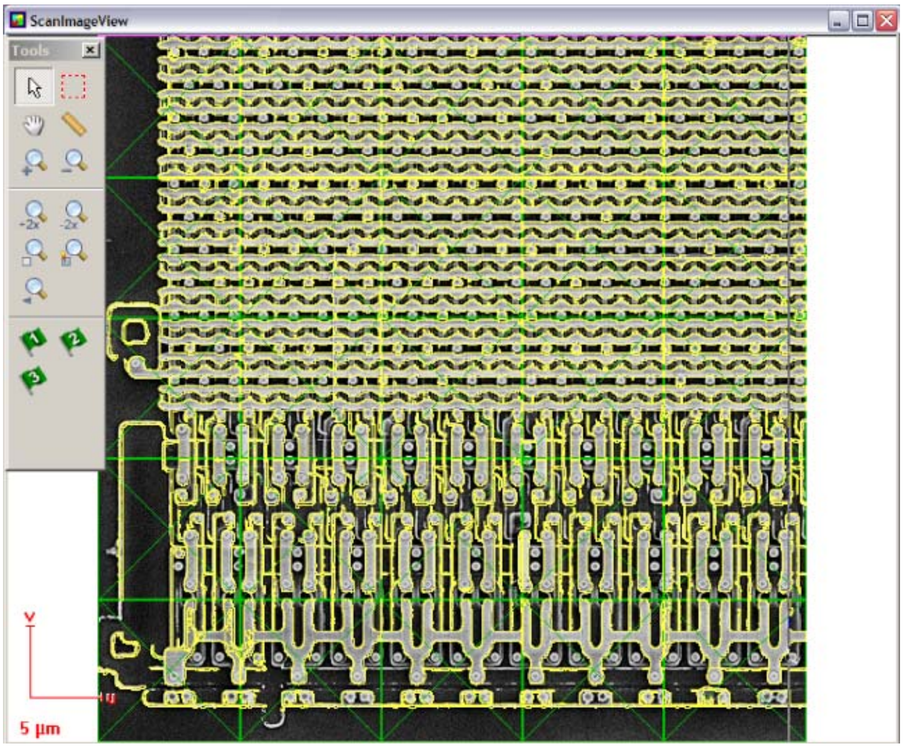


Figure 20-2: Scan Image View with overlaid GDSII

2 Other applications for Preview Images

The **Preview Images** option can also be useful for applications other than chipscanning for reverse engineering. For example, when a large number of chips are patterned and the user would like to see an overview of all chips, a preview image can be created. This will aid checking that the marks are at the expected positions. It can be used as a generic quality check.

For example, if the GDSII design does not fit the previously created mark, this can be easily identified using Preview Images. Alternatively, if a very large structure has to be written, an overview can be very useful.

- Use **3-Points Adjustment** if you wish to carry out an overlay with a GDSII structure. The **Overlay** can be performed in the UV coordinates system.
- The order should be always as follows:

STEP 1: Perform adjustment

- Perform adjustment, such as **Origin Correction**, **Angle Correction** and **3-Points Adjustment**.

STEP 2: Perform alignment

- Perform the **Writefield Alignment**.
- Create and execute the Positionlist with a matrix of imagescans covering the area of structure to be visualized, (⇒ *Imagescan Options* on page 10-39).
- Open **File➞Scan Image View** to view the preview images.

3 Performing Chipscanning

In this chapter, the sequence of steps for a single layer Chipscanning procedure will be described in some detail.

To meet the requirements of this procedure, the sample to be investigated must be loaded and located and the system must be in normal operation. Complete operation can be done via a dual graphical user interface, one for operation of the electron optic system, the other for dedicated chipscanning applications. Described below are the individual steps for a typical application, run in interactive mode.

STEP 1: Sample preparation

Preparation of the sample is very important for retrospective CAD analysis of chips, particularly de-layering processes.

The sample should be mounted carefully on the large area sample carrier, preferably roughly oriented along the geometrical XY coordinate system of the carrier.

- De-process your sample to the layer of interest. It should have the same height all over the die.
- Use either mechanical polishing, chemicals or a combination of both.
- If you need to work on a strongly charging layer, try to sputter it with gold or other suitable coating.
- Try to fix it on the sample holder so that charging can be avoided, with good contact to the sample holder through the reverse side, or even better, with an additional metal clamp from above).
- Via layers can be extracted from other layers. It is not always necessary to process them separately.

STEP 2: Performing the stage adjustment

Firstly, a **Deskew** operation must be performed - along any visible line close to the X axis of the system.

Please refer to the chapter **Stage to Sample Adjustment** for further explanation of the **Deskew** and **Adjustment** procedures, (⇒ *Coordinate systems and transformations* on page 12-3).

Next, it is recommended to define a sample origin using the **Origin Correction** tab of the **Adjust UVW** window, for example at the lower left of the chip or the area of interest. This is quite important if you plan to investigate more than one layer of a chip, or to repeat the job later on. The site defined in this procedure will be the reference point for any subsequent inspection runs on the same chip.

- ▶ Use electronic scan rotation to obtain an orthogonal image.
- ▶ Drive along one line to the left side of your sample and read this point as the first adjustment point using the **Angle Correction** tab of the **Adjust UVW** window.
- ▶ Drive along the same line to the right side of your sample and read the second adjustment point.
- ▶ Drive to, for example, the bottom left corner of the area that you would like to scan and set U, V to **0, 0**.

STEP 3: Selecting the column parameters

Within this step, the precise scan field needs to be adjusted to correspond exactly to the laser stage metrology.

Please refer to the chapter **Beam to Sample Alignment** for further explanation of the **Writefield Alignment** procedures. This chapter also explains the relationship between adjustments and alignment routines using diagrams, (⇒ *Coordinate systems and transformations* on page 12-3).

As a first step, we recommend setting the scan rotation so that the displayed image shows the lines to be oriented approximately horizontally and vertically.

- ▶ Switch off the automatic brightness and contrast functions for each detector.
- ▶ Set the High Voltage as low as possible. A value in the range 0.5 kV to 5 kV is usually appropriate.
- ▶ Try different detectors, (InLens, SE, BSE, InLens+BSE, SE+BSE, SE+InLens) to achieve the greatest possible contrast. Structures should be white, background should be black.
- ▶ Try different apertures. The smaller the aperture, the smaller the beam current, and so the lower the charging.

STEP 4: Selecting the imaging parameters

Within this step, a typical area of the chip to be inspected should be imaged with the electron optic system, in order to find the image acquisition conditions best suited to the task.

The criteria used to judge which are the best conditions are:

- Sufficient image contrast between the lines and spaces of the layer to be converted
- Low intensity background signal from underlying layers
- Small contrast variations within the structures themselves
- No charging and stable image conditions
- Sufficient signal to noise ratio.

The main electron optics parameters to be used for these optimizations are:

- Selection of the most suitable detector (InLens, standard ET, backscattered electron detector)
- Selection of the beam current/image resolution.
- Contrast and brightness settings should be manually adjusted to allow good discrimination between the gray scale intensities of lines and spaces.

Before starting the measurement, it is important to inspect and optimize the image quality with respect to resolution and contrast. In this step, fine tuning of the settings, as mentioned in Step 3, section **Select Imaging Parameters**, is essential. Also check the image contrast after acquiring a digital image by choosing the menu commands **File➡New image** and **Imagescan➡Single** with the same settings and then choose the command **Slowscan➡Histogram**. The intensity distribution should fit into the range. If not, a re-adjust the brightness/contrast.

In order to allow a good CAD data generation, the **Scan Conditions** should be selected according to the following criteria:

- Set the image resolution to a value so that the smallest distance or feature size has at least 5 pixels.
- Typical scan field resolutions are between 256 and 1024 pixels.
- Typical writefields are from 100 µm down to 15 µm: e.g. for a CD of 250 nm, one pixel corresponds to 50 nm. You can therefore use, for example, 1024 pixels with a 50 µm writefield or 512 pixels with a 25 µm writefield.
- Point by point average should have a value of at least 2, otherwise the scan speed will be too fast, leading to deterioration and distortion of images.

- Line by line and frame by frame averages are not very useful.

Example and rules for image parameter definition
Assuming a critical dimension, or minimum linewidth, of 1 µm, the pixel distance within the image acquisition runs should be somewhere between 0.2 µm and 0.25 µm, which corresponds to between 5 and 4 pixels per smallest line or feature.
We could, for example, define the image step size as 0.2 µm. Based on this selection of the image resolution, the image size in terms of pixels and magnification/scan field can be selected.
As a general rule, the scan field, or image size, should be within the following guidelines:
Image Size = 500 x (Image Increments) = 100 x (Minimum Linewidth), or
Image Size = 1000 x (Image Increments) = 200 x (Minimum Linewidth)

In this example, the image sizes should be between 100 µm and 200 µm. Selecting 200 µm corresponds to a magnification setting around 500x, and 1,000 pixels, defined in the **Writefield Manager** dialog.
Smaller image sizes would result in enhanced resolution within the image acquisition and conversion process. On the other hand, for extended chip sizes, the measurement time and the amount of data to be handled for a run can become excessive, so a compromise needs to be found.

The next step is to carry out the **Writefield Alignment** procedure.

STEP 5: Performing the Writefield Alignment

Open a new Positionlist. In the **Scan Manager** dialog, select an appropriate **Writefield Alignment** procedure and drag and drop this procedure into the Positionlist. Writefield Alignment will be carried out as soon as the Positionlist is executed. (⇒ *Writefield Alignment procedures* on page 11-33).

This procedure requires a small feature to be centered on the screen. The matching sequence normally should be executed in manual mode, starting from relatively large mark scan areas, perhaps 20% of the scan field. After the first run, the mark scan ranges should be decreased stepwise down to a few microns. Typically, 3 iterations may be used to get to a 0.1-0.2 µm calibration of the scan field.

- Find a small (<200 nm) point of interest, such as a particle or contamination dot and center it.
- Set a writefield with the **Writefield Manager** dialog. Magnification should correspond quite accurately to the field size that you would like to use for image scanning.
- Repeat the sequence with smaller scan fields until no decrease in the correction factors is observed.

- Take a normal image, check the real size and write it down.

STEP 6: Positionlist setup

After the writefield alignment procedure is implemented, complete the Positionlist.

- Drive to U,V = **0,0**
- Reset the image counter and define a self-explanatory name for the images.
- Now place the first image via drag and drop from the **Scan Manager** into the **Positionlist** and perform a test scan.
- Inspect the test scan after it has been scanned by double clicking onto the line in the Positionlist. The acquired image will be displayed and you may check it with respect to location and image quality.
- Check the exact size of the image, by choosing the command **Edit➡Image information**. This size may differ slightly from the size defined in the **Writefield Manager** dialog - depending on the ratio of pixel resolution and image size.
- Calculate the optimal image distance with a small overlap.
- Use the **Matrix copy** filter to create the image matrix.
- Place additional macros, such as to save the Positionlist or auto convert, within the Positionlist.
- Save the Positionlist under a self-explanatory name.
- If you use the auto conversion macro make sure that, before you start the total scan, you have defined acceptable GDSII conversion parameters.

After successful completion of this step, save and close the Positionlist.

- Now start the data acquisition by choosing the command **Scan➡All**. Take care that all checkboxes, except the option **Show Info**, have been checked.

The scan automatically moves the stages, starts the image acquisitions at the individual locations and stores all images onto the hard disk. Each scanned image will temporarily be displayed and then removed from the screen.

- After scanning has been completed, the Positionlist should be stored.
- If required, you can obtain a tiled overview of all the images, by choosing the command **File➡Open Scan Image View**.

The displayed image preview consists of all images, reduced in pixel number, and tiled in a UV representation. A double click on any image tile gives access to the full corresponding image. Since both the individual images, as well as the tiled previews, are real UV windows, all related navigation tools are available, as described in **Wafermap** and **Defect-map**.

The Positionlist is now prepared for the next step, image optimization.

STEP 7: Optimize images

Choose the command **File➡Open optimize images** and select the following checkbox options, if required:

The **Gray level** option increases the contrast of all images in the **Positionlist**. This is recommended for each Positionlist before conversion, since it makes it much easier to find the threshold values for conversion.

The **Low pass** option eliminates single pixels with another gray scale from the surrounding area. It should be noted that this function decreases the image sharpness (i.e. the edge sharpness of the structures).

The **Merge images** option removes overlaps at the image level. It deletes the pixels at the edge of one image, if there is an overlap with other images.

The **New previews option** allows you to re-create new previews with any resolution up to that range used for the original images.

STEP 8: Conversion to GDSII

The functionality of data conversion into CAD files is described in detail in the chapter **Positionlist Filters**, section **Image to GDSII**. During conversion, a hierarchical GDSII CAD file will be generated, having the same name as the Positionlist itself. This file is organized as a main structure, with the same name as the Positionlist, referencing all the converted images as substructures, or structure references.

Please read **Positionlist filters, Image to GDSII**, for further information, (⇒ *Image to GDSII* on page 9-57).

- If only parts of the list need to be converted, or a group of positions should be converted in a second step, for example with different parameters, the group selection features of the Positionlist allow such operations very easily.
- Within the conversion routine, there is a check implemented, which assigns a warning, if GDSII elements with more than 2048 vertices are generated. In such cases, excess coordinates are actively removed from the data in order to meet the standard GDSII requirement. As this action sometimes can result in irregularities within the converted and stored structures, this image conversion is highlighted in the Positionlist by a preceding red marker in the respective line. A manual inspection of the

red-marked lines is recommended after conversion of the whole list has finished.

- The created GDSII data file can be opened for inspection. To begin, it is useful to select the main structure and inspect the details by zooming, increasing hierarchy selection etc. from this starting point.

Please read the Chapter **GDSII** for further information, (⇒ *GDSII Data Handling (Design Module)* on page 5-1).

- Any opened image of the acquired sequence will be automatically overlaid at the correct coordinates onto the **GDSII Viewer/Editor**, if the option **Options**➡**Show video** is selected.
- A right mouse click within the **GDSII database** module area accesses standard database operations, such as duplicating, renaming structures etc. There is also a copying command available under the **Structure** menu, which allows the transfer of hierarchical or flat structures between different databases.
- If critical areas should be re-inspected, this is possible by using **Ctrl** + **right mouse** button on the pattern of interest.
- Check the **Scan Image View** matrix for brightness, contrast, focus or particle charging problems
- Try to optimize the images within the matrix by using a gray scale or low pass filter.
- Convert some images, at least one at the bottom left side and one on the upper right side, and try to find the best parameters by experimentation.
- Convert all images within the Positionlist. This will be automatic if the **AUTOCONVERT** macro is used within the Positionlist.

STEP 9: Optimizing GDSII

The following post processing tools are accessible via the **Tools** menu within the GDSII Database module window.

- Use any of the GDSII tools, when you think they will optimize the result, but always make a backup of the GDSII file beforehand.
- Typical tools recommend include UNIFY, SMOOTH, OVERLAP, MERGE and SEPARATE.
- Use the GDSII editor to change single polygons and delete, for example, converted dust particles.
- If you wish, you can combine the different GDSII designs for each layer within one GDSII file, for example to overlay them.

Choose the **Unify Structure** tool in order to convert the GDSII files, organized as main structure plus referenced, tiled, substructures to one common flat file, for each scanned layer. This adds the combined flat structure to the existing database as a separate item with a name to be defined by the user.

The **Merge Elements** tool allows the closing of gaps within the structures, such as at stitch field boundaries. The user must enter the linking radius in nm. All substructures in the selected file which have data points at a distance from each other that is smaller than the specified radius, will be connected.

Smooth Elements

This tool allows the reduction of the roughness of structures, since it may result from the image to CAD conversion process. The smoothing range (Max range for smoothing) has to be defined. This range should be defined as half of the smallest features size.

C Software Reference

1 Scan Image View dialog

The **Chipscanner** is not an individual module within the software, which can be called up. Instead, it represents the process of reverse engineering, which calls up a number of software modules. Please refer to **Task B, Performing Chipscanning**, to obtain a detailed overview of which software modules will be used for the chipscanning process. Links to the relevant chapters are given in the task, (⇒ *Tasks* on page 20-6).

The **Scan Image View** dialog can be opened via the main menu while a Positionlist is open and selected.

2 Main menu

The following menu items are available.

File menu	Function
Save as	When **File➡Save** is selected, the preview image will be saved either as a <***.bmp**> or <***.tif**> file with the resolution as currently displayed on the monitor, including the overlay.
Save as high resolution	This will create one very large bitmap, the sum of all pixels from all images. Please note that special programs may be required to open and process such large bitmap files. The high resolution bitmap is used for printing, not for chipscanning.

It may be that the image size is larger than displayed on the monitor. The resolution is set in **Extras➡Settings➡Image➡Preview image height ➡Preview image width,** where the image height and width are defined. If the resolution is set to 512 x 512 pixels for example, the preview image quality will initially be high.
If a resolution of 128 x 128 pixels has been entered, the pixel resolution will be much lower in the preview image format.
Even when preview resolution is low, the user can still save higher resolution images via **File➡Save as high resolution.** The software will now create one large preview image file, from a number of stitched imagescans at their original resolution, as defined in the **Scan Manager**.

Chapter 21
GIS (option)

This chapter gives you an overview of:

- The principle of patterning with electron beams (EBID) or ion beams (IBID)
  (⇒ *Principle of Patterning with electron beams* on page 21-10)
  (⇒ *Principle of Patterning with gases and ion beams* on page 21-11).
- The operating mode of the GIS
  (⇒ *Functionality of the GIS* on page 21-12).
- Process rules and hints for the optimum parameter selection of temperature, system vacuum and patterning
  (⇒ *Process rules* on page 21-14).
- How to change settings and configurations in the SYS-level
  (⇒ *Workflows in SYS-level* on page 21-28).
- How to define set temperatures for reservoirs and capillaries in EXP-level (⇒ *Workflows in the EXP-level* on page 21-40).
- How to perform a patterning process
  (⇒ *Preparing and starting a patterning process* on page 21-24).

A Functional Description

1 System description

This chapter provides the user with an overview about the assembly, the function, and the installation requirements of the Gas Injection System.

1.1 Operational overview

The Gas Injection System (GIS) is an option for the Raith NANOSUITE systems eLINE, ionLINE and RAITH150 Two. It serves as an additional option for nano-structuring of samples by adding a gaseous precursor to the electron- or ion-beam. Introduction of these precursors makes it possible to perform local etching and deposition processes on suitable samples, in order to structure in the nanometer and micrometer regime.

The GIS consists of the following components:

- GIS hardware (GIS),
- Gas injection controller (GIS-Controller),
- GIS software module (GIS-Module).

The control of the GIS is fully integrated into the NANOSUITE Software of the Raith lithography systems and is exclusively carried out via software functionalities.

1.2 Hardware overview of the GIS

The GIS is assembled in the following way:

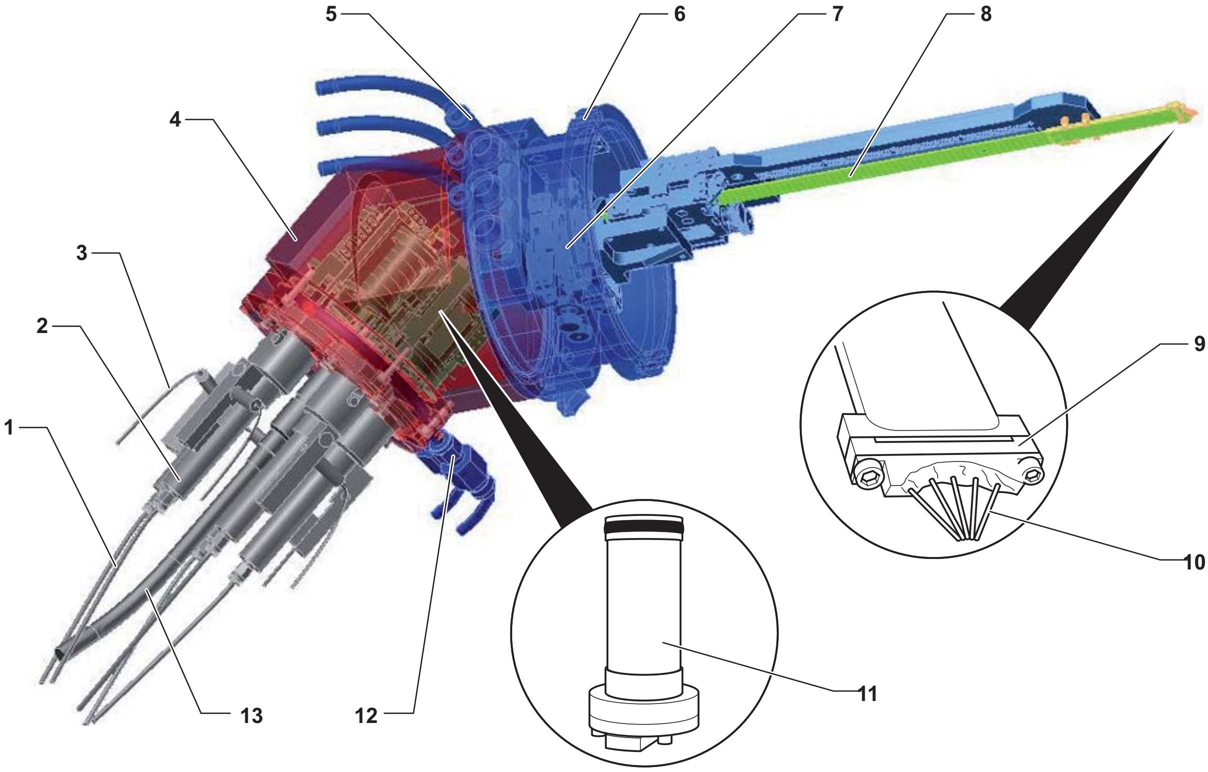


Figure 21-1: Schematic of the GIS hardware

Legende

1	pressure tubes for pneumatic actuators	**2**	pneumatic actuators for opening the reservoir valves
3	sensor cable	**4**	vacuum bell jar
5	MOT connectors for the control of the micropositioners	**6**	flange for adapting the GIS to the vacuum chamber
7	micropositioner for positioning of the nozzle block	**8**	capillaries for connecting the reservoirs with the nozzle block
9	nozzle block	**10**	nozzle for injection of gaseous precursors
11	reservoir, containing the precursor	**12**	water connector for PTML element
13	control cable for heating elements and thermometers		

1.3 Hardware overview of the GIS controller

The GIS controller has the following indicator and control elements on the front panel:

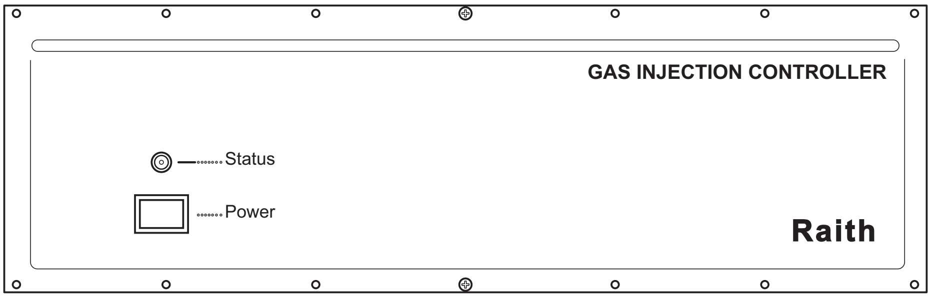


Figure 21-2: Front view of GIS controller

Indicator	Function
Status	green when the GIS is ready to use
Power	switches the GIS controller on and off. White when the controller is switched on

The GIS controller has the following connectors on the rear side:

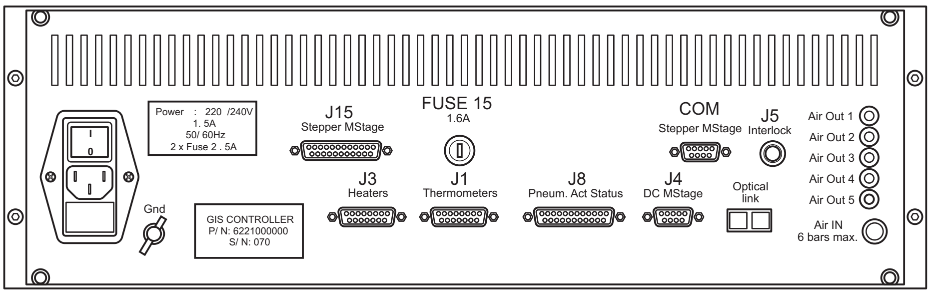


Figure 21-3: Rear view of GIS controller

Connector	Function
I/O	I/O port for switching of the 24V of the main board power supply
J15 Stepper MStage	25 Sub-D connector for control of the micropositioner (to MOT connectors on the GIS)
FUSE 15 1.6A	main fuse
COM Stepper MStage	RS232 connector for control of the micropositioner

Connector	Function
J5 Interlock	not used
J3 Heaters	Sub-D connector for heating elements on the GIS
J1 Thermometers	Sub-D connector for thermometer on the GIS
J8 Pneum. Act Status	Sub-D connectors for the pneumatic actuators on the GIS
J4 DC MStage	not used
Optical Link	optical connector for control of heaters and thermometers
Air Out 1-5	pressure connectors for pneumatic actuators
Air IN 6 bars max.	connector for 6 bar pressure supply

1.4 GIS software module

The control of the GIS can be performed in three user levels with different authorization and dialogs:

- USR-Level:

  A user with authorization for the USR-Level has a standard control of the GIS and can perform structuring processes with preset values.

- EXP-Level:

  In addition to the capabilities of the USR-level the EXP-level user can

  – define the set-temperature for the reservoir and capillary,

  – stepwise movement of the nozzle block in XYZ-axes.

  (⇒ *Workflows in the EXP-level* on page 21-40)

- SYS-Level:

  The SYS-level user has full control of the GIS option. In addition to the capabilities listed above, the SYS-level user has control over settings of parameters and process procedures.

  (⇒ *Workflows in SYS-level* on page 21-28)

Functions or dialogs which refer only to a specific user level are marked with USR, EXP or SYS.

(⇒ Figure 21-4 on page 21-7) and (⇒ Figure 21-5 on page 21-8) on the following pages show the user interfaces of USR-, EXP- and SYS-levels:

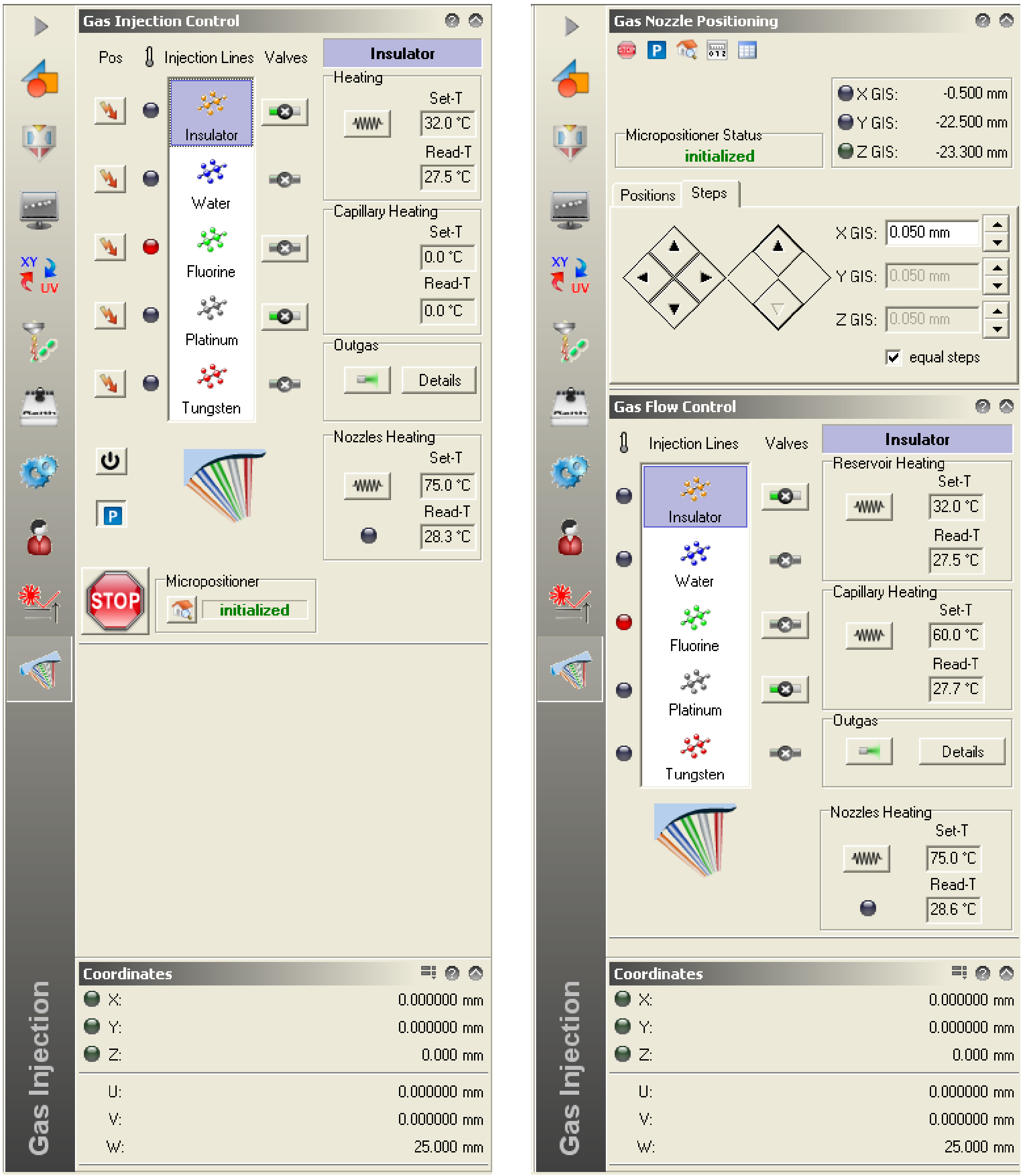


Figure 21-4: Software interface of the GIS module USR-level (left) and EXP-level (right)

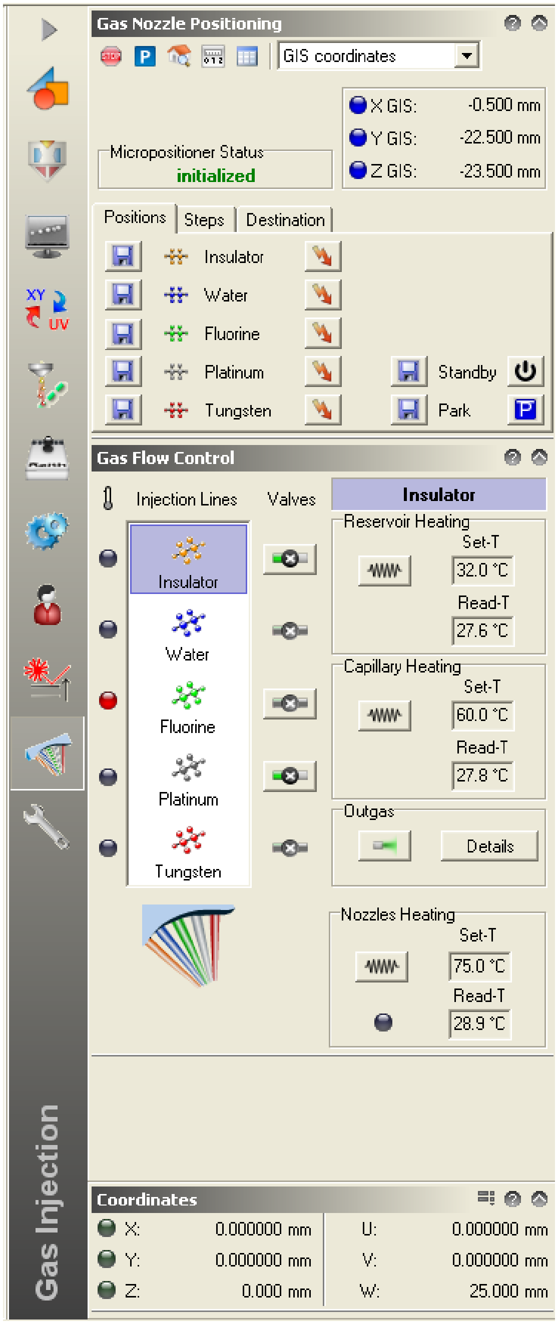


Figure 21-5: Software interface of the GIS module in SYS-level

1.5 Installation of the GIS

The GIS can be integrated with the following Raith lithography systems:

- eLINE electron beam lithography system,
- ionLINE ion beam lithography system,
- RAITH150 Two electron beam lithography system.

The functionality and the basic control of the GIS via a NANOSUITE application is the same in the three systems.

Differences are in:

- the beam type:

  The local deposition and etching processes for ion beams and electron beams are rather different. The interaction of each beam and the way that material is removed are dissimilar and these properties have to be considered in the selection of parameters.

- installation:

  In the eLINE and in the ionLINE, the mounting of the GIS is angular, to the rear of the chamber. The nozzle block moves from the left corner on the rear to the working position. In the RAITH150 Two the GIS is mounted on the rear plane of the chamber.

  The movement of the GIS arm, to reach a particular position, depends on the method of installation and therefore the visualization of the movement is different. The movement of the GIS has to be considered, while working with it, in order to avoid collisions of the nozzle block with other parts of the chamber.

- the load procedure:

  Depending on the system, the samples are loaded either via the front door, or via the load lock. The load procedure varies accordingly.

2 Basics of Patterning with precursors

The following sections will explain the patterning principles.

2.1 Principle of Patterning with electron beams

If a sample in an evacuated chamber is exposed to a focussed electron beam, the following types of particles are generated:

- secondary electrons,
- backscattered electrons,
- x-ray photons.

This principle is generally used for imaging, lithography, metrology and various analytical methods.

If a gaseous precursor is added to the system, gas molecules are cracked and adsorbed by the electron beam at the point of incidence. Two different processes take place which are mainly caused by secondary electrons:

- electron beam induced deposition (EBID):

  Precursor gases containing either metal or a semiconductor lead to local deposition on the sample surface allowing micro- or nano-scale patterning.

  By the application of suitable precursors micro- and nano-structures can be generated.

  (⇒ Figure 21-6 on page 21-10) shows the main principle of EBID:

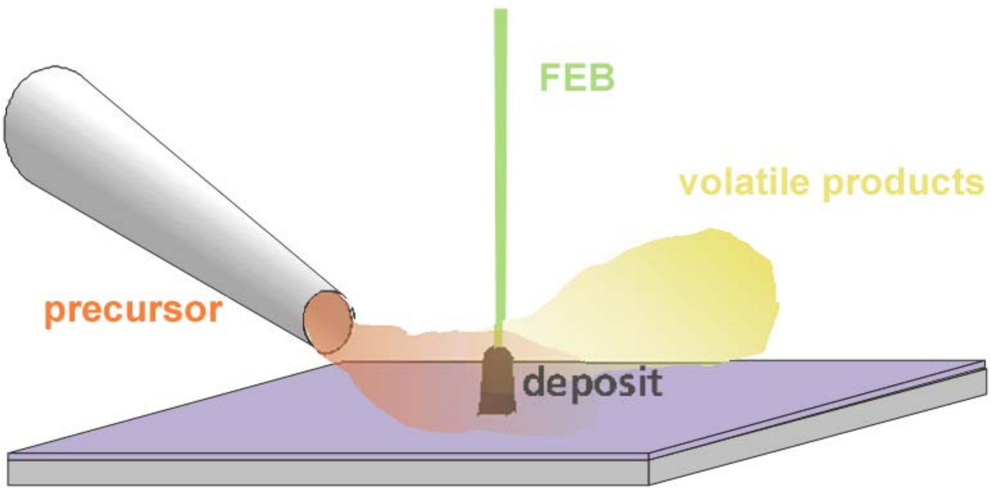


Figure 21-6: Principle of electron beam induced deposition (EBID)

- electron beam induced etching (EBIE):

By using reactive gases, such as fluorine, water or chlorine, selective etching processes of certain materials can be induced, due to local chemical reactions.

Process parameters can be adjusted to optimize the EBID or EBIE process (⇒ *Process rules* on page 21-14).

2.2 Principle of Patterning with gases and ion beams

The principle of patterning with ion beams and gases is similar to that of working with electron beams. A focussed ion beam cracks the gas molecules which leads to a deposition or etching structure on the sample.

The main difference is that the ion beam is better for etching than for deposition processes. The ions hitting the sample surface effect a mechanical treatment of the sample surface (sandblast principle), which an be enhanced in combination with etching precursors (reactants). For example, in combination with xenon fluoride the etching is more effective than in combination with an electron beam.

2.3 Functionality of the GIS

The GIS is equipped with five precursor gases, which can be used individually or in combination with each other.

2.3.1 Functionality of the injection lines

The precursor material inside a reservoir is either in a solid or in a liquid state. It has to be heated or cooled.

- to be injected in gaseous state,
- be injected with optimum pressure.

A reservoir is heated by its holder and the valve is opened by a pneumatic actuator. The gas streams through the capillaries to the nozzle. The nozzle injects the precursor to the sample surface.

An increasing temperature gradient from reservoir to nozzle block is necessary. In this way, condensation of the precursor on cooler surfaces of the injection line is avoided. The following components of the injection lines have heating elements and can be temperature controlled:

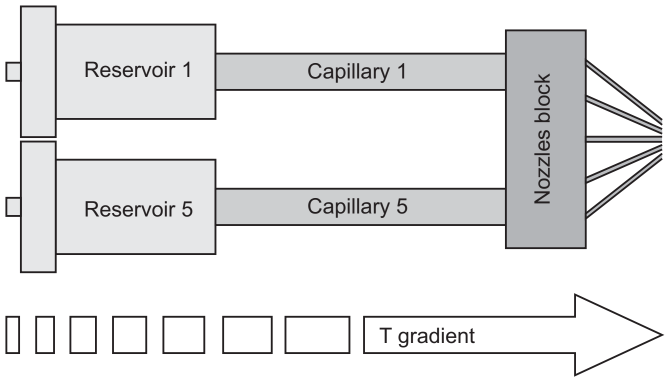


Figure 21-7: Temperature gradient of the injection lines

By heating the nozzle block, the nozzles of all injection lines are all maintained at the same temperature.

Precursors which are already at room temperature, such as xenon fluoride, are a special case. The reservoir is cooled by a PTML element, in order to keep the gas at the required pressure. The capillary is not heated, as the temperature gradient is already sufficient.

The GIS software module controls the injection lines and the micropositioner via two RS232 ports. This allows automated procedures for handling the GIS.

2.3.2 Samples

The GIS is basically suitable for structuring of all sample materials. It is recommended to use conducting materials such as silicon-based semi-conductor materials. On insulating materials such as glass, or synthetic materials, the results can be influenced by charging effects in the sample.

2.3.3 Types and application of precursors

The standard precursors consist of a mixture of different chemicals.

The standard configuration of the GIS is:

- PMCPS
- $(Me_3)MeCpPt$
- $W(CO)_6$
- $MgSO_4.7H_2O$
- XeF_2

The precursors have different properties to create specific structures on sample surfaces. They can be divided into two groups due to their principle of patterning:

- deposits for local deposition (EBID)
- reactants for local etching (EBIE)

In the GIS software module, the injection lines are named referring to their precursors. The naming scheme refers to the main component or the property. The names can be modified by the SYS-level user (⇒ *Renaming injection lines* on page 21-38).

The following tables give an overview of the application of the different precursors:

Local deposition (EBID)

Table 21-1: Application of the deposits

<table>
<tr><th>Deposit/
Injection Line</th><th>Precursor</th><th>Application</th></tr>
<tr><td>insulator</td><td>PMCPS</td><td>– electrical insulation
– micro- and nano-structures</td></tr>
<tr><td>platinum</td><td>$(Me_3)MeCpPt$</td><td rowspan="2">– electrical contacts
– micro- and nano-structures
– etch mask for the following processes</td></tr>
<tr><td>tungsten</td><td>$W(CO)_6$</td></tr>
</table>

Local etching (EBIE) and process optimization

Table 21-2: Application of the reactants

Reactant/ Injection Line	Precursor	Application
water	$MgSO_4.7H_2O$	– etching of coatings (e.g. PMMA) – etching of carbon – reduction of carbon content during deposition – increase of oxygen content for insulator
fluorine	XeF_2	– etching of silicon and silicon oxide – etching of tungsten

Depending on the application, several precursors can be used in combination. For example, the addition of water during insulator deposition improves the deposition (⇒ *Using multiple precursors together* on page 21-16).

The application of further precursors is evaluated. If you have a specific requirement, please consult Raith Support.

2.4 Process rules

The process rules in this chapter refer to the use of the GIS in the eLINE and RAITH150 Two. For questions about process rules in the ionLINE please contact Raith Support.

When patterning samples with precursors, a number of parameters are interdependent. These parameters have to be adjusted to optimize deposition or etching processes. Depending on the precursor, the following parameters play an important role:

- energy potential of the electron beam
- method of patterning
- the system vacuum during injection of the gas
- temperature of the reservoir, capillary and nozzle block
- sample material
- purity and dose of the injected gas

The following sections provide recommendations and suggestions for temperature, system vacuum and process strategies.

2.4.1 Temperature and system vacuum

The precursor needs specific vacuum conditions inside the chamber, related to its properties, to create structures. The reservoir temperature of an injection line influences the gas stream significantly and therefore also the system vacuum during the injection of the gas. If the pressure is too high or too low you can adjust it by increasing or decreasing the reservoir temperature.

The temperature values refer to a system base pressure of 1 to 3 x 10^{-6} mbar.

Table 21-3: Precursor-specific values for temperature and system vacuum

Injection line	Precursor	T (reservoir) [°C]	T (capillary) [°C]	T (nozzle block) [°C]	System-vacuum [mbar]
1	insulator (PMCPS)	35 to 50	55 to 65	75	4 to 8x10^{-6}
2	water ($(Me_3)MeCpPt$)	30 to 40	50 to 55	75	0.5 to 1.2 x 10^{-5}
3	fluorine ($W(CO)_6$)	12 to 15	room temperature	75	0.8 to 1.5 x 10^{-5}
4	platinum ($MgSO_4.7H_2O$)	50 to 65	60 to70	75	4 to 8 x 10^{-6}
5	tungsten (XeF_2)	60 to 70	65 to 75	75	4 to 8 x 10^{-6}

The adjustment of reservoir, capillary and nozzle temperatures is described in the section (⇒ *Setting temperatures for reservoirs, capillaries and nozzle* on page 21-36).

2.4.2 Special precursors

The following sections explain the applications of precursors.

Application of fluorine

Since fluorine is a gas at room temperature, the corresponding reservoir has to be cooled. During the application of fluorine, the other injection lines should not be heated, otherwise the fluorine reservoir will be heated at the same time and the cooling procedure will be made difficult.

Application of platinum

The platinum injection line is vented via a bypass, situated directly behind the reservoir valve. This allows the residual gas, which is inside the capillary and the nozzle after a structuring process, to be purged. This is necessary to avoid unwanted residuals on the sample surface.

To purge the residual gas in the capillary and nozzle, the bypass is automatically opened after closing the reservoir valve. As a result, the system pressure increases and decreases again quickly. This is a normal effect. The pressure increase takes place in the part of the capillary which is not connected to the chamber, the pressure impulse is neither harmful for the electron column nor for the sample.

During the outgas-procedure this effect can cause a higher pressure as during a single opening of the reservoir-valve (⇒ *Outgas procedure* on page 21-19).

Using multiple precursors together

Depending on the application, you can use two or more precursors at the same time. As a standard, the insulator and water can be combined to increase the deposition rate of the insulator. In this case you have to open the valves for both injection lines at the same time. As a result, the system vacuum degrades a little, but it should not exceed a value of 2×10^{-5} mbar.

To avoid exceeding this value you need to experiment with the reservoir temperature. Take into consideration which of the two parallel injected precursors is expected to achieve the greater degree of deposition or etching. The greater the etching or deposition, the higher the temperature has to be set.

Proceed as follows to find the optimum temperature for the reservoirs:

The adjustment of the set temperature is only available for users with EXP- and SYS-Level.

- Switch off the high voltage.
- Close the column chamber valve.
- Set the temperature for the combined precursors with respect to the suggested values in (⇒ Table 21-3 on page 21-15).
  (⇒ *Setting temperatures for reservoirs, capillaries and nozzle* on page 21-36, ⇒ *Adjusting the set temperatures for reservoirs and nozzle block* on page 21-40).
- Open both reservoir valves.

- Monitor the effect on of the system vacuum.
- Close the reservoir valves immediately, if the system pressure approaches 2 x 10^{-5} mbar.
- Adjust the temperature step by step until the system pressure with opened reservoir valves stays below 2 x 10^{-5} mbar.

If the system pressure does not stabilize below 2 x 10^{-5} mbar, please contact the Raith support.

- Note the set temperature values for use in future structuring processes with the corresponding precursors.
- Perform a patterning process as required.

2.4.3 Electron potential and process strategy

Depending on the sample material and the application, you need to test the parameters for the energy potential of the electron beam and the patterning. Consider the rules and hints in the following sections.

Single parameters need to be adjusted in the NANOSUITE application of the related system and then assigned to a design in the GDSII-Editor.

Use the basic documentation of the lithography system to see how to adjust the electron potential and patterning parameters.

Electron potential and deposition rate

For most of the precursors for EBID processes, the deposition rate is higher for a lower electron voltage (~1 to 3 kV). As a result, the lowest possible deposition rate is achieved if the electron beam has a comparable high acceleration voltage (~20 kV).

Largely, most of the applications on a sample surface or with nano-objects, such as carbon nanotubes, are performed at ~ 3 kV. In this regime, the brightness of the image and the stigmator settings change when the nozzle block is driven to its working position. Accordingly, a stigmation correction is required.

Patterning and deposition rate

The settling time parameter, which is actually used to compensate for dynamic effects of the electromagnetic deflectors, plays an important part in EBID processes. It serves as a refresh time for the precursor flux. Since the precursor is used up during the EBID-processes and the diffusion is too slow for some deposition processes, the adsorbed precursor concentrates close to the deposition. Therefore a fast deposition and several repetitions of the design (loops) cause a higher deposition rate.

Conductivity and processing

Experiments to optimize the conductivity of deposited tungsten and platinum films have shown that adjusting the settling time damages the process. It is recommended to use a large dwell time and no loops.

Process strategy

For typical structuring processes the following settings and strategies have been proven:

Table 21-4: Parameters for deposition (EBID)

Process	**Acceleration voltage [kV, pA]**	**Step [nm]**	**Dwell Time [ms]**	**Settling Time [ms]**	**Loops**	**Comment**
W, Pt, SiO_2 high deposition rate	3 kV 150 pA	10	0,01	1	1000	
W high conductivity	3 kV 150 pA	20	300	1	1	perpendicular meander mode w/o blanking
Pt high conductivity	3 kV 150 pA	20	50	1	1	perpendicular meander mode w/o blanking
SiO_2 improvement of C	3 kV 150 pA	10	0,01	1	1000	additional injection of water

Table 21-5: Parameters for etching (EBIE)

Process	**Acceleration voltage [kV, pA]**	**Step [nm]**	**Dwell-Time [ms]**	**Settling-Time [ms]**	**Loops**	**Comment**
thermical for SiO_2 for SiF-etching	3 kV 150 pA	10	0,005	1	10000	

B Tasks

1 Outgas procedure

The following tasks will explain the outgas procedure.

1.1 Aim and principle of the outgas procedure

The aim of an outgas procedure is to reduce residual gas pressure inside the reservoir, in order to inject the precursor at a specific pressure.

By heating the reservoir, or by using ambient temperature, the precursor changes state from liquid or solid to the gaseous state. If a heated reservoir remains closed for a period of time, an increased gas pressure can occur. When heating the reservoir again and opening the reservoir valve to start a patterning process, the gas is injected with a consequent high pressure. This can lead to voltage sparkovers at the beam blanker or at the filament and may therefore harm the electron column.

Always perform an outgas procedure after heating a reservoir, especially:
– before first patterning of the day
– if the reservoir was closed between two patterning processes for several hours.

The outgas procedure must be performed for each single heated injection line.

During an outgas procedure, the reservoir valve is cycled between open and closed several times, by the pneumatic actuator. This ensures that the residual gas is evacuated step by step.

If required, perform the outgas procedure several times with different settings until a significant pressure decrease in the system vacuum is observed and the precursor-specific pressure is reached.

When performing an outgas procedure with the platinum precursor, the system pressure can increase for a few seconds after closing the reservoir valve. The system vacuum will improve after a short time. This is a normal effect for this injection line, which is not harmful for the electron column. (⇒ *Application of platinum* on page 21-16).

1.2 Performing an outgas procedure

The outgas procedure is done in the same manner for all user levels, as described in the following steps.

Proceed as follows:

STEP 1: Preparation

▶ Select the dialog **Gas Injection Control** (USR) or **Gas Flow Control** (EXP/SYS):

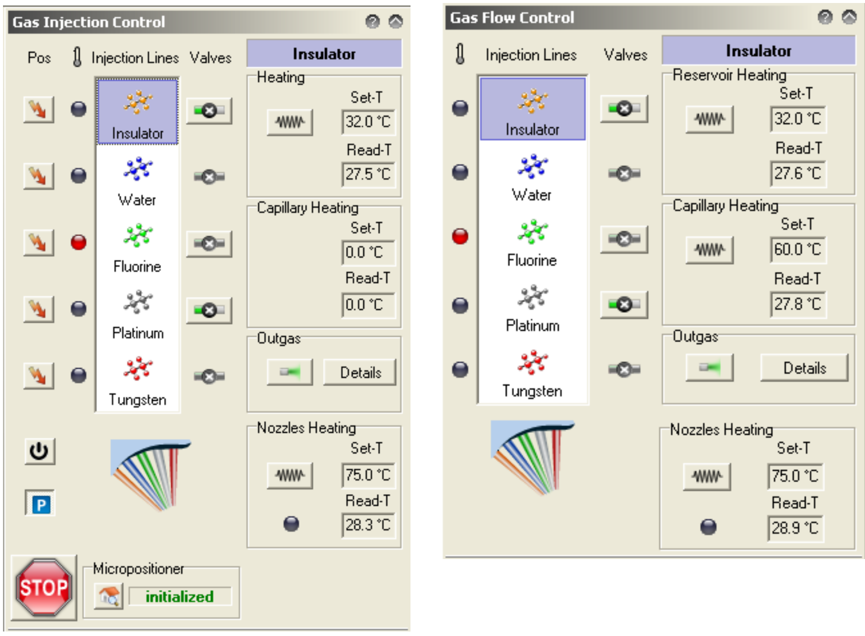


Figure 21-8: dialog **Gas Injection Control** (USR) and **Gas Flow Control** (EXP/SYS)

▶ Make sure that

- the set temperature for reservoirs, nozzle and capillaries is reached. If so, the status LEDs for the injection line and for the nozzle block are green.
- the system vacuum has a base pressure of 1 to 3 x 10^{-6} mbar.

▶ Drive the nozzle block to **Park** position.

▶ Switch off the high voltage.

▶ Close the column chamber valve.

STEP 2: Outgas procedure

▶ Choose the injection line for which you want to perform an outgas procedure by clicking on the button.

▶ Open the subdialog **Properties** (**Outgas**) by clicking on **Details** in the section **Outgas**:

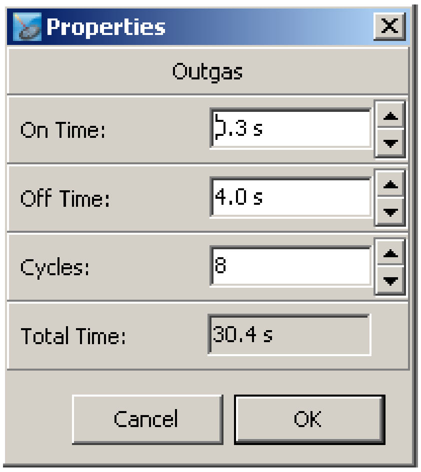


Figure 21-9: Subdialog **Properties** (**Outgas**)

The values in the subdialog **Properties** (**Outgas**) have standard presettings. Depending on the development of the system vacuum after the first outgas attempt, these values can be changed. Do not go below the standard value for the valve open time of (**On Time**) 0.3 s.

- ▶ Type in a value for the valve open time in the field **On Time**. Use the standard value of 0.3 s for the first outgas attempt.
- ▶ Click **OK**.
- ▶ Click , in the dialog **Gas Injection Control** (USR) or **Gas Flow Control** (SYS) to start the outgas procedure.

During the outgas procedure, monitor the development of the system vacuum in the field **vacuum**: It should stay in the region of 10^{-6} mbar or increase to between 1x 10^{-5} and 1,5 x 10^{-5} mbar.

You can stop the outgas procedure at any time by clicking  again and restarting the outgas procedure afterwards. The outgas procedure is then performed with the last set values.

- ▶ When there is no further pressure increase at the end of an outgas procedure, you can increase the valve open time. Repeat the outgas procedure with valve open time of, for example, 0.4, 0.5 or 0.7 seconds. Keep the system vacuum in the range of 10^{-6} mbar.
- ▶ Open the reservoir valve by clicking  and check the development of the system vacuum. It should stay below 1,5 x 10^{-5} mbar and finally reach the precursor-specific pressure (⇒ *Temperature and system vacuum* on page 21-15).
- ▶ Close the reservoir valve by clicking  when the precursor-specific pressure for the chosen injection line is reached.

2 Correcting stigmation

When the nozzle block is driven to the working position, the stigmation will change. Due to shadowing effects under the nozzle block the focus of the image can change.

Please see in the basic documentation how to adjust stigmation.

2.1 Loading the sample holder

NOTICE

Damage to the system may occur by inappropriate loading of the sample holders. Always use the **Load** and **Unload** procedures from the Exchange Navigator or the load lock module.

NOTICE

Danger of damaging the nozzle block due to collisions. Do not use:

– samples or sample holders with surface roughness of more than 50 µm,
– samples which are tilted on the sample holder.

Depending on the system, the sample holder is loaded via the front door or via the load lock. In either case, the automatic load and unload procedures of the navigator are used. The load and unload procedures have three GIS-specific steps added, to ensure safe loading of sample holders:

- In a load procedure, the nozzle block is automatically driven to its park position.
- All heating elements of the injection lines are automatically switched off during an unload procedure. In a load procedure, all heating elements of the nozzle block and the capillaries are automatically switched on.
- All valves are unlocked during a load procedure and locked during an unload procedure.

Please see in the basic documentation how to load via the navigator.

2.2 Initializing the micropositioner (find home procedure)

The initialized status of the micropositioner (home position) is preset as standard. The home position is the reference for the working positions of the nozzle block, the standby position and the parking position. During the use of the GIS it might be necessary to initialise the micropositioner again. The initialization is performed automatically via the find home procedure.

Perform a find home procedure

- regularly, every two or three weeks to ensure the positioning accuracy of the micropositioner,
- when the GIS controller has been switched off.

When a find home procedure is necessary a status message appears in the dialog **Gas Injection Control** (USR)/**Gas Nozzle Positioning** (EXP/SYS):

Figure 21-10: Status: micropositioner not initialized

Proceed as follows to initialize the micropositioner:

▶ Click the  button for the find home procedure.

The laser stage is driven to Z = 1 mm and the micropositioner is automatically initialized. This procedure may take a few minutes. When the find home procedure is successful the following message appears:

Figure 21-11: Confirm the initialization of the micropositioner

▶ Confirm with **OK**.

The micropositioner is initialized and the status changes to:

Figure 21-12: Status: micropositioner initialized

The system asks to which stage position the laser stage needs to drive in z:

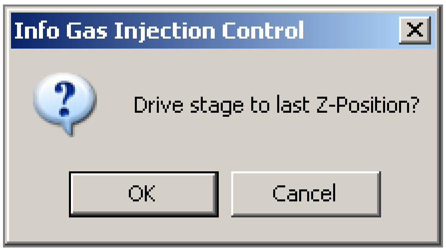


Figure 21-13: z-position of the laser stage

▶ Click **OK** when the laser stage has to drive to the last z-position.
Click **Cancel** if the laser stage is supposed to stay at the current position.

It is possible that the find home procedure is not successful. In this case proceed the following way.

▶ Close the NANOSUITE Application.

▶ Switch off the GIS controller and restart it.

▶ Open the NANOSUITE Application.

▶ Select the GIS module.

▶ Repeat the find home procedure by clicking .

If the find home procedure is not successful after the second attempt contact the Raith support team.

---

3 Preparing and starting a patterning process

The procedure for an EBID or EBIE process is almost the same in USR-, EXP- and SYS-levels. You will find the necessary commands and functions in the related dialogs. The following section shows how to perform an EBID or EBIE procedure in the different user levels.

STEP 1: Load sample holder

Load a sample holder. Proceed as described in section (⇒ *Loading the sample holder* on page 21-22).

STEP 2: Prepare the GIS

▶ Make sure that

- the lithography system is switched on and ready to use,
- PC and monitors are switched on,

- the NANOSUITE Application has been started,
- the chamber is evacuated.

▶ Select the GIS module. Depending on the user level the following dialogs will open:

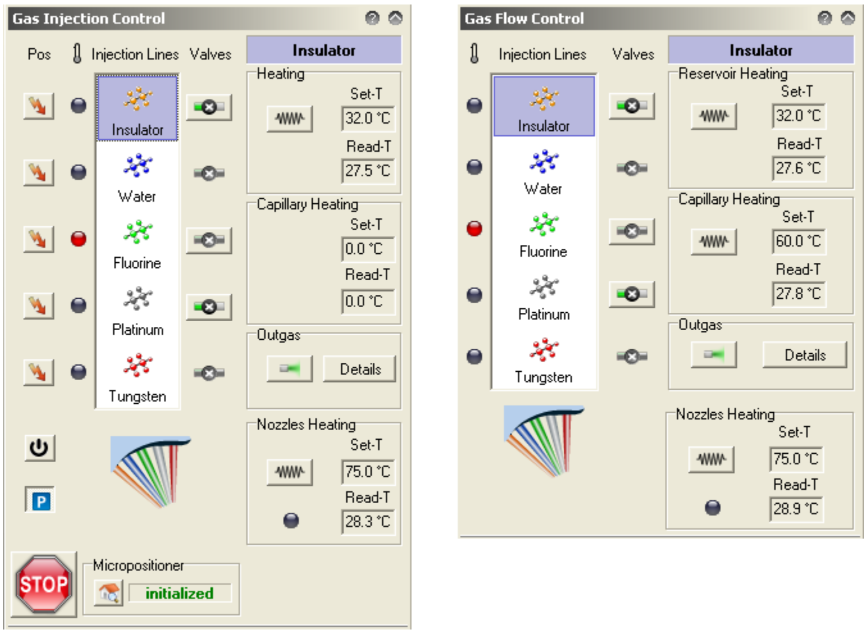


Figure 21-14: **Gas Injection Control** dialogs (USR) and **Gas Flow Control** (EXP/SYS)

After starting the lithography system, the nozzle block is parked and all heating elements are switched off.

▶ Monitor the development of the system vacuum.

When the system has reached a pressure in the range of 10^{-6} mbar you can switch on the heating element of the reservoir:

▶ Select a precursor in the column **Injection Lines**.

You can combine several precursors such as insulator and water. Which precursors can be combined is indicated by the status of the buttons in column **Valves**: If two or more buttons show the state , you can inject them at the same time.
As a SYS-level user you can define a different combination of precursors in dialog **Gas Nozzle Positioning** (⇒ *Using multiple precursors together* on page 21-16).

▶ Click in the section **Reservoir** to switch on the reservoir heating.

In USR-level the heating elements for capillaries and nozzle block are switched on automatically.
In EXP- and SYS-levels these elements are switched on manually by clicking [button] in the corresponding section.

The status LEDs for the reservoirs, capillaries and the nozzle block change to yellow.

The current temperature development is shown in the field **Read-T**. When the set temperature is reached the status LEDs switch to green.

- Wait for 10 min until a thermal equilibrium between the components and the injection lines is reached before continuing the patterning process.

STEP 3: Setting processing parameters and energy potential

- Set the parameters for patterning and energy potential in the GDSII Editor and assign them to the corresponding design.
Orientate on section (⇒ *Electron potential and process strategy* on page 21-17).

Please see in the basic documentation how to work with the GDSII Editor.

STEP 4: Set a working distance of 10 mm

NOTICE
Danger of collision for nozzle block and sample.
Always keep a working distance of 10mm between sample surface and column.

- Switch on the acceleration voltage.
- Switch on the imaging software.
- Focus on the sample close to the position for the patterning process.
- Set a working distance of 10 mm by moving the laser stage to a proper z-position. Check this by precise focussing on the sample surface.
- Adjust the uv coordinate system on the sample.

STEP 5: Perform outgas procedure

NOTICE

Danger of voltage sparkovers at beam blanker and filament due to high pressure impulse. Always perform an outgas procedure before opening a reservoir valve if:

– a reservoir was heated for the first time that day,
– a reservoir valve was closed for several hours after heating.

▶ Do the outgas procedure as described in the section (⇒ *Outgas procedure* on page 21-19).

STEP 6: Preparing a patterning process

▶ Wait until the system vacuum has reached a base pressure of between 1 to 3 x 10^{-6} mbar.

▶ Switch on the acceleration voltage.

▶ Make sure a working distance of 10 mm is set.

▶ Drive the nozzle block to the working position of the required injection line by pressing .

▶ Adjust contrast and brightness of the image.

▶ Check focus and stigmation and adjust, if needed.

▶ Perform a writefield alignment if necessary.

▶ Prepare a positionlist depending on the patterning process.

Please see in the basic documentation how to adjust the settings.

STEP 7: Performing a patterning process

▶ Open the reservoir valve of the selected injection line by clicking .

▶ Monitor the development of the system vacuum. Wait until it has reached a stable value suggested in (⇒ Table 21-3 on page 21-15).

▶ Start the patterning process by scanning the positionlist. Proceed as guided by standard lithography processes.

▶ When the scanning of the positionlist is finished, close the reservoir valve by clicking .

The patterning process is finished and the reservoir valve is closed. The valve button changes to .

4 Workflows in SYS-level

In the SYS-level you can change settings and configurations for certain parameters and procedures.

4.1 Defining the target positions of the nozzle block

The target positions of the nozzle block are set for every injection line. During use of the GIS it might be required to adjust these values again.

The nozzle block can have the following target positions:

- working positions of the single injection lines,
- parking position which you address if
  - the GIS is not used,
  - the chamber is opened,
  - the laser stage has to be driven over longer distances, such as to the Faraday Cup,
  - the system is switched off.

The nozzle block assembly is first moved to the standby position before moving to the park position to prevent possible collisions with the column.

- standby position as a waiting position during interruptions of the workflow, such as between patterning processes.

You always have to adjust one or more target positions if:

- the nozzle block has been exchanged. In this case you have to adjust all target positions.
- you have seen that the working positions of one or more injection lines are no longer exact.

The target positions of the nozzle block are addressed via , , in dialog **Gas Injection Control** (USR)/**Gas Nozzle Positioning** (EXP/SYS).

The nozzle block is driven to its target position by the micropositioner in x-, y-, z-coordinates.

The target position of the nozzle block is set in dialog **Gas Nozzle Positioning** (SYS):

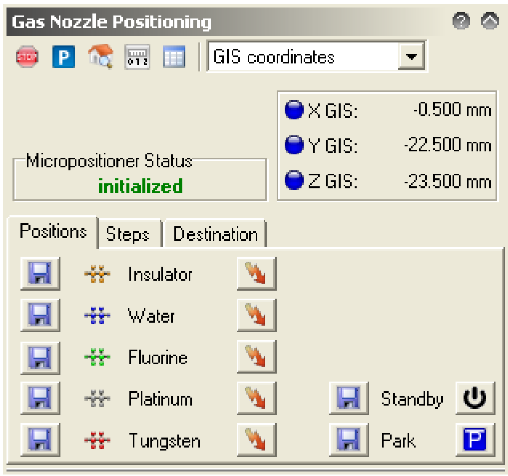


Figure 21-15: dialog **Gas Nozzle Positioning** – Tab **Positions**

There are two possibilities to adjust the values for the target positions of the nozzle block:

- In tab **Steps** by moving the nozzle block stepwise (⇒ *Steps tab* on page 21-55),
- in tab **Destination** addressing absolute or relative coordinates (⇒ *Destination tab* on page 21-56).

The procedure of adjustment of the target positions is the same for both tabs. The following section describes how to adjust the target position via tab **Steps**:

STEP 1: Preparations

▶ Ensure that the laser stage is at Z = 1 mm.

▶ Heat all reservoirs, capillaries and the nozzle block to the set temperature:

- Switch to dialog **Gas Flow Control**.
- Select the injection line in the column **Injections Lines**.
- Switch on the heating elements for reservoirs and capillaries by clicking in sections **Reservoir Heating** and **Capillary Heating**.
- Switch on the heating element for the nozzle block by clicking in section **Nozzles Heating**.

  When the set temperatures are reached the LEDs will turn to green.

▶ Wait for approximately 10 min until a thermal equilibrium between the different components is reached.

Always switch on all the heating elements before setting the target positions.
If you adjust the position without heating it will not fit for later process conditions due to thermal drift.

- Call up the image in the Electron Optics Software and activate the cross-hairs.
- Switch to dialog **Gas Nozzle Positioning**.
- Select **GIS coordinates** in the drop-down list.

STEP 2: Calibrating the nozzle block

- Switch to tab **Positions**.
- Drive to the preset target position by clicking .
- Switch to tab **Steps**:

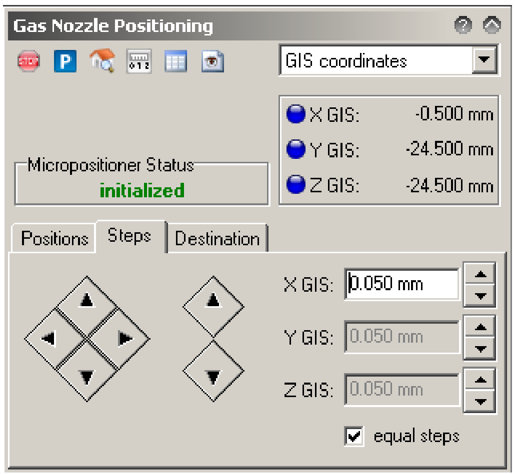


Figure 21-16: dialog **Gas Nozzle Positioning** – Tab **Steps**

- Define the stepsize that you want to use for positioning the nozzle block with the arrow keys in x-, y-, z-direction.
- Activate the checkbox **equal steps** if the same stepsize for x-, y-, z-axis has to be used.
- Select the tool for measuring circle dimensions in the Electron Optics Software and create a circle with a diameter of 600-700 µm with the same center as the image center (⇒ Figure 21-17 on page 21-31).

The lateral distance between nozzle and beam center has to be 300 to 400 µm. The lateral distance is equal to the circle radius. Take care that the circle tool always is displaying the diameter.

- Drive the nozzle by using the arrow keys in x- and y-direction close to the circle at low magnification.
- Focus on the front edge of the nozzle.

Make sure that the focus of the nozzle edge is at Z = 9.4-9.5 mm.

- Use the z-arrow key to get a focus of 9.4-9.5 mm and focus again.
- Move the nozzle block with the arrow keys in x- and y-direction until the center of the nozzle front edge is on the circle tool edge.

  (⇒ Figure 21-17 on page 21-31) shows the correct position using the example of the center nozzle:

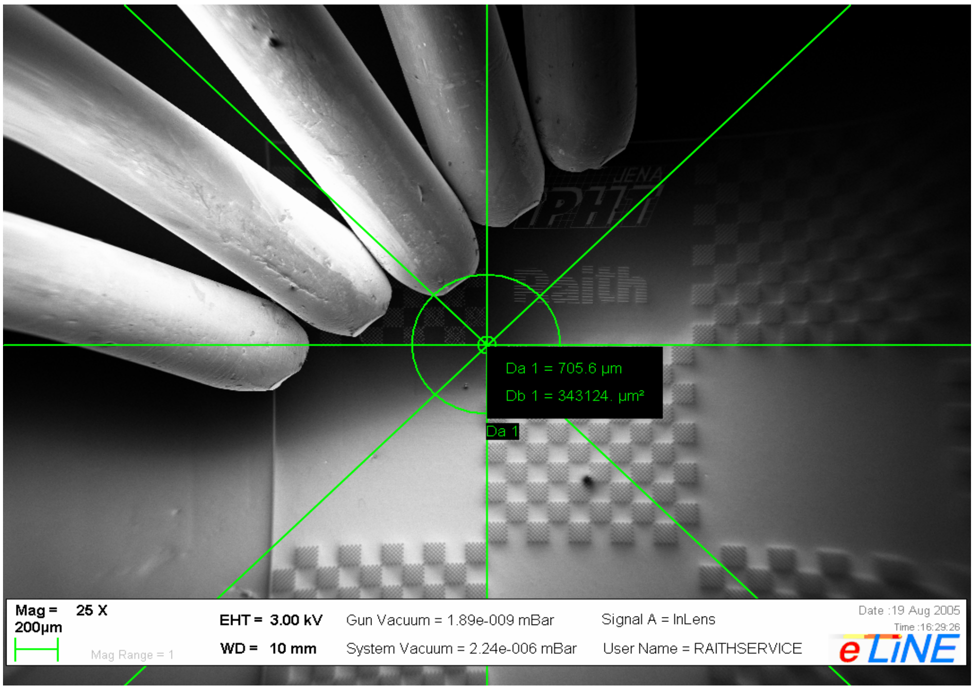
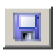


Figure 21-17: Correct position of the nozzle block using the example of the center nozzle. The lateral distance between nozzle and beam center is 352 µm which is half of the **Da 1**-value.

STEP 3: Saving the target position

- Switch to tab **Positions**.
- Save the adjusted target position for the selected injection line by clicking .

  The target position for the selected injection line is set and can be called up by clicking  in the dialog in USR-, EXP- or SYS-level.

4.2 Calibrating the NE coordinate system

Besides the coordinate system of the laser stage there are three coordinate systems available for working with the GIS. The coordinate system can be selected in the pull-down menu **Gas Nozzle Positioning**:

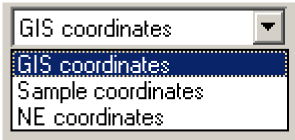


Figure 21-18: Pull-down menu coordinate systems

The menu contains the following coordinate systems:

- **GIS coordinates** – the physical GIS coordinate system in which the micropositioner moves in XYZ-coordinates.
- **Sample coordinates** – the sample coordinate system which is adjusted at the sample edge. You can define a specific coordinate system for every sample in U, V and W coordinates.

See in the basic documentation how to adjust a sample coordinate system.

- **NE coordinates** – the virtual coordinate system which links the GIS and sample coordinate systems.

  The NE coordinate system

  – is the basis for the visualization with which you can follow the movements of the nozzle block in a scheme (⇒ *Coordinates status window* on page 21-71). To work reliably with the visualization the NE coordinate system has to be adjusted correctly.

  – allows you to navigate the nozzle block without knowing the real positions in the different coordinate systems.
  If the NE coordinates are e.g. **X NE**, **Y NE**, **Z NE** = 0, the nozzle of a selected injection line is in the center of the electron beam and therefore in the center of the image and the visualization – and at the preadjusted working distance of 10 mm.

The NE coordinate system is adjusted by Raith during installation. During use of the GIS it might be necessary to adjust it again, for example:

- after a nozzle block exchange.
- if you see that the image and the visualization do not fit together.

STEP 1: Preparation of the adjustment procedure

Make sure that

- the micropositioner is initialized,

- the chamber is evacuated. During the adjustment procedure the electron beam is needed.

STEP 2: Starting the adjustment procedure

▶ Call up the dialog **Gas Nozzle Positioning**.

▶ Click on  to open the dialog **Adjust GIS coordinates**:

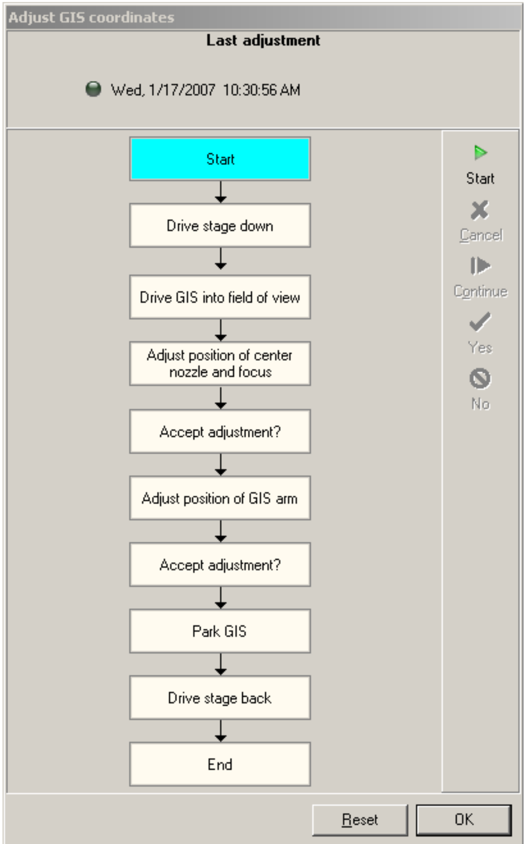


Figure 21-19: Adjustment procedure – Start screen

This dialog guides you step by step through the adjustment procedure by highlighting the according field in the flow chart. You can control the procedure with the following buttons:

Button	Function
Start	starts the adjustment procedure
Cancel	stops the adjustment procedure
Continue	continues the adjustment procedure after an interruption
Yes	confirms the message

Button	Function
No	negates the message and closes the dialog

Under **Last adjustment** is shown when the NE coordinate system has been successfully adjusted the last time. The status LED is green when the adjustment procedure has been finished successfully.

▶ Click **Start** to start the adjustment procedure.

The system does the following steps automatically:

- the laser stage is driven to Z=1 mm.

During driving of the laser stage the status message **Driving to Destination** is shown. You can stop the laser stage at any time by pressing **Stop** below the status message.

- The nozzle block is driven into the field of view.

STEP 3: Positioning the nozzle and focussing

The system asks you to adjust the center nozzle and the focus in step **Adjust position of center nozzle and focus**.

The positioning is done in the tabs **Steps** or **Destination** in the dialog **Gas Nozzle Positioning**. Check the settings in the image.

▶ Switch to tab **Steps** e.g.

▶ Move the nozzle block into the image center with the arrow keys at low magnification.

▶ Switch on the crosshairs in the image.

▶ Focus exactly on the center nozzle.

▶ Move the nozzle block with the arrow keys to adjust the center of the nozzle‘s front edge in the center of the crosshairs (⇒ Figure 21-20 on page 21-35).

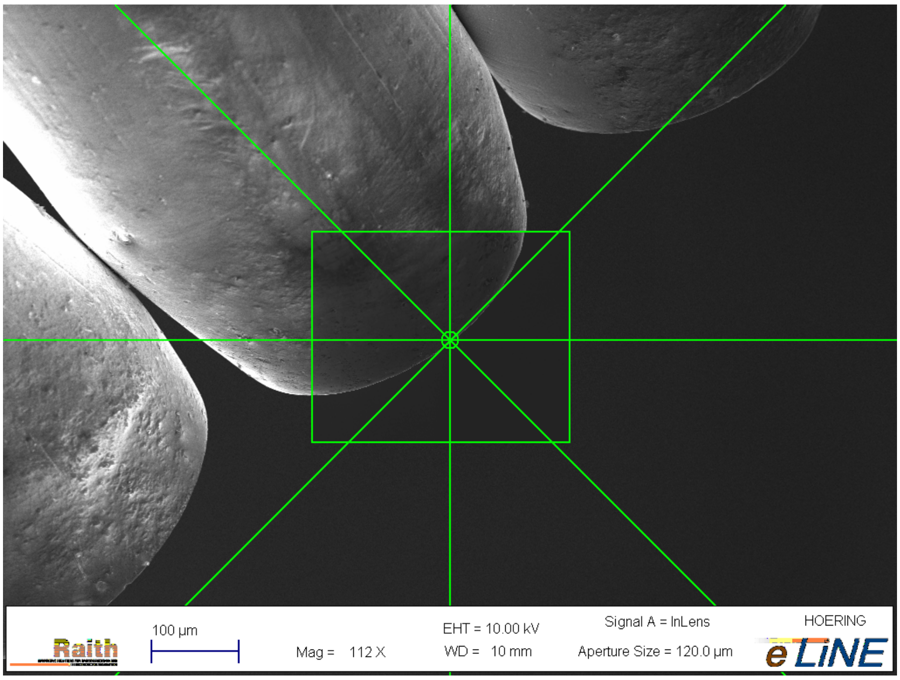


Figure 21-20: Focus on the center nozzle

Place the center of the nozzle's front edge to within 20 to 50um of the center of the crosshairs.

- Click **Continue**.

The dialog **GIS: Offset Correction** opens:

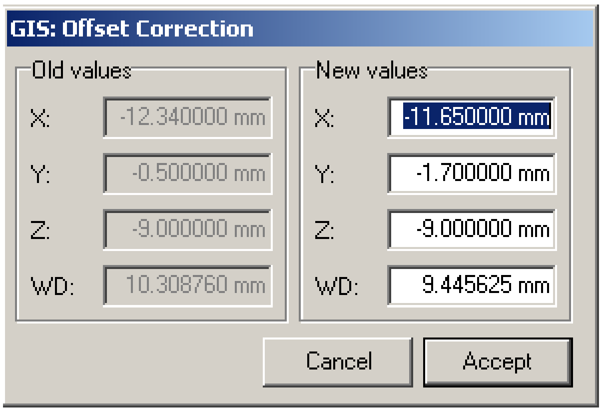


Figure 21-21: **GIS: Offset Correction** dialog

- Check the new offset values. Make sure that the **WD** value is 10 mm +/- 50 µm.

The **WD** value is the working distance which is the basis for adjusting the NE coordinate system.
If it differs significantly from 10 mm the offset values will not be reliable and the adjustment procedure has to be repeated. Make sure that during focussing a working distance of 10 mm is reached (**WD** value in ⇒ Figure 21-20 on page 21-35).

- Click **Accept** to accept the current settings.
Click **Cancel** to stop the adjustment procedure and restart it if necessary.
- Dialog **Adjust GIS coordinates** shows, in **Last adjustment,** date and time of the current adjustment. The status LED changes to green.

The adjustment procedure is continued by the step **Park GIS** and the nozzle block is parked.

In step **Drive stage back** the laser stage is driven to its last working position.

While the laser stage is driven the status message **Driving to Destination** appears. You can stop the laser stage any time by pressing **Stop** below the status message.

STEP 4: Closing the adjustment procedure

When the adjustment procedure is finished, the field **End** is highlighted.

- Click **OK** to close the adjustment procedure.

4.3 Setting temperatures for reservoirs, capillaries and nozzle

The temperatures for reservoirs, capillaries and nozzle block significantly influence the pressure of the system vacuum during injection. The recommended temperature values for the standard precursors are preset. Depending on the application and on alternative precursors it might be sensible to set new temperature values.

Always make sure that there is an increasing temperature gradient from reservoir to nozzle block. Orientate as recommended in section (⇒ *Temperature and system vacuum* on page 21-15).

Set the set temperatures as follows:

- Click in dialog **Gas Nozzle Positioning** on .

The dialog **Properties** opens:

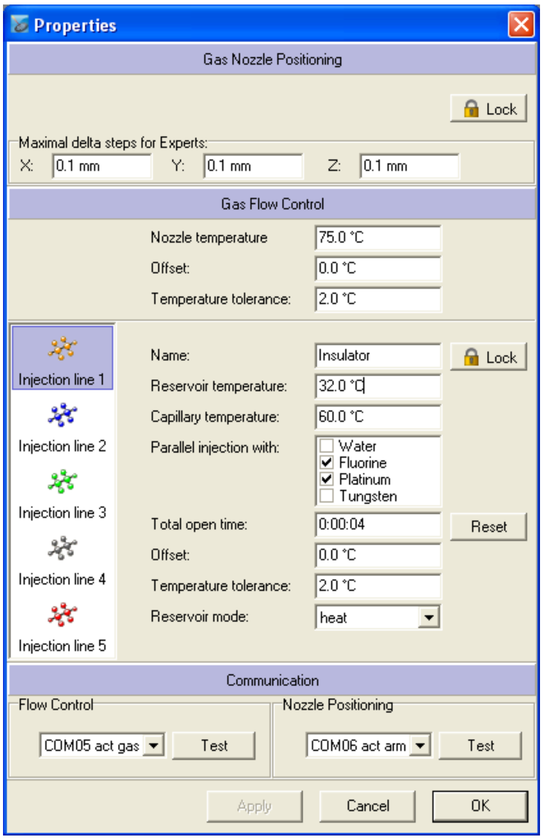


Figure 21-22: Dialog **Properties** (SYS)

- In the field **Nozzle temperature**, the set temperature for the nozzle block, in °C, has to be typed in.
- In the field **Temperature tolerance**, set a tolerance value for the behavior of the status LED in the dialog **Gas Injection Control** (USR)/**Gas Flow Control** (EXP/SYS) **Nozzles Heating.**

  The tolerance value defines the temperature range over which the status LED changes to green. In the example of (⇒ Figure 21-22 on page 21-37), the status LED in the **Nozzles Heating** section is green between 73 °C and 77 °C.

- Select the injection line for which you want to define the temperatures for the reservoir and capillary.
- Type in the set temperatures, in °C, in the fields **Reservoir temperature** and **Capillary temperature**.
- In the field **Temperature tolerance**, define a tolerance value for the behavior of the status LEDs in dialog **Gas Injection Control** (USR)/**Gas Flow Control** (EXP/SYS)➡column .
- Click **Apply** to confirm the changes.
- Click **OK** to close the window.

Always click **Apply** first, to confirm all changes.
By clicking **OK** the window is closed. Only the last change is then confirmed. All the other changes will be reset to previous values.

The set temperatures are preset and will be shown in field **Set T** in the following windows:

- **Gas Injection Control** (USR) section **Reservoir Heating**
- **Gas Flow Control** (EXP/SYS) section **Nozzles Heating**

4.4 Renaming injection lines

By default, the five injection lines are named by the main component of the precursor:

Table 21-6: Standard names of the injection lines

Injection line	Name	Precursor
1	insulator	PMCPS
2	water	$MgSO_4 \cdot 7H_2O$
3	fluorine	XeF_2
4	platinum	$(Me_3)MeCpPt$
5	tungsten	$W(CO)_6$

The names can be changed in the following way:

▶ In dialog **Gas Nozzle Positioning** click on .

The dialog **Properties** opens:

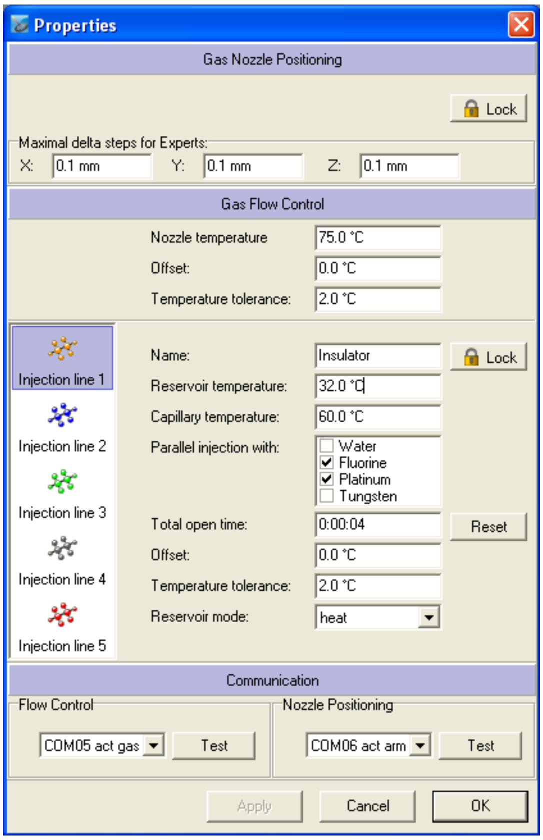


Figure 21-23: **Properties** dialog (SYS)

- Select the injection line by clicking on the button in order to rename it.
- Type in the new name in the **Name** field.
- Type in new names for further injection lines, if needed, and confirm all changes by clicking **Apply**.
- Click **Apply** to confirm the changes.
- Click **OK** to close the window.

Always click **Apply** first to confirm all changes.
By clicking **OK** the window is closed. Only the last change is then confirmed. All the other changes will be reset to previous values.

The injection lines are renamed and will be shown in the following windows:

- **Gas Injection Control** (USR) column **Injection Line**
- **Gas Flow Control** (EXP/SYS) column **Injection Line**

5 Workflows in the EXP-level

In the EXP-level, you have the same rights for using the GIS as in the USR-level, with the following additional capabilities:

- adjusting the set temperatures for reservoirs and capillaries of the single injection lines
- moving the nozzle block in step mode.

5.1 Adjusting the set temperatures for reservoirs and nozzle block

The temperatures for reservoirs, capillaries and nozzle block determine significantly the pressure of the system vacuum during injection. The recommended temperature values for the standard precursors are preset.

As a user in the EXP-level, you can adjust the set temperature for reservoirs and capillaries according to your application. The set temperatures influence the heating in window **Gas Flow Control** (EXP) and are shown in the **Set-T** fields.

Adjust the set temperatures as follows:

▶ In the **Gas Nozzle Positioning** dialog, click on .

The dialog **Properties** opens:

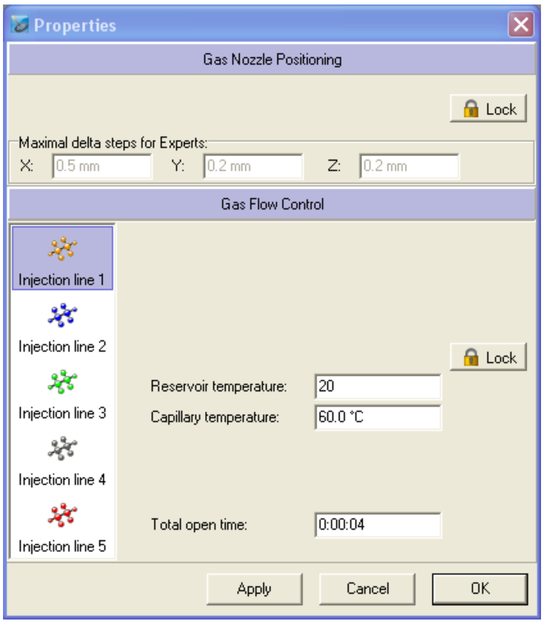


Figure 21-24: **Properties** dialog (EXP)

▶ Select the injection line in the left column.

▶ Type in the set temperature in °C in the fields **Reservoir temperature** and **Capillary temperature**.

The temperature range for changing the set temperatures for reservoirs and capillaries is limited. Orientate the set temperatures as recommended in (⇒ *Temperature and system vacuum* on page 21-15) and are set by SYS-level users. Theses values can be changed in the range of +/- 10°C.

If the new values are outside this range, the following message appears:

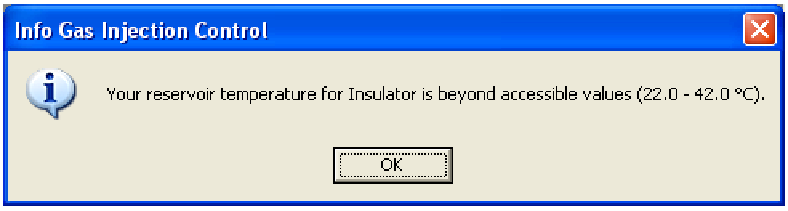


Figure 21-25: Error – temperature out of range (example)

- Click **OK** and correct the temperature value.
- Click **OK** to confirm the values and to close the window.

The set temperatures for reservoirs and capillaries are set and are available in dialog **Gas Flow Control**.

5.2 Driving the nozzle block to the target position

Drive to the target position as follows:

- Select dialog **Gas Nozzle Positioning** tab **Positions**:

Figure 21-26: **Gas Nozzle Positions** dialog (EXP) – **Positions** tab

- Drive the nozzle block to the target position by clicking on the following buttons:
  -  – preset working position for each injection line
  -  – standby position

-  – park position

The nozzle block is driven to the target position.

5.3 Driving the nozzle block in step mode

As an EXP-level user you can approach the nozzle block stepwise towards the sample surface. This can be sensible in certain applications but it affords a certain experience with the GIS.

The maximum possible stepsize is preset. These values for XYZ-axes are shown in:

Gas Nozzle Positioning ➡  ➡ dialog **Properties** ➡ section **Maximal delta steps for experts**.

Proceed as follows to approach the nozzle block stepwise:

▶ Select **Gas Nozzle Positioning** dialog - **Steps** tab:

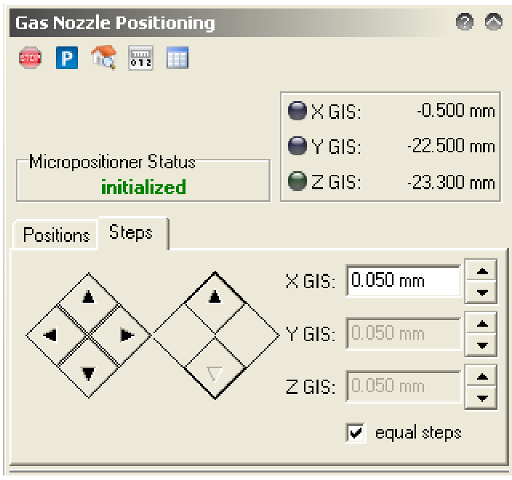


Figure 21-27: **Gas Nozzle Positioning** dialog (EXP) – **Steps** tab

▶ Type in the stepsize which shall be used to move the nozzle block in XYZ-axes.

If you exceed the maximum possible stepsize, the arrow keys will be deactivated and the nozzle block cannot be moved.

▶ Activate **equal steps** if the same stepsize for all axes should be used.

▶ Approach the nozzle block carefully to the sample surface by using the arrow keys and check the result in the image.

▶ Repeat this procedure, if necessary, with smaller step sizes until the nozzle block is in the correct position.

The approaching process is complete and you can start a patterning process (⇒ *Preparing and starting a patterning process* on page 21-24).

C Software Reference

This chapter contains an overview of all functions and control elements of the GIS module dialogs and subdialogs.

1 Gas Injection Control dialog (USR)

With the help of the dialog **Gas Injection Control**, the user may control the GIS at the USR-level on the basis of predefined values. The user can:

- perform full patterning (⇒ *Preparing and starting a patterning process* on page 21-24),
- perform an outgas procedure (⇒ *Outgas procedure* on page 21-19),
- initialize the micropositioner (⇒ *Initializing the micropositioner (find home procedure)* on page 21-22).

The window is called up by selecting .

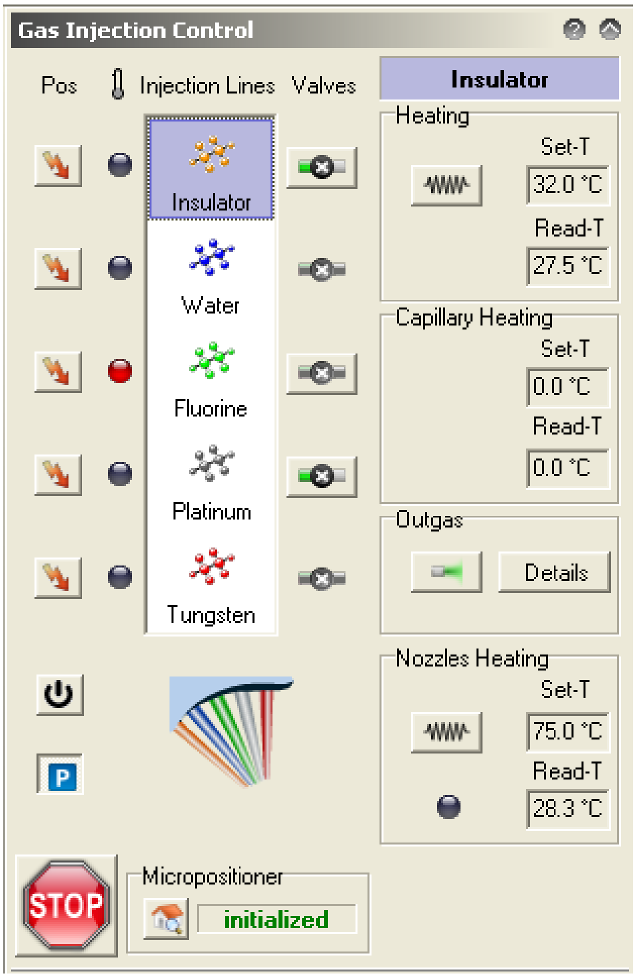


Figure 21-28: **Gas Injection Control** dialog (USR)

The window is divided into the following sections:

Section	**Function**
for control of the injection line	control of the injection lines (⇒ *Section for control of injection lines* on page 46).
Heating and **Capillary Heating**	heating of reservoirs and capillaries (⇒ *Heating/capillary heating section* on page 47).
Outgas	to perform outgas procedure (⇒ *Outgas section* on page 48).
Nozzles Heating	control of nozzle temperature (⇒ *Nozzles Heating section* on page 48).
Micropositioner	initialization of micropositioner (⇒ *Micropositioner section* on page 49).

1.1 Section for control of injection lines

The columns of this section contain the following functions:

Pos column

With the buttons of the column **Pos**, you control the micropositioner and drive the nozzle block to the requested position:

Button	Function
	– drives the nozzle block to the working position. During this procedure, the nozzle block is driven to a predefined position which ensures that the chosen injection line has a certain orientation to the patterning area. – pressed button marks that nozzle block is in working position.
	– drives the nozzle block to the Standby position. – pressed button marks that the nozzle block is in the Standby position.
	– drives the nozzle block to the park position. – pressed button marks that the nozzle block is in the park position.

1.1.1 Temperature column 

The status LEDs in the column **Temperature** show the heating status of the selected injection line. The status LEDs can have the following states:

Button	Function
	Injection line is cold and all heating elements are switched off.
	current temperature is higher than the set temperature. The injection line is overheated.
	Injection line is heated. The current temperature is less than the set temperature.
	Injection line has reached the set temperature and the precursor is ready for structuring processes.

Button	Function
	clicking on the **STOP** button stops the micropositioner immediately and the nozzle block stays at the current position. To continue driving the nozzle block, select a destination in column **Pos**. **hint:** always click **STOP** when you see that the nozzle block can collide with other parts in the chamber.

1.1.2 Injection lines column

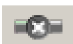
The symbols in the column **Injection Lines** mark the single injection lines and the corresponding precursors. By clicking on one of the symbols you select the injection line.

The commands for the positioning of the nozzle block (column **Pos**), the heating procedures for reservoirs and capillaries (sections **Heating**/**Capillary Heating**) and the outgas procedure (section **Outgas**) always refer to the selected injection line.

1.1.3 Valves column

Via the buttons in the column **Valves**, you open the reservoir valves in order to start a patterning process.

The buttons have the following states:

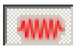
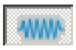
Button	Function
	Shows that the reservoir valve is closed and cannot be opened.
	– Shows that the reservoir valve can be opened. – Opens the reservoir valve for the selected injection line.
	– Shows that the reservoir valve is open. – closes the reservoir valve

1.2 Heating/capillary heating section

In this section you start and control the heating procedure for the reservoir and capillary of the selected injection line.

In USR-Level you can only start the heating procedure for the reservoir of an injection line. The heating elements of the capillaries and the nozzle block will be automatically switched on and heated to the set temperature.

The control and indicator elements have the following states:

Button	Function
	– shows that the heating element is switched off – starts the heating procedure for the capillary
	– shows that the capillary is heated – stops the heating procedure for the capillary
	– shows that the reservoir is cooled – stops the cooling procedure for the reservoir Note: this is only the case for the **Fluorine** injection line.
Set-T	shows the set temperature
Read-T	shows the current temperature

1.3 Outgas section

The buttons in the **Outgas** section allow the user to perform the outgas procedure for the selected injection line:

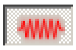
Button	Function
	performs an outgas procedure with the parameters from the subdialog **Properties** (⇒ *Outgas procedure* on page 19).
Details	calls up subdialog **Properties** (**Outgas**) (⇒ *Properties (Outgas) subdialog* on page 49).

1.4 Nozzles Heating section

In the **Nozzles Heating** section, the user control the heating procedure for the nozzle block. The status LED shows the heating state of the nozzle block.

In the USR-Level this heating element is automatically switched on when you start the reservoir heating procedure. The nozzle block is then heated to the set temperature.

The button and the status LED can have the following states:

Button/LED	Function
	shows that the heating element for the nozzle block is switched off
	shows that the heating element for the nozzle block is switched on
	nozzle block is cold
	current temperature is higher than the set temperature. Nozzle block is overheated.
	Nozzle block is heated. The current temperature is less than the set temperature.
	nozzle block has reached the set temperature.
Set-T	shows the set temperature
Read-T	shows the current temperature

1.5 Micropositioner section

Section **Micropositioner** allows the user to initialize the micropositioner (⇒ *Initializing the micropositioner (find home procedure)* on page 21-22). The button and the status field can have the following states:

Button/status field	Function
	performs a find home procedure (⇒ *Initializing the micropositioner (find home procedure)* on page 22).
initialized	micropositioner is initialized
not initialized	micropositioner is not initialized perform a find home procedure (⇒ *Initializing the micropositioner (find home procedure)* on page 22).

1.6 Properties (Outgas) subdialog

Here the user can define the settings for the outgas procedure. Outgas procedures have to be done individually for each injection line one time or several times. In subdialog **Properties** you can set the necessary values or keep the current values for further outgas procedures.

The subdialog is called up via the dialog **Gas Injection Control**/**Gas Flow Control** section **Outgas** Details.

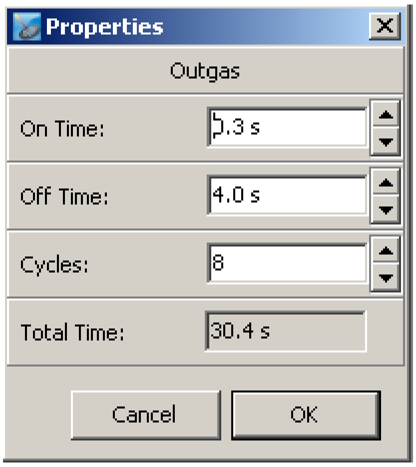


Figure 21-29: **Properties** (**Outgas**) subdialog

In the pull-down menu you can type in values or select values by using the arrow keys:

Menu	**Function**
On Time:	open time for the reservoir valve in seconds
Off Time:	close time for the reservoir valve in seconds
Cycles:	number of opening cycles of the reservoir valve during the outgas procedure
Total Time:	shows the overall time of the outgas procedure in seconds
Cancel	stops the properties definition
OK	confirms the parameters and closes the window

2 Gas Nozzle Positioning dialog (SYS)

In the **Gas Nozzle Positioning** dialog, the user can

- initialize the micropositioner (⇒ *Initializing the micropositioner (find home procedure)* on page 21-22),
- calibrate the NE coordinate system (⇒ *Calibrating the NE coordinate system* on page 21-32),

- define the target position of the nozzle block for every single injection line as well as the park and standby positions (⇒ *Defining the target positions of the nozzle block* on page 21-28).

The window is called up via the control bar .

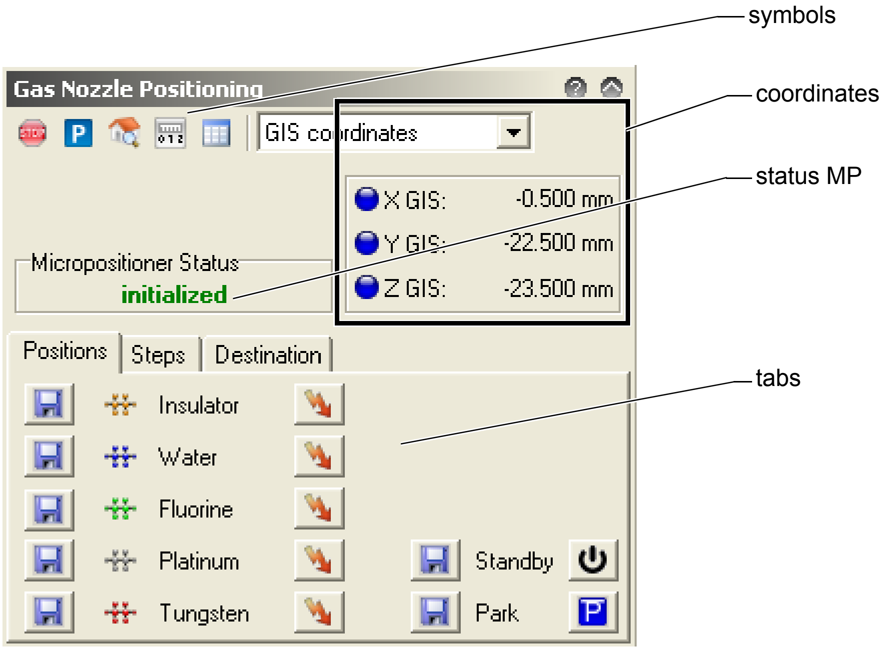


Figure 21-30: **Gas Nozzle Positioning** dialog

The window is divided into the following sections:

Section	Function
symbols	controlling the micropositioner and configuration of parameters (⇒ *The buttons in the symbols have the following functions:* on page 52).
coordinates	selection of the coordinate system (⇒ *Coordinates section* on page 52)
status MP	indicates if the micropositioner is initialized or not

Section	Function
tabs	In the tabs you have various options to position the nozzle block: – **Positions**: move the nozzle block to a predefined target position (⇒ *Positions tab* on page 54). – **Steps**: to move the nozzle block stepwise (⇒ *Steps tab* on page 55). – **Destination**: to move the nozzle block in absolute or relative coordinates (⇒ *Destination tab* on page 56).

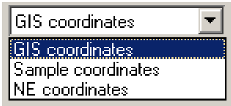
The buttons in the symbols have the following functions:

Button	Function
	Stops the micropositioner immediately. **Note:** Click on **STOP**, when you see that the nozzle block might collide with parts of the chamber.
	drives the nozzle block to the park position
	performs a find home procedure (⇒ *Initializing the micro-positioner (find home procedure)* on page 22).
	calls up the window **Adjust GIS coordinates** (⇒ *Adjust GIS coordinates subdialog (SYS)* on page 56).
	calls up the window **Properties** (⇒ *Properties subdialog (SYS)* on page 58).
	calls up the visualization (⇒ *Coordinates status window* on page 71). **Note:** The visualization is only available in combination with the option nanomanipulators. Accordingly, the button is only visible if at least one nanomanipulator is installed.

2.1 Coordinates section

In the coordinates section, the current position of the micropositioner is shown in the xyz-axis of the selected coordinate system. The status LEDs mark the status of the micropositioner axis.

The control and indicating elements have the following functions:

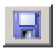
Control, indicating element	Function
	pull-down menu to select the coordinate system for the transformation of the coordinates – **GIS coordinates**: physical coordinates system, in which the nozzle block and the micropositioner drive in the XYZ-direction – **Sample coordinates**: sample coordinate system which orientates on the edges of a sample – **NE coordinates**: virtual coordinate system in which the GIS and sample coordinate system are adjusted
X GIS: **Y GIS:** **Z GIS:**	shows the position of the micropositioner on the XYZ-axis in the selected coordinate system
	micropositioner is out of the park position
	micropositioner is moving
	micropositioner is in the park position

2.1.1 Positions tab

The **Positions** tab allows the user to drive the nozzle block and store the values for the target positions.

Figure 21-31: **Gas Nozzle Positioning** dialog – **Positions** tab

The buttons have the following functions:

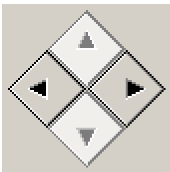
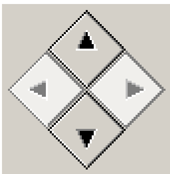
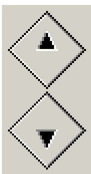
Button	Function
	saves the target position of the nozzle block for the corresponding injection line. The target position is defined with the help of the tabs **Steps** or **Destination**.
	– drives the nozzle block to the predefined working position of the corresponding injection line (⟹ *Defining the target positions of the nozzle block* on page 28). – depressed status marks that the nozzle block is in the working position
	– drives the nozzle block to the standby position (⟹ *Defining the target positions of the nozzle block* on page 28). – depressed status marks that the nozzle block is in the standby position.
	– drives the nozzle block to the park position (⟹ *Defining the target positions of the nozzle block* on page 28). – depressed status marks that nozzle block is in the park position

2.1.2 Steps tab

The **Steps** tab allows the user to move the nozzle block stepwise in XYZ-axis. You can define the stepsize for each of the three axes in mm.

Figure 21-32: Dialog **Gas Nozzle Positioning** – Tab **Steps**

The control elements have the following functions:

Control element	Function
	move the nozzle block stepwise along the x-axis
	move the nozzle block stepwise along the y-axis
	move the nozzle block stepwise along the z-axis
X GIS:	for typing in or selecting the stepsize along the x-axis in millimeters
Y GIS:	for typing in or selecting the stepsize along the y-axis in millimeters
Z GIS:	for typing in or selecting the stepsize along the z-axis in millimeters
	fixes that the nozzle block moves with the same stepsize in y- and z-axis as in x-axis

2.1.3 Destination tab

The **Destination** tab allows the user to move the nozzle block by typing in fixed values for the target position in XYZ-axes. You can define absolute values or values relative to the home position.

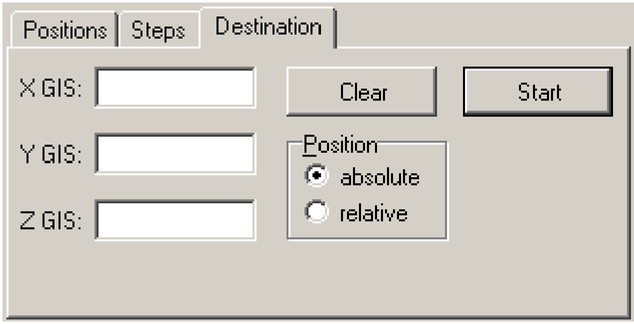


Figure 21-33: **Gas Nozzle Positioning** dialog – Tab **Destinations**

The control elements have the following functions:

Control element	Function
X GIS: **Y GIS:** **Z GIS:**	type in the requested target position on the related axis in mm
Clear	deletes the values
Start	moves the nozzle block to the target position
Position absolute relative	to select if the values for **X GIS**, **Y GIS**, **Z GIS** are absolute or relative

2.2 Adjust GIS coordinates subdialog (SYS)

With the **Adjust GIS coordinates** subdialog, the user calibrates the NE coordinate system. You need the NE coordinate system in order to create a correct visualization of the NE options. The window guides you through the calibration procedure step by step.

The GIS coordinate system has to be calibrated if:

- the nozzle block has been exchanged,
- it is clear that the working distance in the visualization does not fit with the working distance in the column control

The **Gas Nozzle Positioning** dialog is called up by selecting .

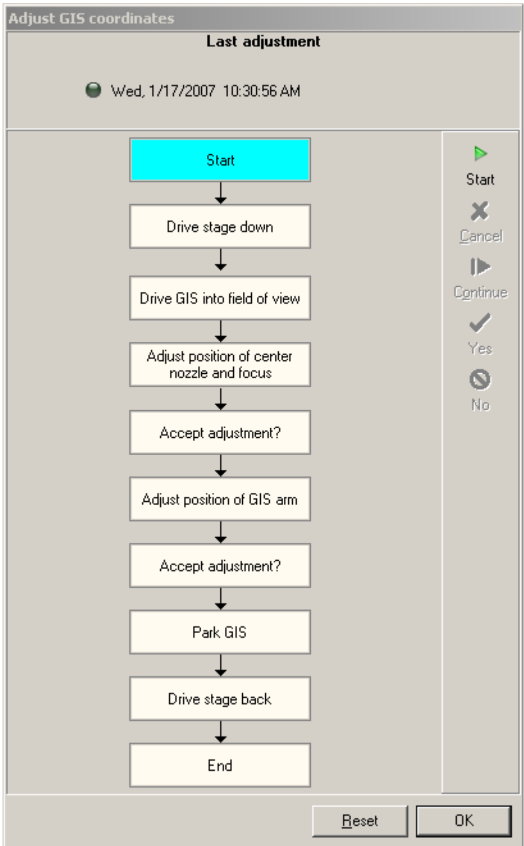


Figure 21-34: **Adjust GIS coordinates** dialog

The control elements have the following functions:

Control element	Function
Last Adjustment	shows date and time of the last calibration procedure
	shows that the calibration procedure was successful
Start	starts the calibration procedure
Cancel	stops the calibration procedure
Continue	continues the calibration procedure after the centre nozzle and the focus have been adjusted (⇒ *Positioning the nozzle and focussing* on page 34)
Yes	confirms a request or an entry in a dialog
No	declines a request

Control element	Function
Reset	resets all settings to the last set values
OK	confirms the calibration procedure and closes the window

2.3 Properties subdialog (SYS)

In the subdialog **Properties** the user can:

- determine the maximum stepsize for movements of the nozzle block in XYZ-axis
- define the set temperatures for reservoir, capillary and nozzle block (⇒ *Setting temperatures for reservoirs, capillaries and nozzle* on page 21-36),
- rename injection lines (⇒ *Renaming injection lines* on page 21-38),
- determine which precursors can be injected together and which ones are locked (⇒ *Using multiple precursors together* on page 21-16),
- check the data connection between the GIS and the PC

The settings have influence on the related functions in the following windows:

- **Gas Nozzle Positioning** (EXP/SYS)
- **Gas Flow Control** (EXP/SYS)
- **Gas Injection Control** (USR)

The user can call up the **Gas Nozzle Positioning** by pressing .

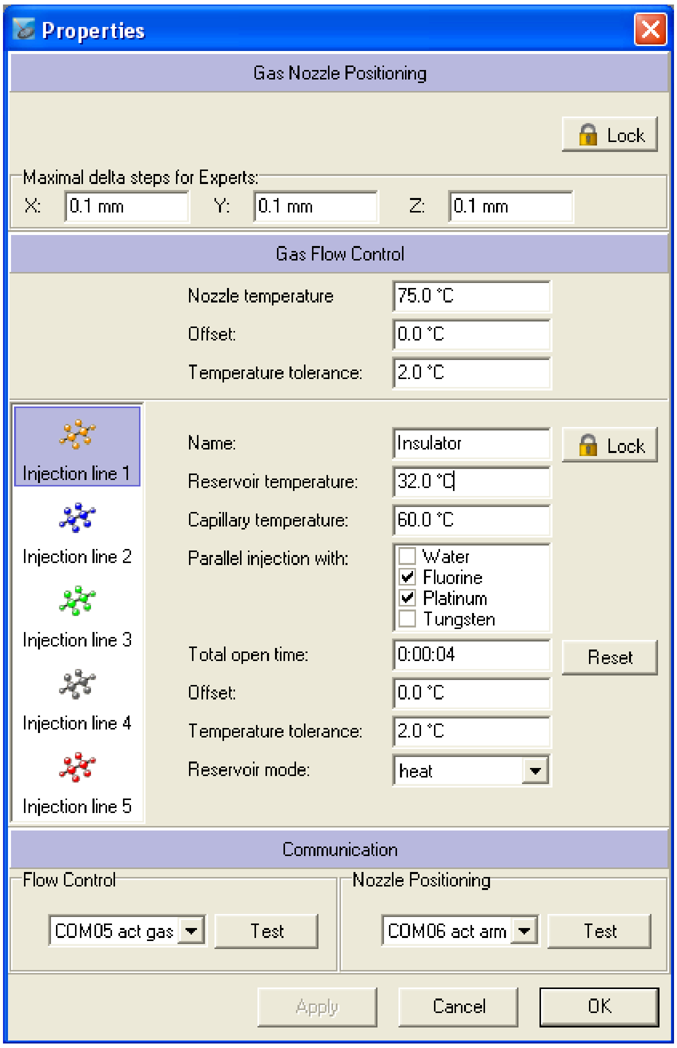


Figure 21-35: **Properties** dialog (SYS)

The window is divided into three sections:

Section	Function
Gas Nozzle Positioning	configuration of the functions in window **Gas Nozzle Positioning** for SYS- and EXP-level user (⇒ *Gas nozzle positioning section* on page 60).
Gas Flow Control	configuration of the injection lines (⇒ *Gas flow control section* on page 60).
Communication	control of the data connection of the GIS and the PC (⇒ *Communication section* on page 61).

2.3.1 Gas nozzle positioning section

The control elements have the following functions:

Field/button	Function
	blocks the XYZ input fields in the **Gas Nozzle Positioning** window for the SYS- and EXP-level user.
Maximal delta steps for Experts: X:, Y:, Z:	configuration of the maximum allowed stepsize for movements of the nozzle block in XYZ-axes. The settings only affect EXP-level user and the related arrow keys in the **Steps** tab of the **Gas Nozzle Positioning** dialog.

2.3.2 Gas flow control section

All settings refer to the selected injection line.

The control elements have the following functions:

Control element	Function
Nozzle temperature:	definition, in °C, of the set temperature for the nozzle block
Offset:	definition of the temperature offset
Temperature tolerance:	definition of the tolerance for the status LEDs in the section **Nozzles Heating** of the dialogs **Gas Injection Control** and **Gas Flow Control**. The temperature tolerance determines the temperature range in which the status LED indicates a reached set temperature (⇒ Figure 21-35 on page 21-59) The status LED in section **Nozzles Heating** is green between 73 °C and 77 °C
Name:	renaming the selected injection line
	locks the selected injection line **Note:** the injection line cannot be selected or controlled in the dialogs **Gas Flow Control** and **Gas Injection Control**
Reservoir temperature:	definition of the set temperature for the reservoirs
Capillary temperature:	definition of the set temperature for the capillaries

Control element	Function
Parallel injection with:	definition of which precursors can be used in parallel
Total open time:	shows the cumulative open time of the reservoir valve in h:min:sec The time measurement gives a hint at the lifetime of the precursor.
Reset	resets the **Total open time:** **Note:** always reset the Total open time when you have exchanged a precursor
Offset:	definition of the temperature offset
Temperature tolerance:	definition of a tolerance value for the behavior of the status LEDs in column of the dialogs **Gas Injection Control** and **Gas Flow Control**.
Reservoir mode:	definition if the reservoir is heated, cooled or switched off

2.3.3 Communication section

In the communication section, the user can check the data connection between the GIS and the PC:

Button/field	Function
Flow Control COM3 busy	selection of the data port for the control of the gas flow.
Nozzle Positioning COM4 busy	selection of the data port for the control of the micropositioner
Test	checks the data connection If there is a failure in the connection an error message or a dialog will appear

2.3.4 Command line

With the buttons in the command line, you confirm the settings in the dialog **Properties**.

Button	Function
Apply	accepts the new settings of the dialog
Cancel	cancels the process and closes the window
OK	confirms the new settings and closes the window

If you have changed several settings in the dialog **Properties** confirm them with the **Apply** button.
By clicking **OK** you can only confirm and apply one single setting. In the case of several new settings only the last one is stored. All the other new values will be reset in case the window is closed.

3 Nanomanipulator visualization dialog

Visualization is only available if at least one nanomanipulator is installed in addition.

In the visualization window the user gets a schematic image of the movements of the GIS nozzle block and the installed nanomanipulators and their toolholders. The visualization provides an overview of the movements of GIS and nanomanipulators inside the chamber and serves as a coarse collision control.

The visualization of the NANOSUITE Application is based on calculated values of the position measurement systems in combination with the calibration procedure. As the accuracy of the calibration procedure is limited the visualization can only be an orientation. It will never replace the checking by the user via image.

The visualization is called up via dialog **Gas Nozzle Positioning**

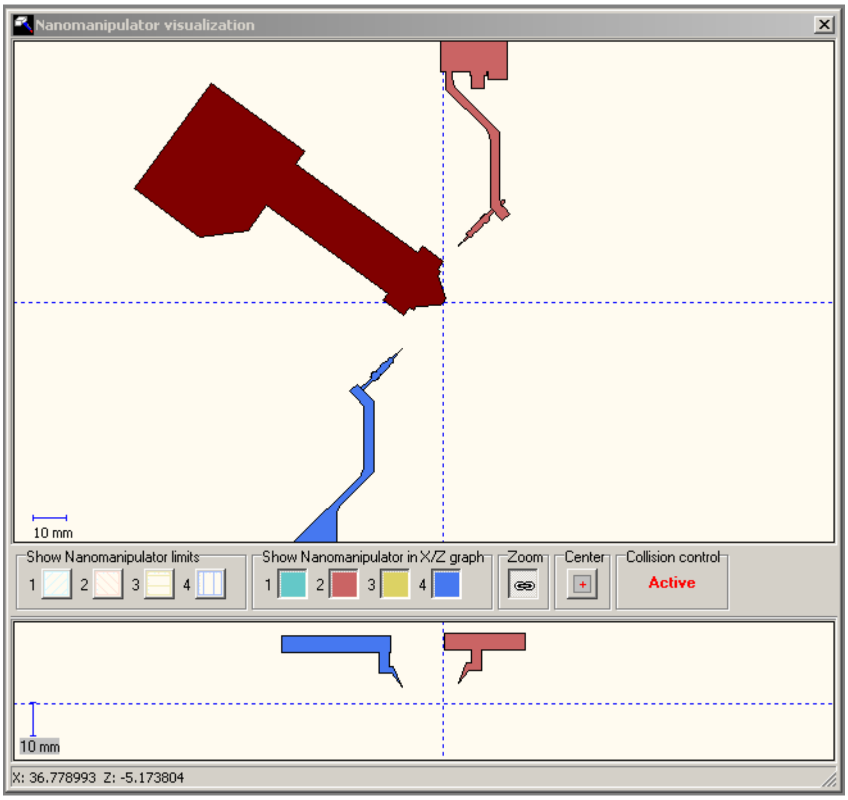


Figure 21-36: **Nanomanipulator visualization** dialog

The graphical illustration in the dialog **Nanomanipulator visualization** consists of three sections:

- The upper section contains a X/Y graphics which is consistent with a view through the chamber ceiling. The crossing of the dotted lines (X=0, Y=0) is equal to the center of the column image.
- In the middle section you can adjust the graphics by optional parameters. The parameters refer to the display of the nanomanipulators. For working with the GIS only, the button **Center** is relevant. By clicking on the button, you can center the display in X/Y.
- The lower section contains a X/Y illustration of the nanomanipulators with which you can estimate the distance between tooltips and sample surface. The crossing of the dotted lines (X=0, Z=0) is equal to the center of the column image at a working distance of 10 mm. The GIS is not shown in the lower section.

You can shift the display with the arrow keys in x- and y-directions.

You can adapt the magnification by moving the mouse wheel within the display (zoom function). During this zooming, the X/Y display does not affect the X/Z display, while zooming in X/Z display adapts the X/Y display in the same way.

The upper section relates to working with the GIS only.

For more information about controlling the display of the visualization window, please refer to the documentation for the specific system being used.

4 Gas nozzle positioning (EXP)

The **Gas Nozzle Positioning** dialog (EXP) enables the user to drive the nozzle block to predefined working positions or address it stepwise.

This dialog in the EXP level is basically analogous to the SYS level. The following functions are not available in the EXP level:

- The **Destination** tab for positioning of the nozzle block in absolute or relative coordinates (**Destination** tab),
- pull-down menu for selection of the coordinate system,
- possibility to save target positions in the **Positions** tab.

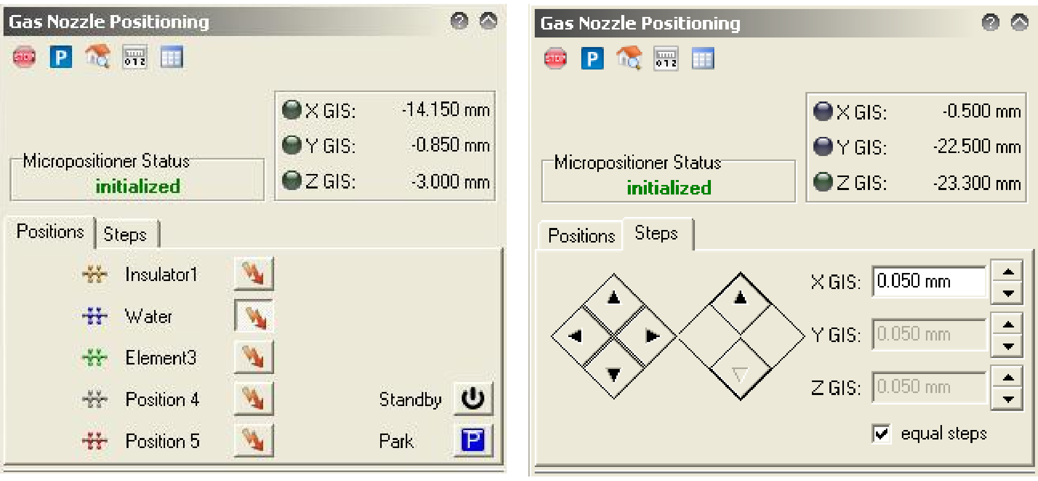


Figure 21-37: **Gas Nozzle Positioning** dialog (EXP) – **Positions**, **Steps** tabs

The control and indicator elements have the same functions as in the SYS-level (⇒ *Gas Nozzle Positioning dialog (SYS)* on page 21-50).

4.1 Properties dialog (EXP)

Properties allows the user to

- check the maximum allowed stepsize for movements of the nozzle block in XYZ-axes,

- set the set temperature for reservoirs and capillaries of each injection line.

  The set temperatures influence the heating functions in the dialog **Gas Flow Control** (EXP) and will be shown in the corresponding fields **Set-T**.

The window **Gas Nozzle Positioning** is called up by selecting .

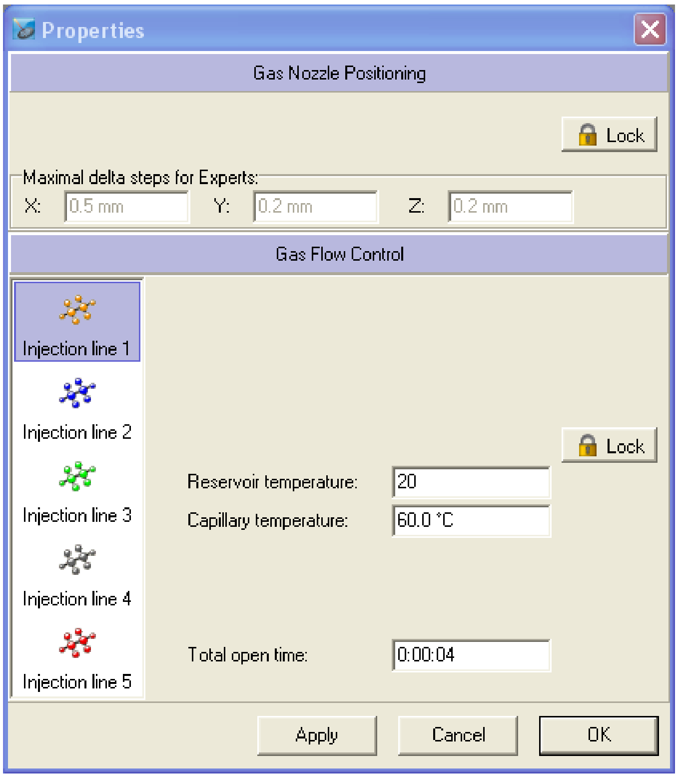


Figure 21-38: **Properties** dialog (EXP)

The window is divided into three sections:

Section	Function
Gas Nozzle Positioning	– indicating the maximum allowed stepsize for stepwise movement of the nozzle block in XYZ-axes. This stepsize is defined in the SYS-level. You move the nozzle block stepwise by using the arrow keys in the dialog **Gas Nozzle Positioning**➡tab **Steps** (⟹ *Gas nozzle positioning (EXP)* on page 64). – locking the XYZ-axes for USR-level.

Section	Function
Gas Flow Control	definition of the set temperature for reservoirs and capillaries (⇒ *Gas flow control section* on page 60). **Note: the possible temperature values depend on preset values.** These values are set by the SYS-level users. In EXP-level you can vary the temperature by +/- 10°C from the preset values. If the typed in values are outside this range, an error message will be shown.

4.1.1 Gas flow control section

All settings affect the selected injection line.

The fields and buttons have the following functions:

Field/button	Function
	locks the reservoir valve
Reservoir temperature:	definition of the set temperature for the reservoir
Capillary temperature:	definition of the set temperature for the capillary
Total open time:	shows the cumulative open time of the reservoir valve in h:min:sec The time measurement gives a hint at the lifetime of the precursor
Apply/OK	applies all new settings
Cancel	interrupts the procedure and closes the dialog
OK	applies all new settings and closes the dialog

5 Gas Flow Control dialog (EXP/SYS)

This window allows the user to

- start deposition and etching processes (EBID/EBIE),
- start the heating procedure reservoir and capillary of an injection line and the nozzle block,
- execute an outgas procedure.

The window is called up by selecting ➡ .

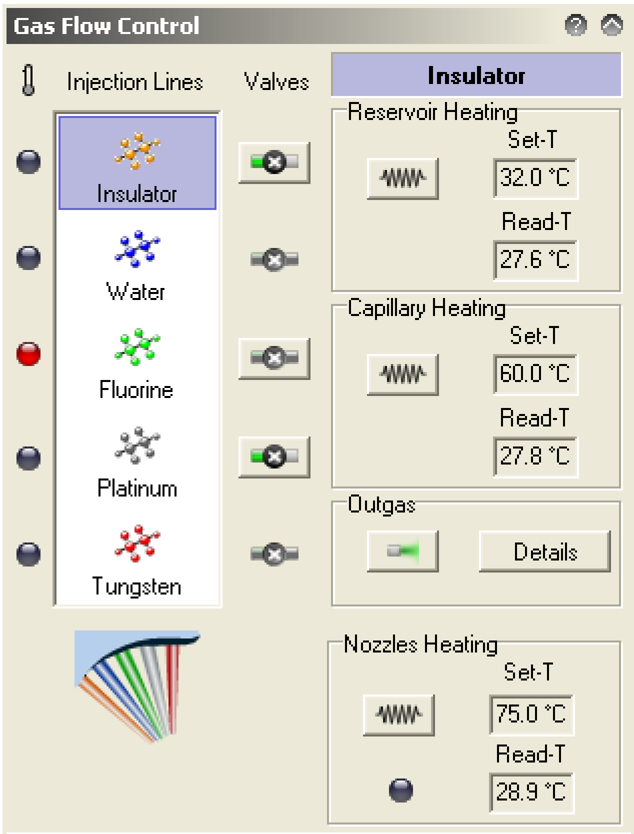


Figure 21-39: **Gas Flow Control** dialog

The columns, sections and control elements have the following functions:

5.1 Temperature column 

The status LEDs in the temperature column show the heating status of the selected injection line. The status LEDs can have the following states:

Button	Function
	injection line is cold and all heating elements are switched off
	current temperature is higher than set temperature, injection line is overheated
	injection line is heated. The current temperature is lower than the set temperature
	injection line has reached the target temperature and the precursor is ready to use

5.2 Injection lines column

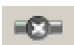
The symbols in the injection lines column mark the single injection lines and their precursors. By clicking on one of the symbols, you select the injection line with the precursor that you want to use for patterning.

The commands for the positioning of the nozzle block (**Pos** column), the heating procedure for reservoirs, capillaries and nozzle block (section **Heating**, **Nozzles Heating**) as well as the outgas procedure (section **Outgas**) always refer to the selected injection line.

5.3 Valves column

Via the buttons in the **Valves** column, you open the reservoir valves in order to start a patterning process.

The buttons are context-sensitive, dependent on the injection line. They can have the following states:

Button	Function
	Shows that the reservoir valve is closed and cannot be opened.
	– Shows that the reservoir valve can be opened. – Opens the reservoir valve for the selected injection line.
	– Shows that the reservoir valve is open. – Closes the reservoir valve

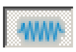
5.3.1 Textfield Insulator

indicates which injection line is selected currently what the states of the status LEDs refer to in dialog **Gas Injection Control** – in this example injection line insulator **Isolator**.

5.4 Reservoir heating and capillary heating sections

In the **Reservoir Heating** and **Capillary Heating** sections, the user can start and control the heating procedure of the reservoir and capillary of the selected injection line. The components are heated up to the set temperature which has been defined in **Properties** (⇒ *Properties subdialog (SYS)* on page 21-58).

The control elements have the following functions:

Control element	Function
	– shows that the heating element is switched off – starts the heating procedure
	– shows that the reservoir is heated – stops the heating procedure for the reservoir
	– shows that the reservoir is cooled – stops the cooling procedure for the reservoir Note: this is only the case for the injection line of **Fluorine**.
Set-T	shows the set temperature for reservoir/capillary
Read-T	shows the current temperature

5.5 Outgas section

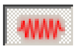
In the **Outgas** section, the user performs one or several outgas procedures for a selected injection line:

Button	Function
	performs an outgas procedure with the parameters from the **Properties** subdialog
Details	calls up **Properties** subdialog (⇒ *Gas Nozzle Positioning dialog (SYS)* on page 50).

5.6 Nozzles heating section

In the **Nozzles Heating** section, the user heats the nozzle block. The status LED shows the heating status of the nozzle block.

The control and indicating elements have the following functions:

Button	Function
	– shows that the heating element is switched off – starts the heating procedure
	– shows that the nozzle block is heated – stops the heating procedure for the nozzle block
	nozzle block is cold
	current temperature is higher than the set temperature. The nozzle block is overheated.
	Nozzle block is heated. The current temperature is less than the set temperature.
	nozzle block has reached the set temperature
Set-T	shows the set temperature for the nozzle block
Read-T	shows the current temperature of the nozzle block

6 e_LiNE load lock navigator

Please refer to the basic documentation for the system being used.

Via the **Navigator e_LiNE load lock** dialog, the user can perform automated load and unload procedures in order to load or unload a sample holder into the chamber.

NOTICE

Impact on system vacuum during loading of sample holders.

Only load or unload the sample holder in combination with an installed GIS using the automated load and unload procedures of the navigator.

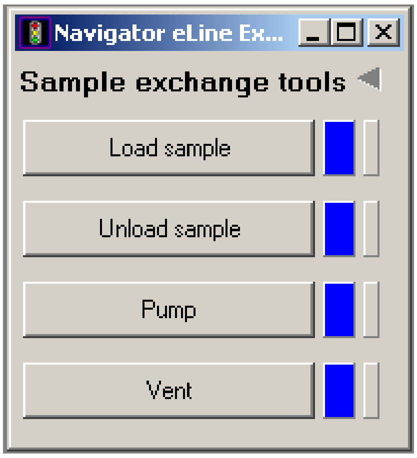


Figure 21-40: **Navigator e_LiNE load lock** dialog

The buttons have the following functions:

Button	Function
Load sample	performs a load procedure
Unload sample	performs an unload procedure
Pump	evacuates the system
Vent	vents the system

7 Coordinates status window

The current coordinates of the sample stage and the sample coordinate system of the loaded sample are displayed. The status LEDs mark the status of the sample stage.

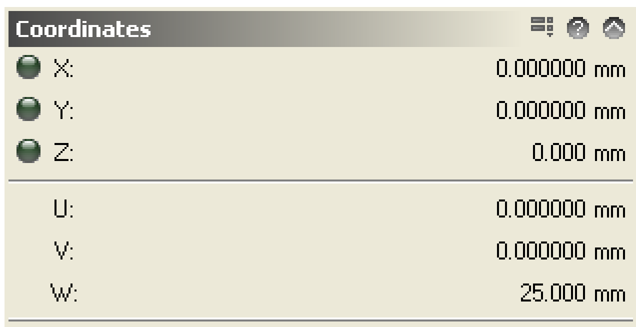


Figure 21-41: **Coordinates** status window (USR)

The indicators and LEDs have the following functions:

Indicator/LED	Function
X:, Y:, Z:	shows the position of the sample stage in XYZ-axes in mm
	indicates that the sample stage is not moving
	indicates that the sample stage is moving
U:, V:, W:	shows the position of the sample in the sample coordinate system

Chapter 22
Nanomanipulators (option)

This chapter describes how to manipulate a sample and make electrical measurements via the **Nanomanipulators** software module in combination with at least one nanomanipulator.

The **Nanomanipulators** module is an optional module of the NANOSUITE software and only available, if the lithography system is equipped with one or more nanomanipulators.
The nanomanipulators can be installed in Raith eLINE and ionLINE systems. In the following, the software control of the nanomanipulators in the eLINE system is described.
NanoSense is an optional module and will replace one manipulator.

For a detailed description of the control and operation of the nanomanipulators, as well as the safety information refer to the *Nanomanipulators Operating Instructions*.

This chapter contains the following topics and tasks:

- Short overview of the hardware and software equipment
(⇒ *Functional Description* on page 22-2),
- How to position a nanomanipulator tip in order to manipulate a sample or make electrical measurements
(⇒ *Positioning a nanomanipulator (approach process)* on page 22-5),
- How to perform linescans for topographic measurements using the NanoSense
(⇒ *NanoLinescans* on page 11-24),
- How to use the scouting functionality using the NanoSense
(⇒ *Working with Follow Scout function* on page 22-19),
- How to avoid collisions automatically
(⇒ *Working with the Collision Control* on page 22-18),
- Description of all functions and control elements of the **Nanomanipulators** software module
(⇒ *Software Reference* on page 22-27).

A Functional Description

This chapter gives a short overview of the basic functions of the nanomanipulators hardware and software equipment.

1 Hardware

The eLINE system can be equipped with up to four nanomanipulators, which are installed inside the chamber. The nanomanipulators can be combined with all other optional eLINE components.

One of the nanomanipulators can be replaced by the NanoSense, which is a sensing manipulator. The NanoSense is equipped with a sensing tip, which enables the possibility of 3-dimensional information.

With the nanomanipulators, the sample can be manipulated, in-situ, right down to the level of nanometers. Electrical measurements can be made.

Due to the combination of a high resolution position measuring system with a closed control loop, the tips of the tool can be precisely visually and digitally positioned on the sample down into a range of less than 10 nm.

Every nanomanipulator can be controlled via its own keypad or via the **Nanomanipulators** software module. Using the keypad, you can simultaneously move in several directions at once. Using the NANOSUITE software you can only move the nanomanipulator along one axis.

The NANOSUITE software also makes it possible to use the positionlist as a tasklist to drive the nanomanipulators to predefined positions. These positionlists can be stored using any filename and subsequently called up again.

Depending on the application, one or more nanomanipulators can be removed or installed by the user. A tip that has been damaged during operation can be changed by the user.

For a detailed description of the nanomanipulators hardware, as well as the safety information refer to the *Nanomanipulator Operating Instructions*.

2 Nanomanipulator software module

The software control for the nanomanipulators has been completely integrated in the NANOSUITE software and consists of four main dialogs:

- **Nanomanipulator** dialog to control and configure one or all installed nanomanipulators as well as the NanoSense
  (⇒ *Nanomanipulator dialog* on page 22-27),
  (⇒ *Nanomanipulator dialog Parameter Field* on page 22-29),
- **Nanomanipulator Control** dialog with functions for moving a selected nanomanipulator
  (⇒ *Nanomanipulator Control dialog* on page 22-42),
- Nano Workbench Control dialog (optional) to edit the settings for the NanoSense
  (⇒ *NanoWorkbench Control (optional)* on page 22-44),
- Nanomanipulator Coordinates dialog to monitor the current XYZ positions and status of the installed nanomanipulators
  (⇒ *Nanomanipulator Control dialog* on page 22-42).

(⇒ Figure 22-1 on page 22-4) gives an overview of the **Nanomanipulators** module:

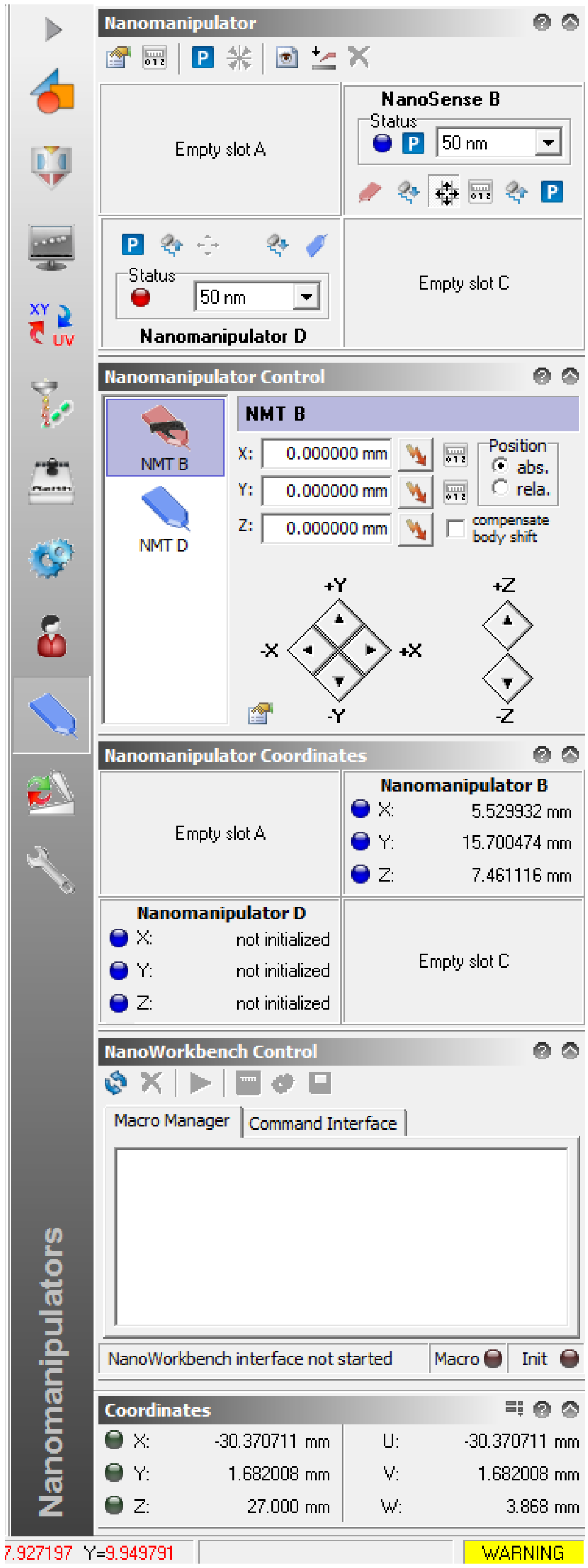


Figure 22-1: **Nanomanipulators** software module – Main dialogs

B Tasks

This chapter describes how to

- position the tip precisely on the sample surface
  (⇒ *Positioning a nanomanipulator (approach process)* on page 22-5).
- work with the collision control
  (⇒ *Working with the Collision Control* on page 22-18).
- work with Linescans to create topographic information
  (⇒ *NanoLinescans* on page 11-24)
- work with scouting function for accurate z movement
  (⇒ *Working with Follow Scout function* on page 22-19)

1 Positioning a nanomanipulator (approach process)

The following sections will explain the positioning of the nanomanipulator.

1.1 Background

To be able to use a nanomanipulator for manipulation of a sample or for electrical measurements, you must be able to reliably position the tip on the surface of the sample without causing any damage.

Positioning the tip is an iterative process. Move the tip towards the surface of the sample in smaller and smaller steps, until the surface is reached.

The precision of the distance values interpolated by the NANOSUITE software is limited.
The image of the electron or ion optics can provide significantly more precise information about the distance. Thus, you must

- adjust the working distance of the electron or ion optics to the working distance of the NANOSUITE software,
- always use the image to check the real distance between the tip and the surface of the sample during the approach process: With a suitable magnification, set the focus alternately on the sample and the tip. Determine the real distance by reading the image values into the NANOSUITE software and calculating the difference (⇒ *Example approach process* on page 22-6).

Additionally to the image, you can use the **Nanomanipulator Visualization** as guidance (⇒ *Nanomanipulator visualization dialog* on page 22-37).

1.2 Example approach process

In the following, an example approach process is described using nano-manipulator D:

STEP 1: Preparing the approach process (starting position)

In the following example the tip is positioned exactly in the center. The description thus concentrates solely on the approach along the Z axis. If, in practice, you would like to aim for a different point on the surface of the sample then you must align the nanomanipulator so that the X and Y values match the desired location.

- ▶ Using the joystick, drive to a point on the sample where you want to apply the nanomanipulator.
- ▶ Move the sample to a working distance (WD) of 10 mm into the field of view.
- ▶ Set the focus on the sample.
- ▶ Move the tip into the field of view with click on in the **Nanomanipulator** dialog (⇒ Figure 22-3 on page 22-8).

  The nanomanipulator will be moved on its Z axis to a distance of 1 mm between tip and sample surface. During the movement, the corresponding **Status** LED illumines green.

- ▶ Center the tip by setting the X and Y axis to 0 mm in the **Nanomanipulator Control** dialog.
- ▶ Synchronize the working distance measured by the electron or ion optics and the NANOSUITE software:
  - Change to the **UVW** module with click on in the control bar.
  - Click on Global to open the **Adjust UVW (Global)** dialog.
  - Change to the **Adjust W** tab.
  - If applicable, disable **Read from CCD Control**.
  - Read the current working distance of the electron or ion optics with click on .

    The system will prompt to adjust the column focus.
  - Check the focus in the image of the electron or ion optics and confirm the prompt asking.
  - In the **Adjust UVW (Global)**, click on **Adjust** to take over the working distance measured by the electron or ion optics.
- ▶ Change back to the **Nanomanipulators** module.

▶ In the **Nanomanipulator** dialog, click on . The **Nanomanipulator: Approach Tool** dialog will be opened (⇒ Figure 22-3 on page 22-8).

▶ In the **Nanomanipulator: Approach Tool** and **the Nanomanipulator Control** dialog select the Nanomanipulator D.

The starting position is set.

(⇒ Figure 22-2 on page 22-7) to (⇒ Figure 22-4 on page 22-8) show the starting position in the NANOSUITE software and the corresponding images:

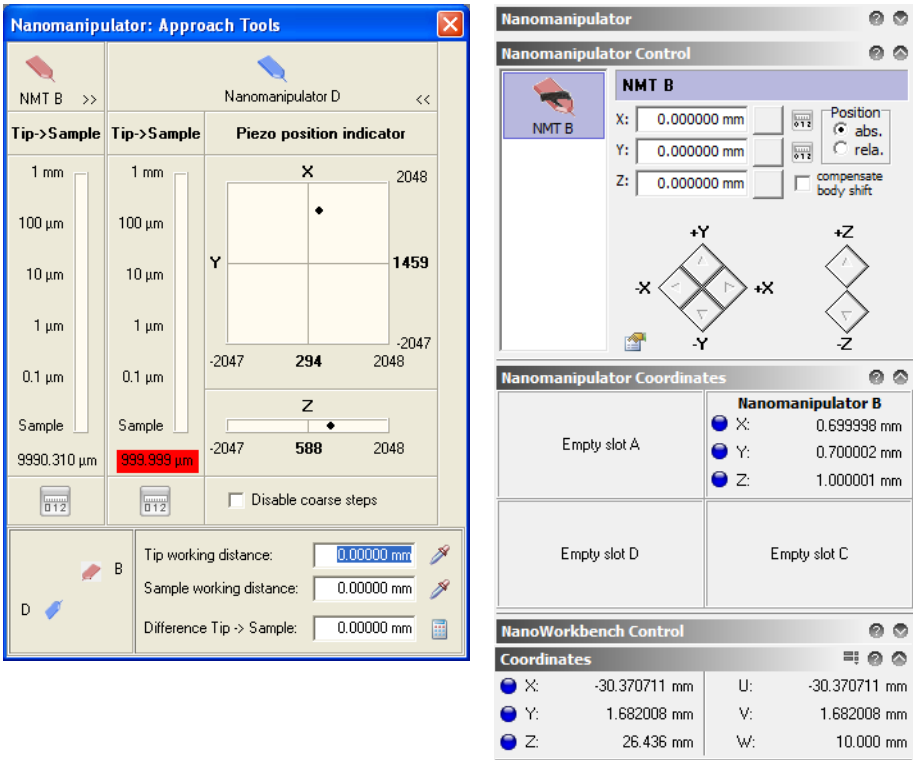


Figure 22-2: Approach process STEP 1a: Starting position. WD to sample: 10 mm. Real distance tip-sample: 1 mm. X, Y values: 0

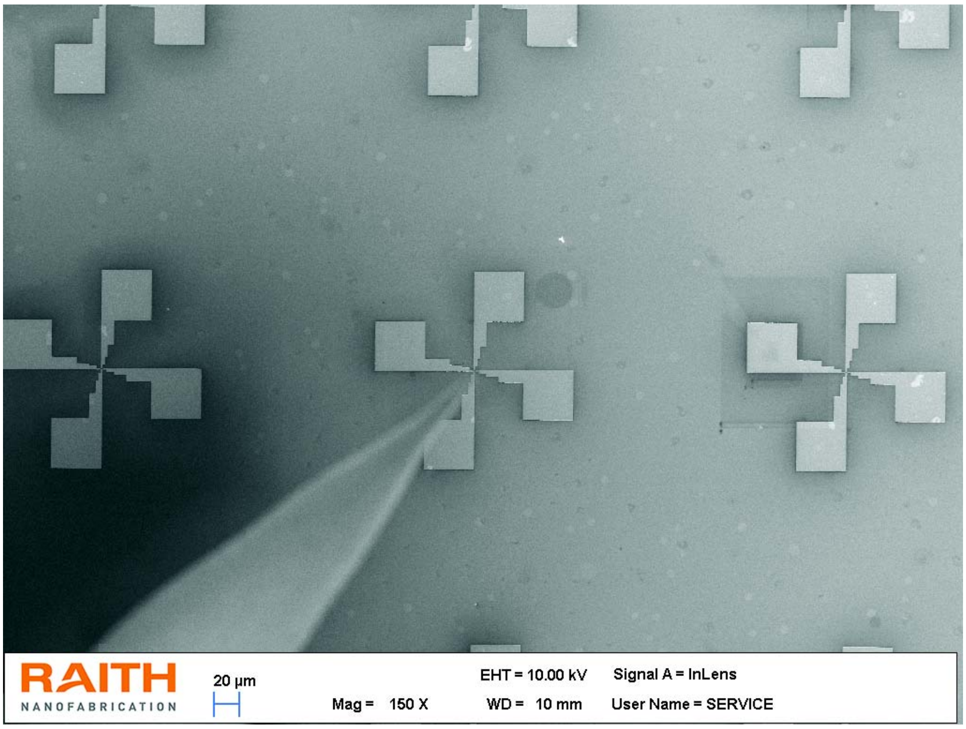


Figure 22-3: Approach process STEP 1b: Image of starting position. Focus on sample, WD at 10 mm (Magn. 150x)

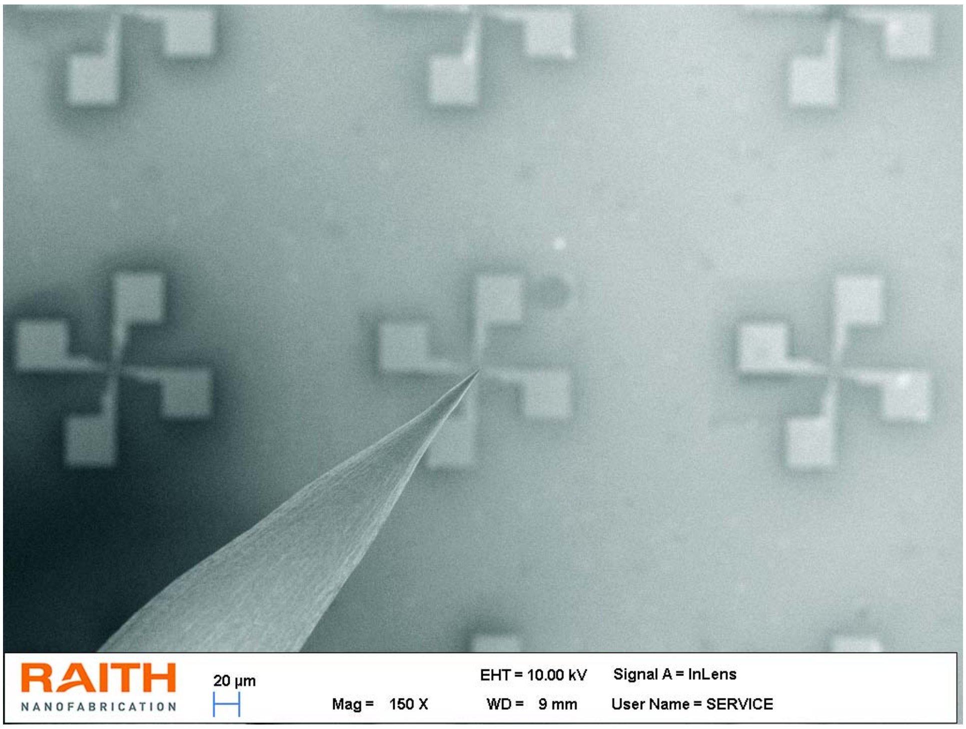


Figure 22-4: Approach process STEP 1c: Image of starting position. Focus on tip, WD at 9 mm (Magn. 150x). Real distance sample - tip = 1 mm

STEP 2: Approaching to about 100 µm using **NANOSUITE** software

- Move the nanomanipulator to a distance of about 100 µm by entering the corresponding value for **Z:** (0.1 absolute) in the **Nanomanipulator Control** dialog.

- Click on next to the **Z:** input field.

  The nanomanipulator will be moved correspondingly and the Z value will be displayed in the **Nanomanipulator Coordinates** dialog (⇒ Figure 22-5 on page 22-10).

STEP 3: Checking real distance between sample and tip

Check the real distance between the sample and the tip using the electron or ion optics and the **Nanomanipulator: Approach Tools** dialog:

- Set the focus on the sample (⇒ Figure 22-6 on page 22-10).

- Take over the focus value by clicking on next to the **Sample working distance** input field (⇒ Figure 22-5 on page 22-10).

- Set the focus on the tip (⇒ Figure 22-7 on page 22-11).

- Take over the focus value with click on next to the **Tip working distance** input field.

- Click on next to the **Difference Tip -> Sample** field to calculate and display the difference between the focus values.

Proceed in the same way to determine the real distance in every following approach step.

(⇒ Figure 22-5 on page 22-10) to (⇒ Figure 22-7 on page 22-11) show the values of the first approach in the NANOSUITE software and the corresponding images:

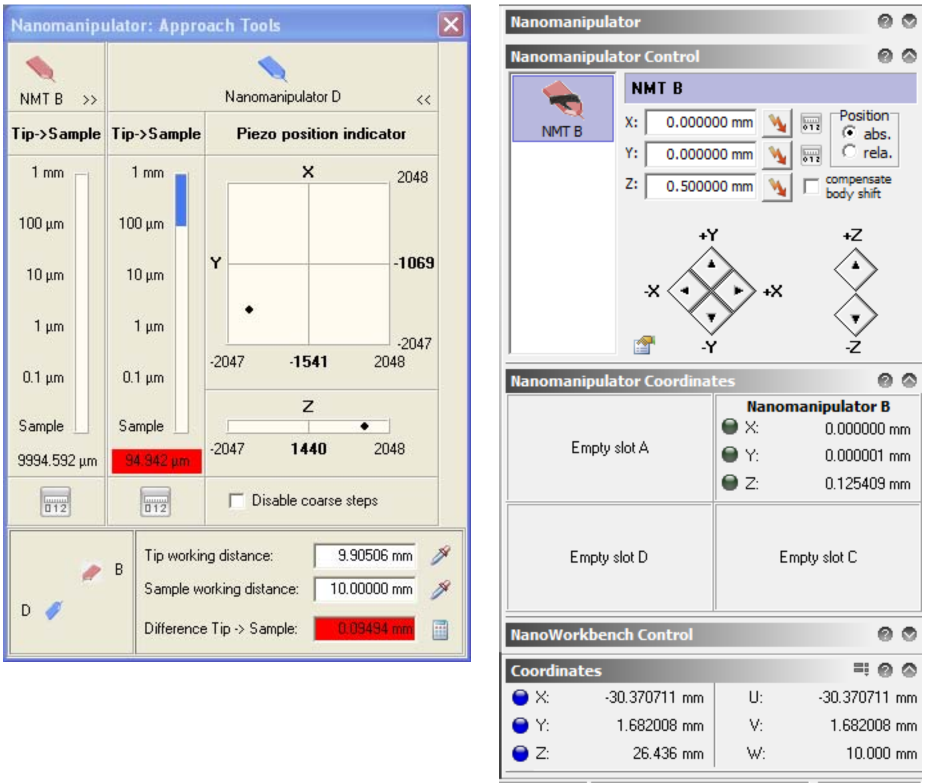


Figure 22-5: Approach process STEP 2a: Software values of the 1st approach. WD to sample: 10 mm. Real distance tip-sample: 125.500 µm. X, Y values: 0

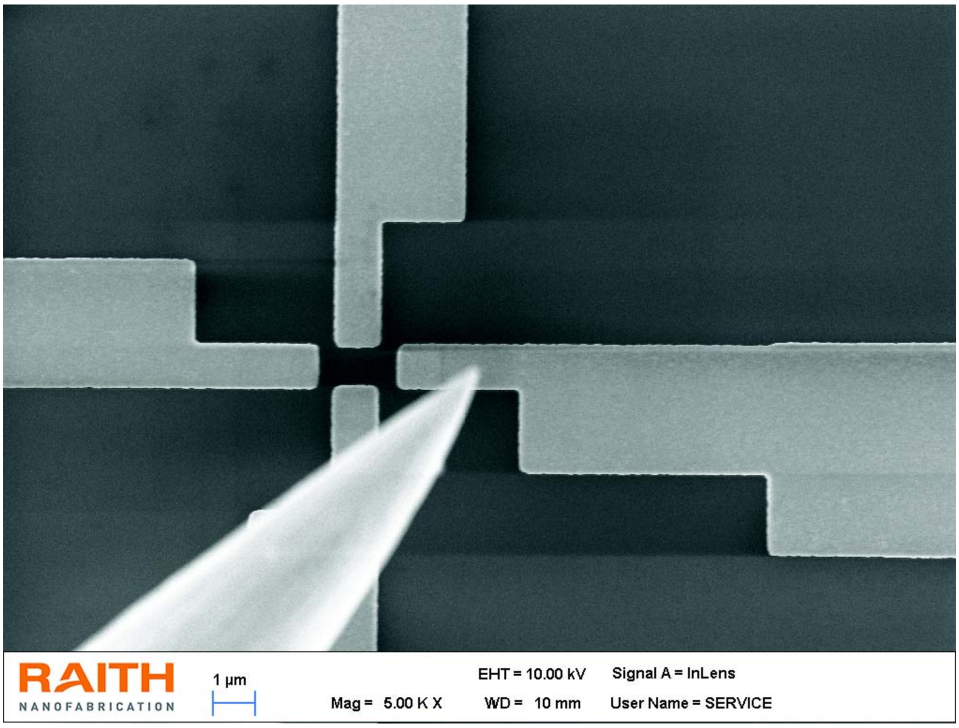


Figure 22-6: Approach process STEP 2b: Image of 1st approach. Focus on sample, WD at 10 mm (Magn. 5 000x)

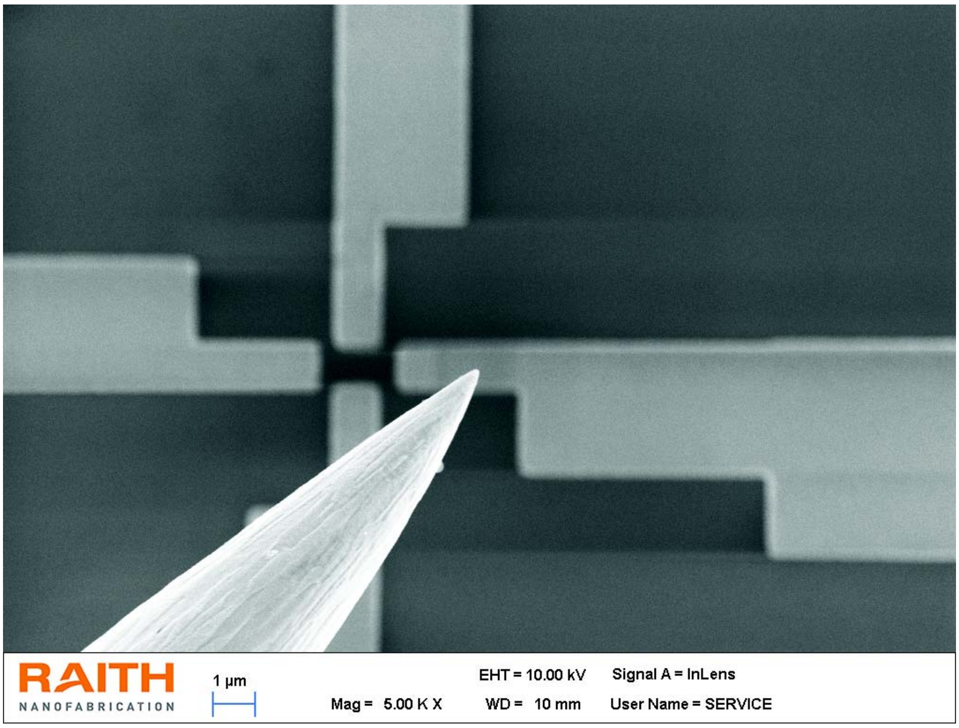


Figure 22-7: Approach process STEP 2c: Image of 1st approach. Focus on tip, WD at 9.90506 mm (Magn. 5 000x). Real distance sample - tip = 125.500 µm

STEP 4: Approaching to about 30 µm using NANOSUITE software

- Move the nanomanipulator to a distance of about 30 µm by entering the corresponding value for **Z:** (0.03 absolute) in the **Nanomanipulator Control** dialog.

- Click on  next to the **Z:** input field.

  The tip will be moved correspondingly and the Z value will be displayed in the **Nanomanipulator Coordinates** dialog (⇒ Figure 22-8 on page 22-12).

- Check the real distance between the sample and the tip using the image and the **Nanomanipulator: Approach Tools** dialog (⇒ *Checking real distance between sample and tip* on page 22-9).

  (⇒ Figure 22-8 on page 22-12) to (⇒ Figure 22-10 on page 22-13) show the values of the second approach in the NANOSUITE software and the corresponding images:

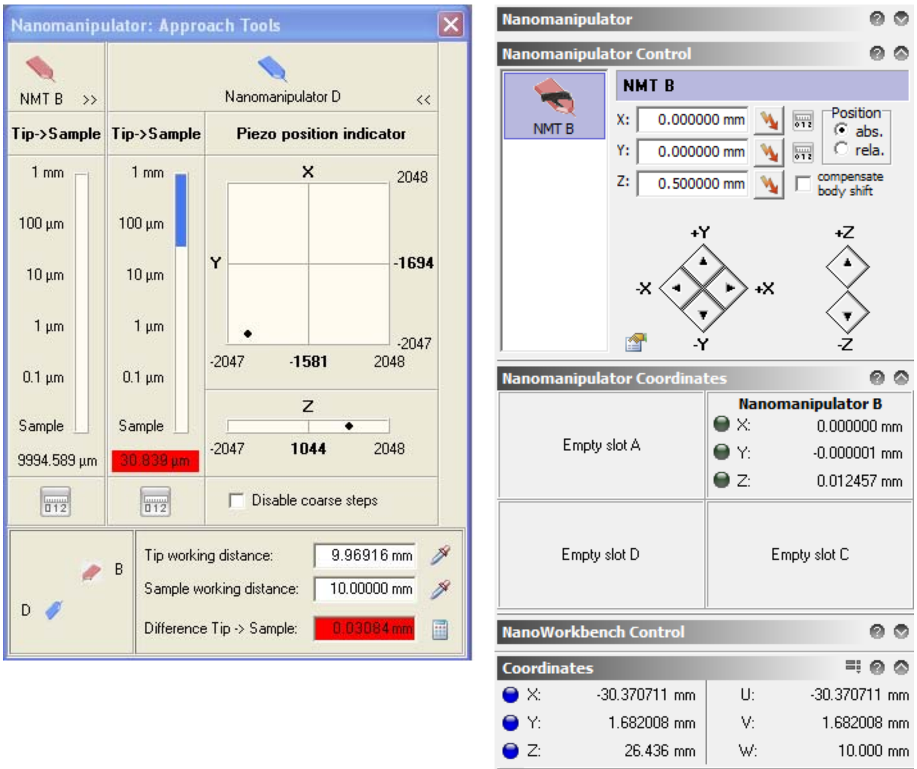


Figure 22-8: Approach process STEP 3a: Software values of the 2nd approach. WD to sample: 10 mm. Real distance tip-sample: 12.548 µm. X, Y values: 0

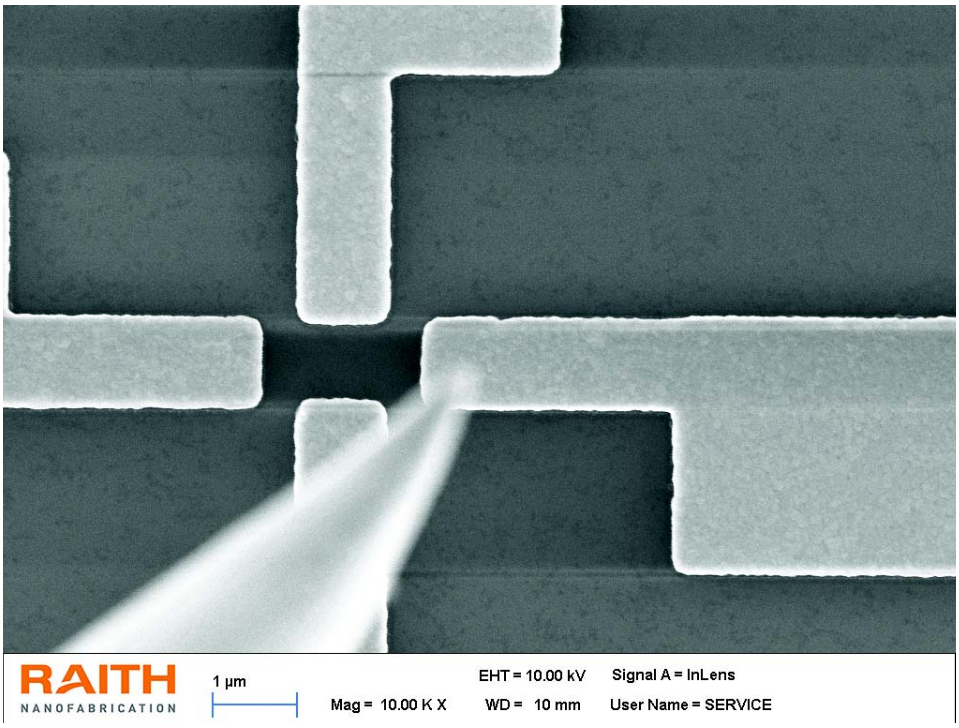


Figure 22-9: Approach process STEP 3b: Image of 2nd approach. Focus on sample, WD at 10 mm (Magnification 10 000x)

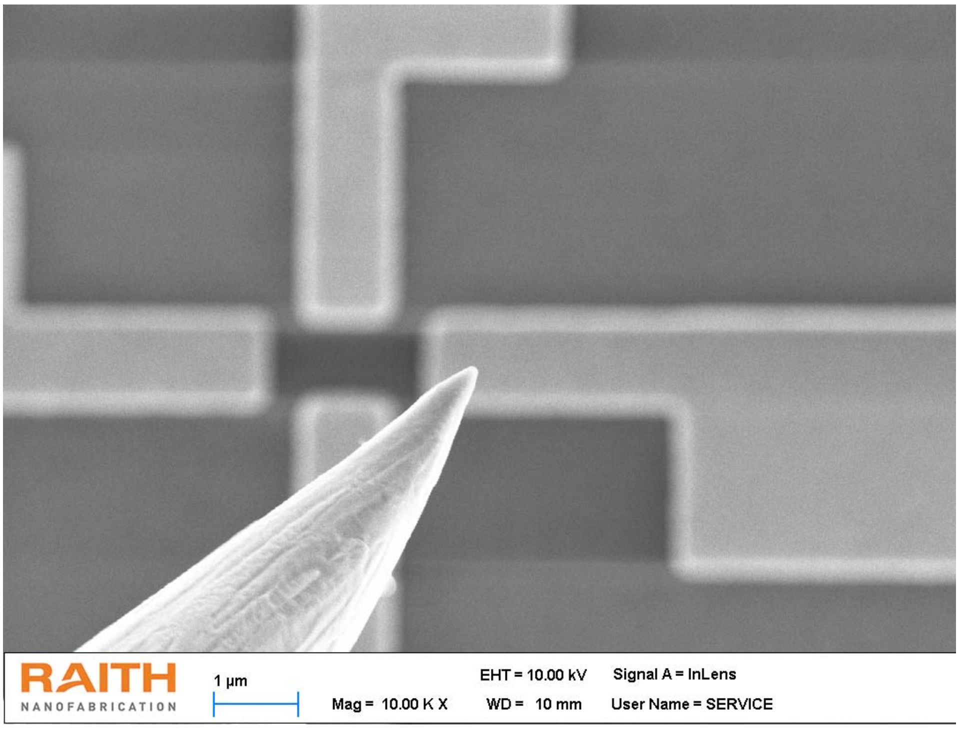


Figure 22-10: Approach process STEP 3c: Image of 2nd approach. Focus on tip,
WD at 9.970144 mm (Magnification 10 000x) Real distance sample - tip = 30.839 µm.

STEP 5: Approach to about 5 µm using the digital pads of the **NANOSUITE** software

From a distance of about 30 µm, the positioning should only be done in very small steps via the digital pads or the keypad.

▶ Move the nanomanipulator to a distance of about 5 µm by entering the corresponding value for **Stepsize Z:** (0.005) in the **Nanomanipulator Control** dialog:

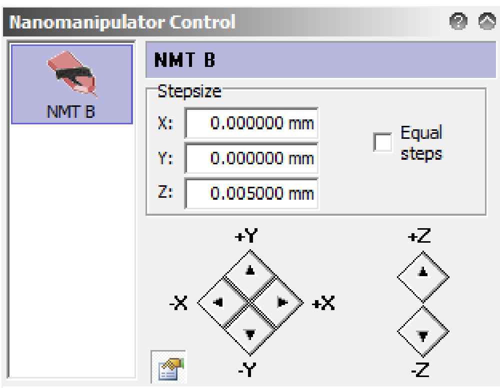


Figure 22-11: Approach process step 4: Lowering step size for digital pads

- Lower the tip by clicking on the **-Z** button whilst constantly visually monitoring the real distance via the image.
- Check the real distance between the sample and the tip using the image and the **Nanomanipulator: Approach Tools** dialog (⇒ *Checking real distance between sample and tip* on page 22-9).

STEP 6: Visually controlled approach to about 1 µm using the keypads with reduced active step size

---

From the distance of about 1 µm the positioning should now only be done with constant visual monitoring, via the image, using the keypad.
The step size of the keypad should be successively reduced as you do this.
In addition you should reduce the size of the scan field for the image and increase the scan frequency so as to get a higher frame repeat rate (please refer to system basic documentation).

- Reduce the active step size by selecting a value in the Status combo box of the Nanomanipulator dialog – in this example **0.5 µm**:

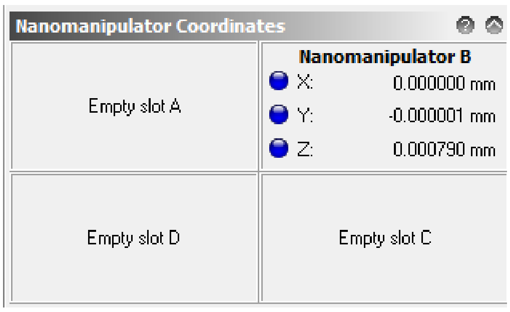


Figure 22-12: Approach process step 5a: Lowering step size for keypad

- Reduce the size of the scan field for the image and increase the scan frequency (please refer to system basic documentation).
- Lower the tip by pushing the **-** key on the keypad whilst constantly visually monitoring the real distance via the image.
- Check the real distance between the sample and the tip using the image and the **Nanomanipulator: Approach Tools** dialog (⇒ *Checking real distance between sample and tip* on page 22-9).

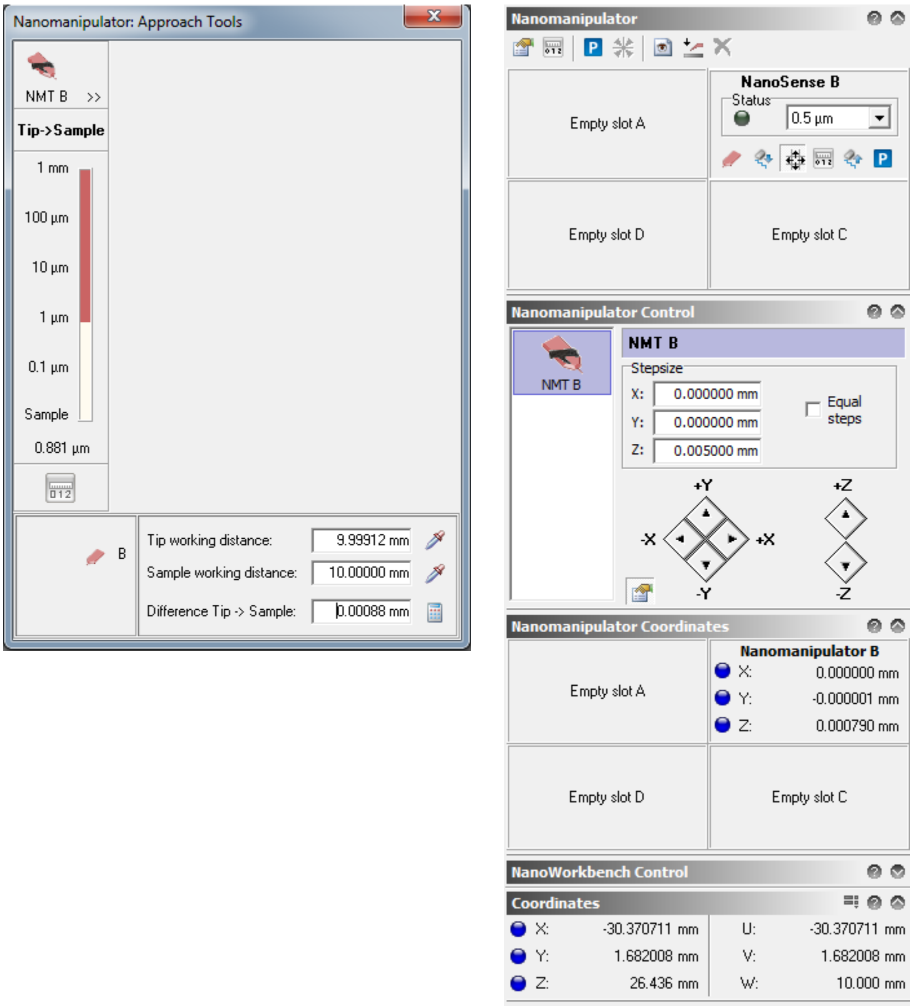


Figure 22-13: Approach process STEP 5a: Software values of the 4th approach. WD to sample: 10 mm. Real distance tip-sample: 0.881 µm. X, Y values: 0

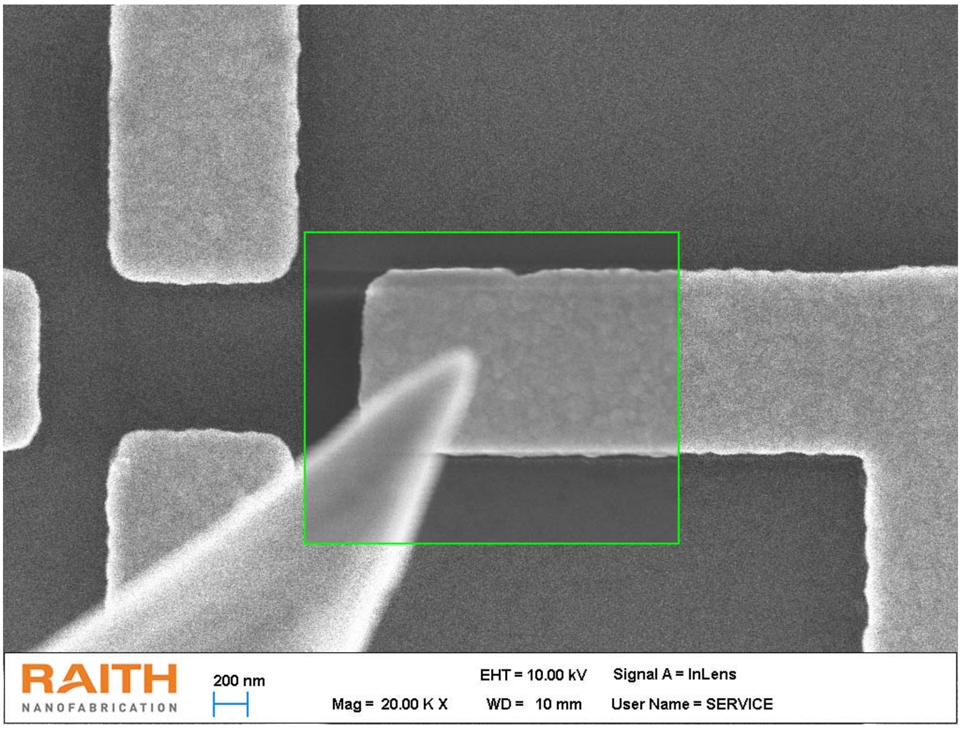


Figure 22-14: Approach process STEP 5b: Image of 4th approach with reduced scan field. Focus on sample, working distance (WD) at 10 mm (Magn. 20 000x)

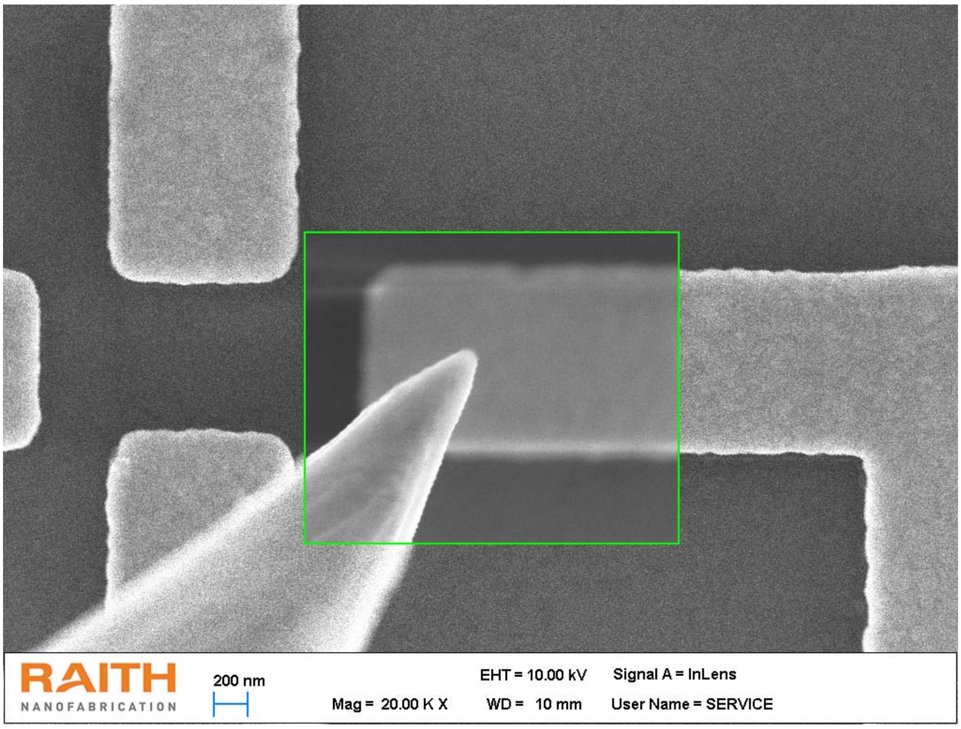


Figure 22-15: Approach process STEP 5c: Image of 4th approach with reduced scan field. Focus on tip, WD at 9.99900 mm (Magn. 20 000x)
Real distance sample - tip = 0.881 µm.

STEP 7: Visually controlled approach until contact is made using the key-pads with active step size reduced even further

In the nanometer range you should again reduce the value of the step size of the keypad – e.g. 100 µm.

- Reduce the value of the active step size further more.
- Move the nanomanipulator in the direction of the sample by pushing the - key on the keypad whilst constantly visually monitoring the real distance via the image.
- If necessary, increase the magnification once more and move the tip carefully onto the surface of the sample. Avoid lowering the tip too far so as not to damage the sample or the tip.

The tip has made contact when

- it has deflected slightly as it slid along the surface of the sample,
- there is a noticeable movement in X- and Y-direction,
- a small shadow appears around the tip.

(⇒ Figure 22-16 on page 22-17) shows a tip which has made contact.

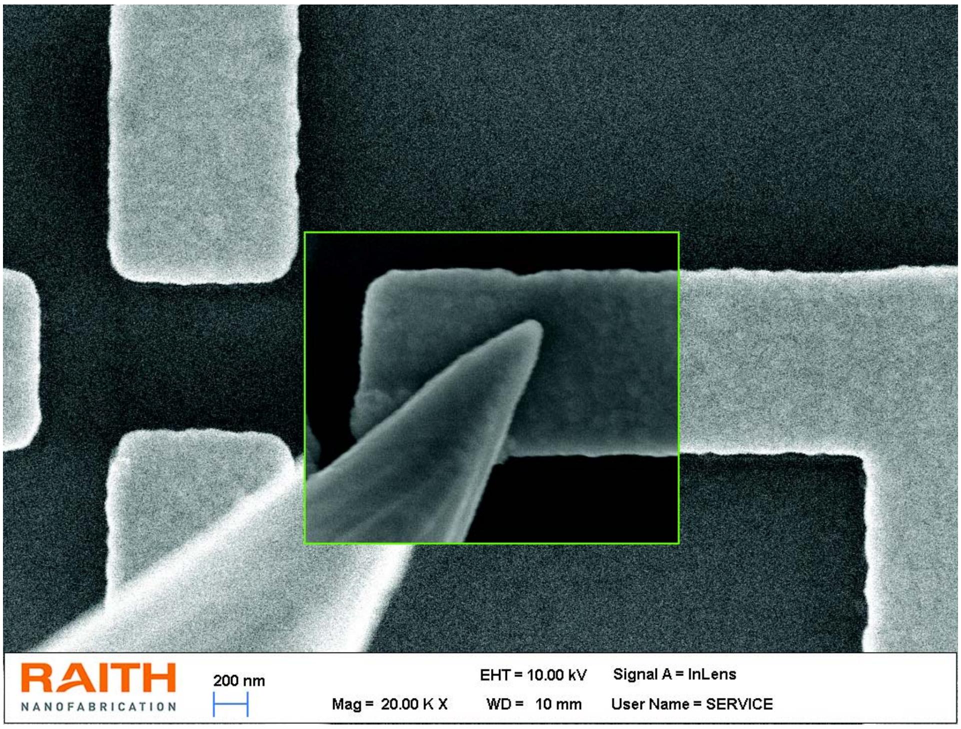


Figure 22-16: Approach process STEP 6: The tip has made contact. Image, motion controlled via the keypad, focus on the sample with a reduced scan field and an increased scan frequency (Magn. 20 000x).

STEP 8: Lifting the nanomanipulator

After the contact is made proceed as follows:

- ▶ Press the **Lift up Arm Save** button.
- ▶ With the keypad, move the nanomanipulator into its quadrant in field of view.
- ▶ Move all nanomanipulators to their parking positions with click on **P** in the **Nanomanipulator** dialog.

2 Working with the Collision Control

With the collision control, the nanomanipulator software module offers a function to automatically prevent collisions of the tips with each other and with the GIS nozzle head.

The collision control is based on a 2-dimensional model, which is equivalent to the view through the roof of the chamber. That means that the control only monitors the movement of the devices in X and Y directions. The corresponding vertical offset of the tips on the Z axis are not considered.

Depending on the application, it could be necessary to position the tips more closely together than the collision control will permit. For those cases it is possible to deactivate the collision control, preassuming that the vertical offset of the tips on the Z axis is adequate. Proceed as follows when working with the collision control:

Before each movement of the nanomanipulators the control will check if there would be a collision with another nananomanipulator or the GIS. If there is a potential collision, the propulsion command will not be executed and the following prompt message will be displayed:

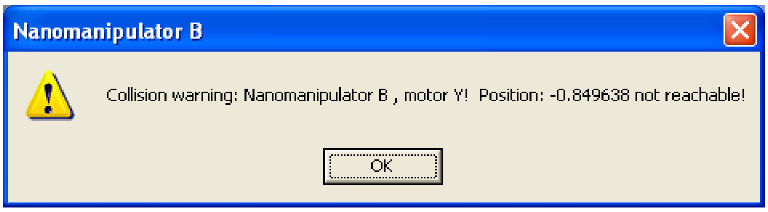


Figure 22-17: Prompt message – Collision control

- ▶ Confirm the prompt message and select a reachable position.
- ▶ As an experienced user with SYS-level rights: If necessary:
  - Deactivate the collision control via the **Nanomanipulator Configuration** dialog ➡ **Collision control** checkbox.
  - Position the tip under permanent control of the working distance via the

– image by focussing alternately on one of the tips.
– **Nanomanipulator: Approach Tool** dialog by monitoring the **Tip -> Sample** values.

3 Working with Follow Scout function

The NanoSense measures defined positions with respect to the Z value. These measured Z values are then used to drive the nanomanipulators precisely to that Z value.

STEP 1: Measuring Z values with NanoSense

▶ Go to the **Nanomanipulator** dialog box.

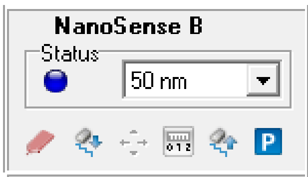


Figure 22-18: **Nanomanipulator** dialog – Parameter field

▶ Select the icon **Use Follow Scout** , which is the 4th icon and drag & drop it into the image.

▶ Repeat this to define several positions, wherever the NanoSense should measure the precise Z value, so that the nanomanipulators can be driven there safely.

▶ The NanoSense will drive to each selected position and measure it. The Z values will then be stored in the Positionlist database. The Z values measured by NanoSense are automatically stored in the Positionlist.

▶ Once all required Z values have been measured by the NanoSense, the Positionlist can be started, to drive the nanomanipulators to the defined Z values.

STEP 2: Defining Follow Scout Briefing for Field of View

▶ Go to the Nanomanipulator dialog and select the icon **Follow Scout to Surface** . The Z values previously measured by the NanoSense will now be used to drive the selected nanomanipulator to the measured Z value.

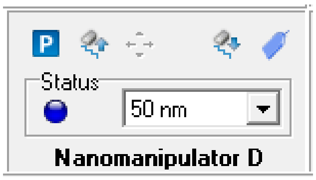


Figure 22-19: **Nanomanipulator** dialog – Parameter field

- The **Follow Scout Briefing** dialog box will open. The user can select one of the following options: **Do nothing**, **Drive stage below** or **Drive tip above**. Place a checkmark next to the required action.

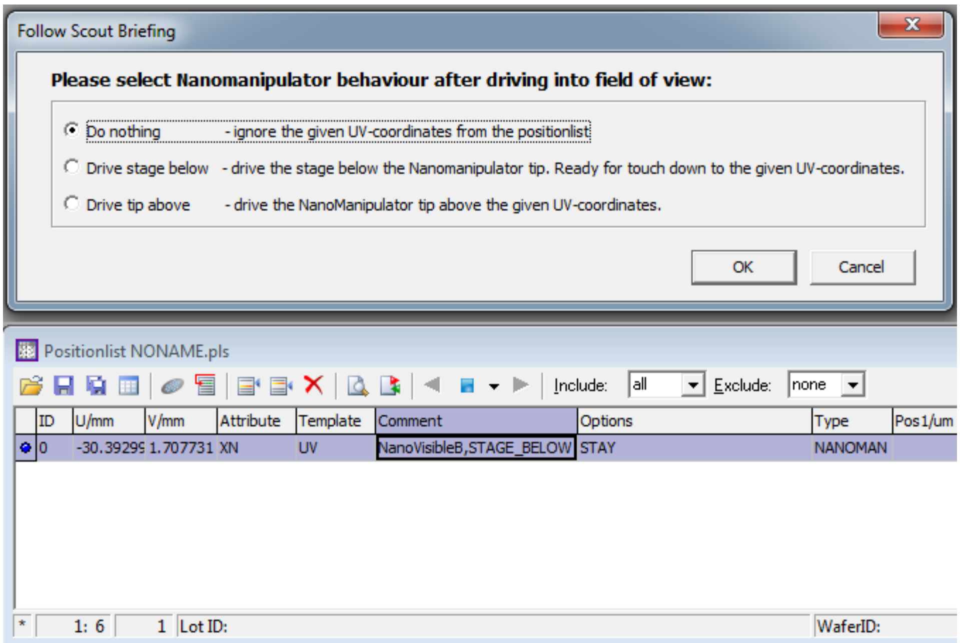


Figure 22-20: Defining the Follow Scout Briefing to move into field of view.
Z values previously measured by NanoSense are displayed in the Positionlist.

STEP 3: Defining Follow Scout Briefing for moving towards surface

- The Follow Scout Briefing dialog enables the user to select either a soft, regular or hard approach to the surface. Using the soft approach, the tip will be placed 5 µm above the sample surface. Regular approach means that the tip will be driven to the contact point. As soon as the tip is in contact with the surface, the tip will be stopped. Hard approach drives the tip a short distance below the contact point to improve the electrical contact. The tip will be driven 1 um into the sample. These values are predefined in the VDB. Place a checkmark next to the required approach.

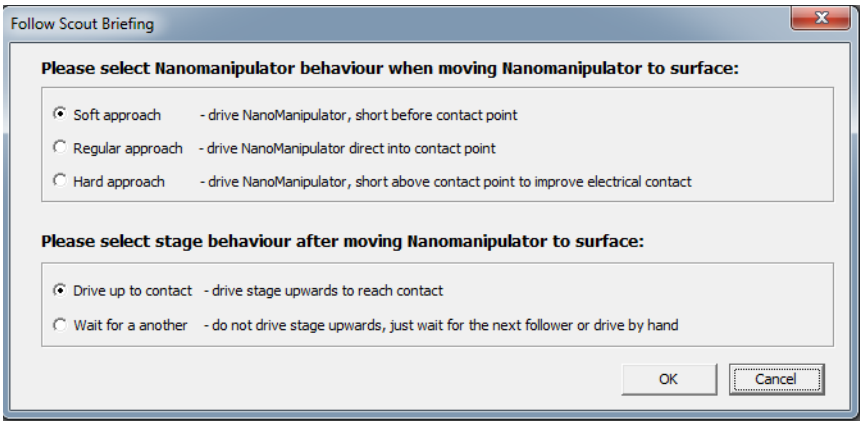


Figure 22-21: Selecting the nanomanipulator and stage behavior.

▶ The user can then select the stage behavior, to either drive the stage up to contact or to wait. In the first case, the stage will now take control over the approach to the tip of the nanomanipulator to establish the contact. Place a checkmark to select the required stage behavior. If the second option is chosen, the user will drive the nanomanipulator manually to the surface.

4 Working with Linescans using NanoSense

This task explains how to perform a Linescan, using the NanoSense to measure the Z values precisely beforehand. This task will only describe the additional functionalities available when the NanoSense option has been installed. All standard functions of Linescans can also be used for the NanoSense. Using the NanoSense Linescans, only the Z values will be measured using a defined linescan. This will result in a topographic linescan, in which the different heights of the sample surface along a linescan are measured. These values can then be used to set the nanomanipulators to precise Z values onto the sample surface along a linescan.

The **Linescan** chapter will familiarize the user with the functioning of the Linescan, particularly with the drag& drop function into an image, which results in a value being written into the Positionlist (⇒ *Drag and drop* on page 9-65).

The **Positionlist** chapter will familiarize the user with the functioning of the Positionlist, (⇒ *Positionlist* on page 9-1).

The **Scan Manager** chapter will familiarize the user with the functioning of the Scan Manager, particularly the drag & drop function, (⇒ *Drag and drop function* on page 11-55).

STEP 1: Entering the NanoSense Linescans in the Positionlist

- Go to **Scan Manager** and select **NanoSense LineScans**. Drag&drop it into the image. This position will now automatically be entered into the Positionlist.

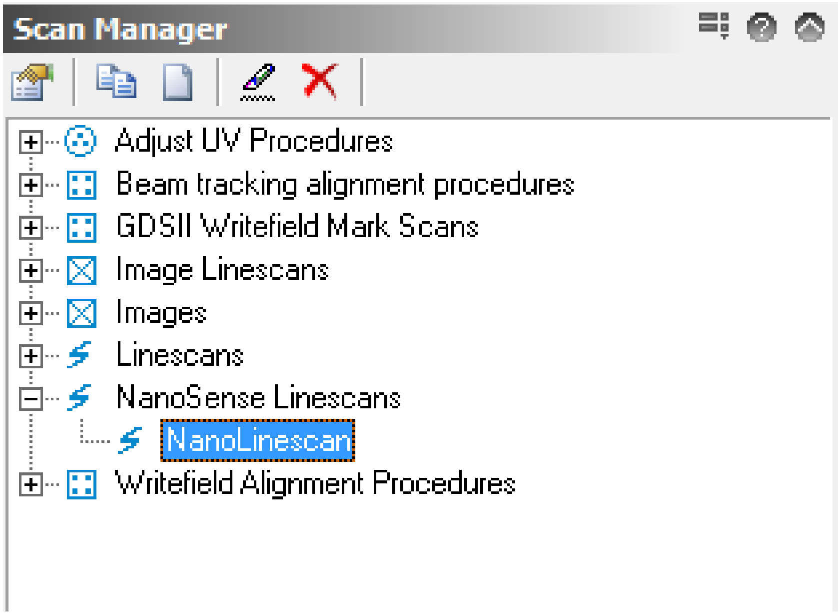


Figure 22-22:Selecting NanoSense Linescans

▶ Repeat the drag&drop function for any other linescans that need to be run. They will all be stored in the Positionlist. Once completed, the Positionlist can be run.

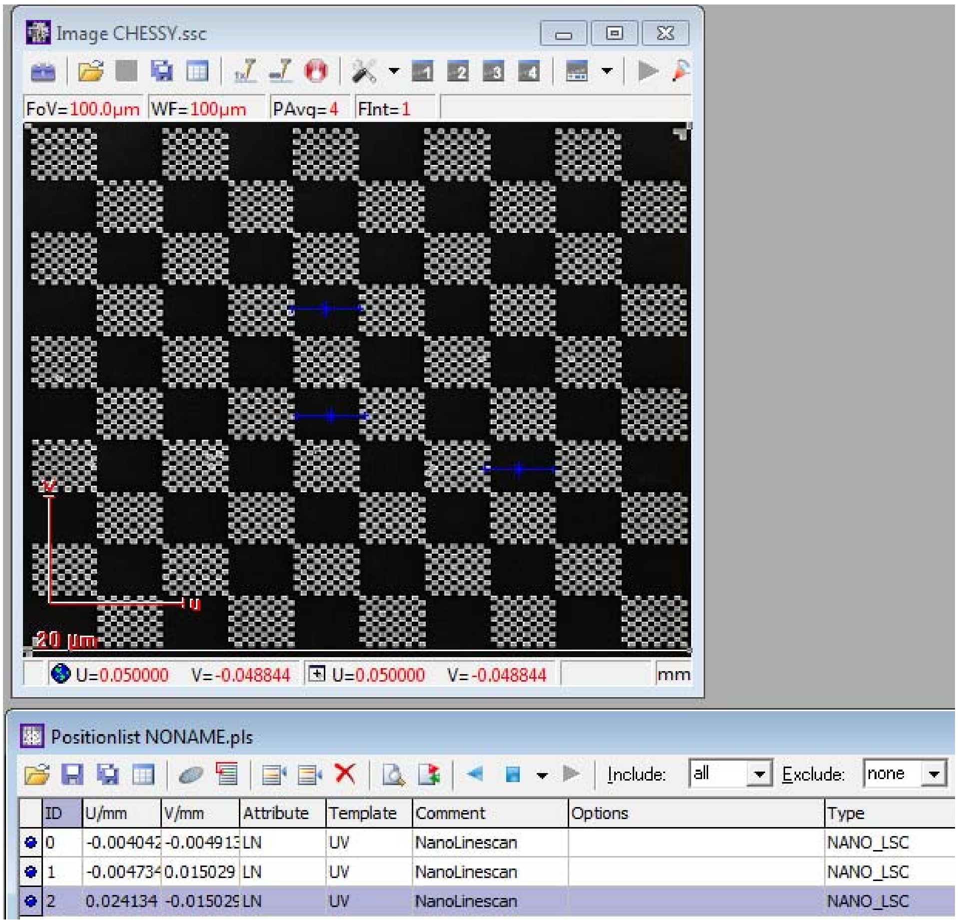


Figure 22-23: Positionlist showing 3 NanoSense Linescans, positioned in the image above.

STEP 2: Running the NanoSense Linescans via the Positionlist

- The next step is to run the Positionlist. Go to **Scan > Selection**, this will now run one linescans sequentially. The linescan using the NanoSense will now run over the distance of the defined line.

- To adjust the length of the NanoSense Linescan, go to **Properties > Placement**. This can be opened via double click of the NanoSense Linescan in the Positionlist.

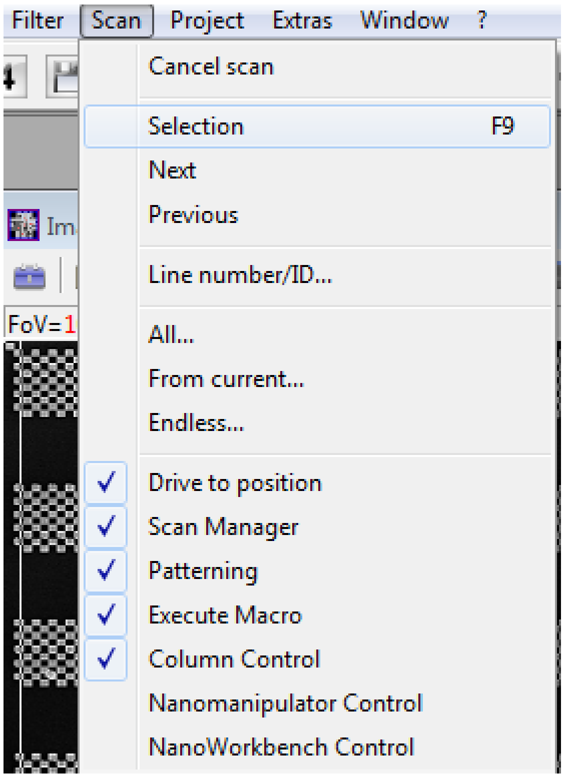


Figure 22-24: Running the NanoLinescans via the positionlist.

STEP 3: Measuring the heights in the NanoLinescans

- The results of the NanoLinescans can now be viewed in order to measure the various heights. The linescan can be opened via a double click on the appropriate linescan name in the Positionlist. The **Linescan** window will now open.

- Select **MAH-manual height** from the dropdown menu. Use the <right mouse> button to select the lower and higher Z values of a peak to be measured in the LineScan graphic, clicking with the <right mouse> button will set the Z values in the graph. The height difference will be automatically displayed.

- In the example below, five different heights are displayed in the NanoLinescan. It is useful to know the height differences within the NanoLinescan.

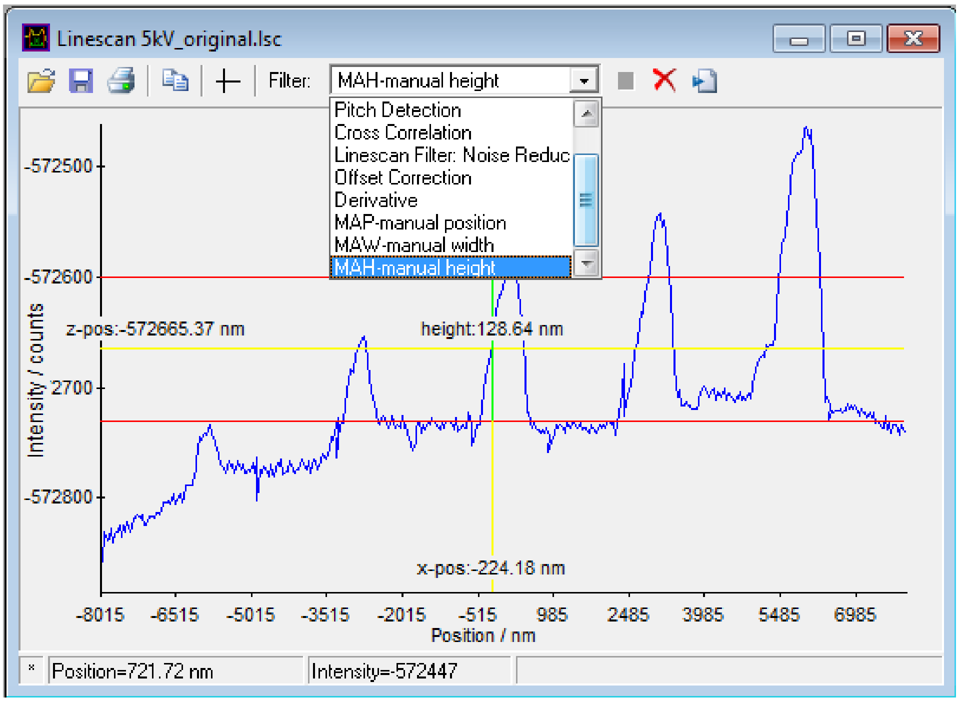


Figure 22-25: Measuring the heights in a NanoLinescan.

- Use the <right mouse> button to measure different heights along the Linescan

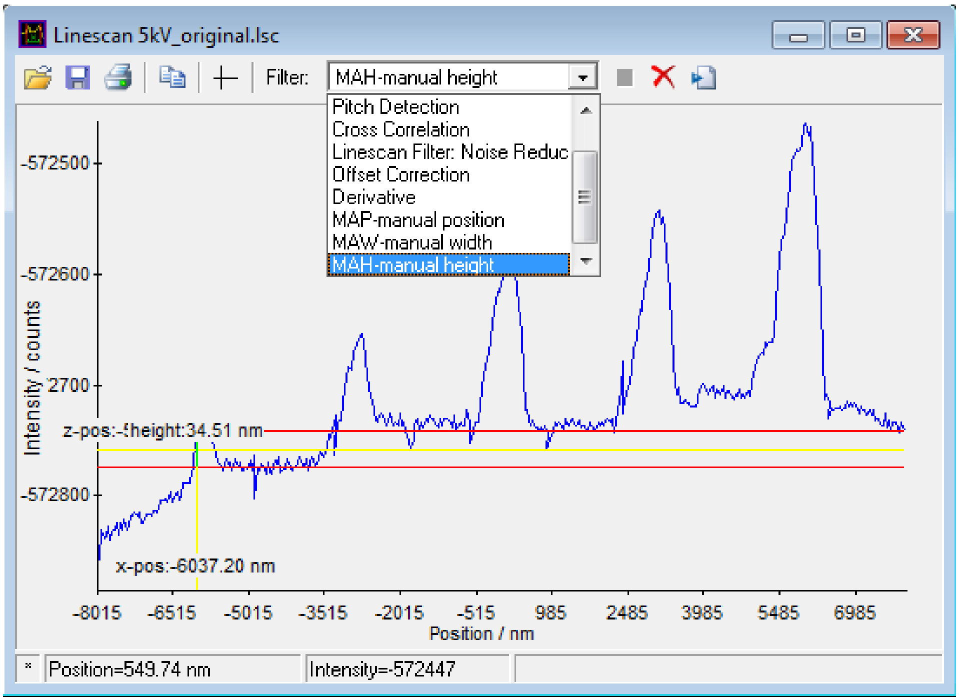


Figure 22-26: Manually measuring the height differences at locations along the Linescan.

- The result of these height measurements can be saved in the Positionlist by clicking on the button **Store Filter Results to Positionlist** . The results will then be displayed the columns **Pos/um** in the Positionlist.

STEP 4: Working with the Linescan filter Noise reduction

- The Noise Reduction filter is one of the Linescan filters which can be applied to the NanoLinescan. Choose this filter to perform numerical averaging after recording the data. Two parameters define the averaging. **Number of average points** is used to define the number of points along the scan direction over which the average is calculated. **Number of iterations** defines how many times the averaging is done.

STEP 5: Working with the Linescan filter Offset correction

- The Offset correction filter is only applicable for the NanoLinescan. An offset subtraction can be applied. For example, it will take the first and the last data point of a sloping surface of a NanoLinescan and will set a straight line between those points, this offset can then be substracted.

C Software Reference

This section describes all functions and control elements of the **Nanomanipulators** module dialogs and subdialogs.

1 Nanomanipulator dialog

You call up the dialog via control bar .

The Nanomanipulator dialog serves to control, adjust, display and configure one or all installed nanomanipulators.

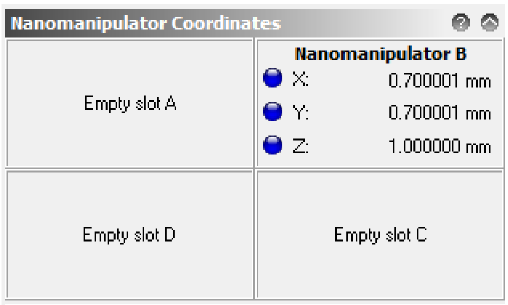


Figure 22-27: **Nanomanipulator** dialog – Example configuration with two nanomanipulators

The dialog consists of

- a tool bar with functions for similar types of control of all nanomanipulators (⇒ *Nanomanipulator dialog Tool bar* on page 22-27),
- four parameter fields to control every single installed nanomanipulator (⇒ *Nanomanipulator/NanoSense dialog Parameter Field* on page 22-29).

The arrangement of the parameter fields corresponds to the arrangement of the nanomanipulators in the chamber and the image.

1.1 Nanomanipulator dialog Tool bar

The tool bar contains the top level control functions. All commands have the same effect on all installed nanomanipulators.

The tools have the following functions:

Figure 22-28: **Nanomanipulator** dialog – Tool bar

Button	Function
	Opens the **Nanomanipulator Configuration** dialog in which the configuration of every installed nanomanipulator can be viewed and edited (⇒ *Nanomanipulator Configuration dialog* on page 32). **Note:** The dialog is only attainable for the local system experts.
	Opens the **Adjust Nanomanipulator coordinates** dialog, which leads you step by step through the adjustment procedure for the nanomanipulator coordinates (⇒ *Nanomanipulator Coordinates dialog* on page 22-48). **Note:** The adjustment procedure must be carried out after every tip change or in case of misalignment.
	Moves all nanomanipulators to their parking positions.
	Moves all nanomanipulators into the field of view.
	Opens the **Nanomanipulator visualization** dialog to monitor the nanomanipulator positions graphically (⇒ *Nanomanipulator visualization dialog* on page 37).
	Opens the **Nanomanipulator: Approach Tools** dialog to set down the tip on the sample exactly (⇒ *Nanomanipulator: Approach Tools dialog* on page 39).

1.2 Nanomanipulator/NanoSense dialog Parameter Field

A parameter field for one nanomanipulator contains functions and status displays, which relate only to the corresponding nanomanipulator. The parameter field changes depending if a nanomanipulator (⇒ *Nanomanipulator dialog Parameter Field* on page 22-29) or if the Nano-Sense option is installed, (⇒ *NanoSense dialog Parameter Field* on page 22-31). The **Status** LED indication is the same for both, as shown below.

Button	Function
Status LED	Indicates the actual position and status of the selected nanomanipulator:
	Gray: Nanomanipulator is initialized and idle.
	Blue: Nanomanipulator is in its parking position.
	Green: Nanomanipulator is moving
	Yellow: Nanomanipulator is in the field of view.
	Red: Nanomanipulator is not initialized or all axes are locked.

1.2.1 Nanomanipulator dialog Parameter Field

The parameter field for one nanomanipulator contains the functions described below, related to the corresponding nanomanipulator. In addition to clicking on an icon to initiate it, the icons can also be initiated via the positionlist (⇒ *Positionlist* on page 9-1). Using the drag & drop functionality using the left mouse button, the icons can be dragged and dropped into a positionlist, from where the functions that correspond to the icon can be initiated.

If the NanoSense is installed, there is an additional function available for the nanomanipulators. This function is called Follow Scout to surface.

As the NanoSense is a piezo-driven sensor, the piezo simulation is altered when the tip approaches a surface. This can be used to measure Z values accurately. Other nanomanipulators can then be automatically driven to this previously measured Z value.

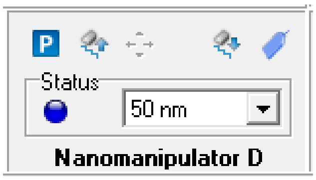


Figure 22-29: **Nanomanipulator** dialog – Parameter field

The control elements have the following functions:

Button	Function
	Moves the selected nanomanipulator to its parking position.
	Lift arm safely: lifts the nanomanipulator arm safely from the sample surface by 50µm.
	Activates the keypad.
	Follow scout to surface: drives the corresponding nanomanipulator to the desired position with a new Z value, previously measured by the NanoSense. The NanoSense has a sensor at its tip, which can sense the approach to a surface. These measured Z values can then be used to automatically drive another nanomanipulator safely to the surface, as the Z value was previously measured by the NanoSense. Different Z positions are measured by the NanoSense and stored, then used to drive a nanomanipulator safely to that position. Note: This icon can be dragged into the image, to define u,v position of the action.
	Moves the selected nanomanipulator into the field of view.
	To specify the step size with which the nanomanipulator will be moved via the keypad.
Nanomanipulator #	Indicates the name of the nanomanipulator.

1.2.2 NanoSense dialog Parameter Field

A parameter field for the NanoSense contains additional functions.

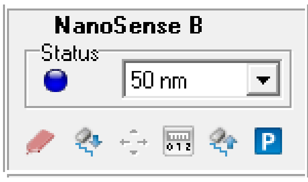


Figure 22-30: **Nanomanipulator** dialog – Parameter field

The control elements have the following functions:

Button	Function
	Moves the nanomanipulator B (NanoSense) into the field of view. The NanoSense will be positioned 1 mm above the sample surface.
	Use NanoSense as surface scout. The NanoSense will now define the Z values of the locations to which the nanomanipulator will drive. The Z values are now measured by the NanoSense. These can then be used to drive a nanomanipulator automatically and safely to that position. Note: This icon can be dragged into the image, after which the Z position will be measured at this location.
	Activates the keypad.
	Use Follow Scout. The NanoSense is moved to the surface, the Z value is measured and then the tip is moved up into a safe position.
	Lift arm safely. This will lift the NanoSense arm safely from the sample surface by 50 µm.
	Moves the selected nanomanipulator to its parking position.
	To specify the step size with which the nanomanipulator will be moved via the keypad.
Nanomanipulator #	Indicates the name of the nanomanipulator.

1.3 Nanomanipulator Configuration dialog

The dialog is only available to a local system expert.

The dialog serves to modify the nanomanipulators‘ configuration.

You call up the subdialog via **Nanomanipulator** dialog .

The functions in the **Nanomanipulator Configuration** dialog are divided into two subsets:

- General settings which affect all of the nanomanipulators (⇒ *Nanomanipulator Configuration dialog general settings* on page 22-33).
- Settings which only affect one selected nanomanipulator (⇒ *Nanomanipulator Configuration dialog – nanomanipulator specific settings* on page 22-34).

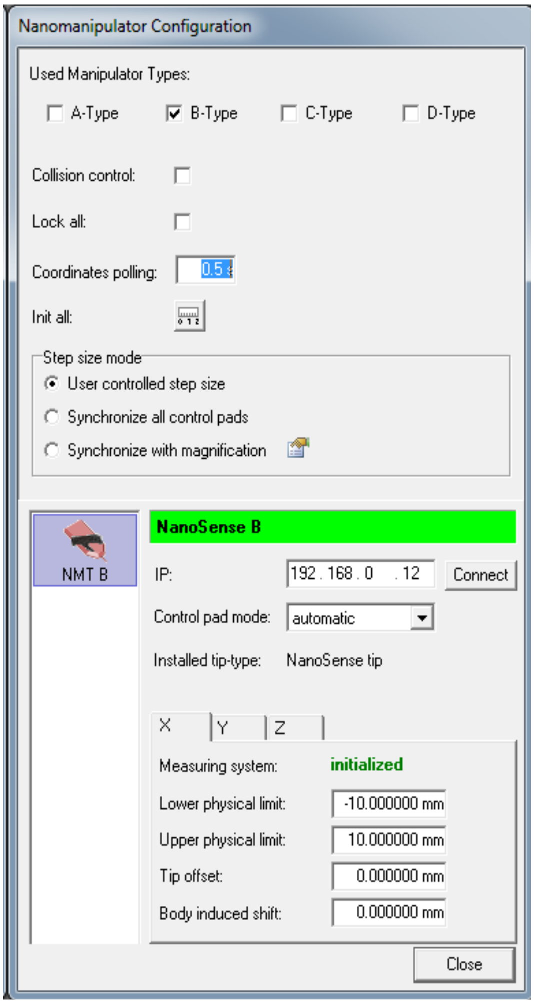


Figure 22-31: **Nanomanipulator Configuration** dialog

1.3.1 Nanomanipulator Configuration dialog general settings

The control elements apply to all nanomanipulators and have the following functions:

Control element	Function
Used Manipulator Types	This uses the stored geometry details to determine the correlation between the nanomanipulators in the chamber and the associated numbers in the NANOSUITE software.

Control element	Function
Collision control	Switches the collision control on or off. **Before each movement of the nanomanipulators, the control will check if there would be a collision with another nanomanipulator or the GIS. The collision control is based on a 2-dimensional model, which is equivalent to the view through the roof of the chamber.** How to work with the collision control refer to (⇒ *Working with the Collision Control* on page 18).
Lock all	This locks all axes of the nanomanipulators.
Coordinates polling	This sets the polling time, which is the interval between successive requests for the position of the nanomanipulator (default is 2 s).
Init all	This initializes all nanomanipulators.
Step size mode	Subset for changing the stepsize of the keypad:
User controlled step size	This defines a stepsize specific to each nanomanipulator by selecting a value from the corresponding **Status** combo box in the **Nanomanipulator** dialog (⇒ *Nanomanipulator/NanoSense dialog Parameter Field* on page 29)
Synchronize all control pads	This defines the stepsize for all nanomanipulators via the selection of a value from the corresponding **Status** combobox in the **Nanomanipulator** dialog (⇒ *Nanomanipulator/NanoSense dialog Parameter Field* on page 29)
Synchronize with magnification	The stepsize is dependent on the image magnification. The corresponding values can be defined with click on  (⇒ *Mag. <-> Control pad subdialog* on page 36).

1.3.2 Nanomanipulator Configuration dialog – nanomanipulator specific settings

The control elements relate to a selected nanomanipulator and have the following functions:

Control element	Function
Header	This shows the name and status of the selected nanomanipulator. If the status is red there is no connection between the nanomanipulator and the NANOSUITE software. If the status is green there is a connection between the nanomanipulator and the NANOSUITE software.
IP	This shows the IP address of the network controller for the selected nanomanipulator.
Connect	This checks the connection between the network controller and the NANOSUITE software.
Control pad mode	This selects the status of the keypad for the selected nanomanipulator (default is **automatic**).
automatic	The keypad can only be used when the nanomanipulator is in the field of view.
active	The keypad can always be used, even if the nanomanipulator is parked.
inactive	The keypad has been disabled.
Installed tip-type	This indicates the actual used tip-type.
XYZ tabs	The settings in the tabs relate to the three axes of motion accordingly.
Measuring system:	This shows whether the measuring system for the selected nanomanipulator has been initialized. **Note:** The nanomanipulator can only be used if it has been initialized.
Lower physical limit:	This shows the lower limit for the software motion commands on the selected nanomanipulator. **Note:** You should not change this value manually. The value is set during the initial commissioning. It corresponds to the lower limit of the displayed field size, which is shown in the **Nanomanipulator visualization** dialog (⇒ *Nanomanipulator visualization dialog* on page 37).

Control element	Function
Upper physical limit:	This shows the upper limit for motion commands on the selected nanomanipulator. **Note:** You should not change this value manually! The value is set during the initial commissioning. It corresponds to the lower limit of the displayed field size, which is shown in the **Nanomanipulator visualization** dialog (⇒ *Nanomanipulator visualization dialog* on page 37).
Tip offset W-Type:	This shows the offset between the nominal target position and the current, measured position of the wire-shaped tip type. **Note:** You should not change this value. This value applies specifically to the installed nanomanipulator and is adjusted during the adjustment procedure.
Tip offset UD-Type:	This shows the offset between the nominal target position and the current, measured position of the Ultra dense probing tip type. **Note:** You should not change this value. This value applies specifically to the installed nanomanipulator and is adjusted during the adjustment procedure.
Body induced shift	This shows the user specified offset between the nominal target position and the current position. Note: The user can change this body induced shift and use it if noticable shift due to charging is visible and needs to be corrected.

1.4 Mag. <-> Control pad subdialog

The dialog is only available to local system experts.

The **Mag. <-> Control pad** subdialog serves to assign an image magnification value to a keypad step. As a consequence, according to the actual magnification the appropriate step size will be set automatically.

You call up the subdialog via **Nanomanipulator Configuration** dialog.

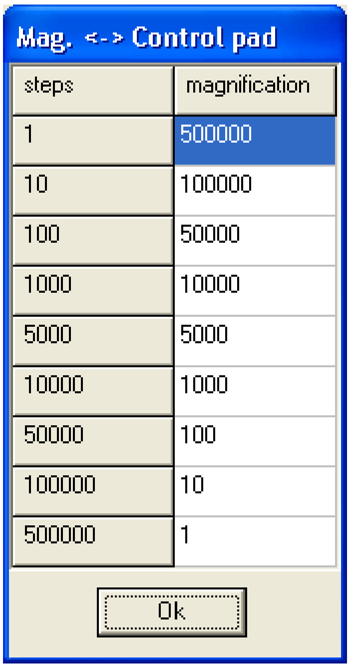


Figure 22-32: **Mag. <-> Control pad** subdialog

In the fields under **magnification**, you can enter the desired magnification value and save the settings with **OK**. The **steps** column displays the pre-defined keypad steps.

1.5 Nanomanipulator visualization dialog

The **Nanomanipulator visualization** dialog shows the current XYZ position and the range of travel of every nanomanipulator graphically. To obtain a first overview, it can be used additionally to the image.

You call up the dialog via **Nanomanipulator** dialog .

The visualization of the nanomanipulators via the NANOSUITE software is based on calculated values from the position measuring system combined with the results of the adjustment procedure.
As the precision of the adjustment is limited, these values can only be used for guidance. They are in no way a substitute for user monitoring via the image.

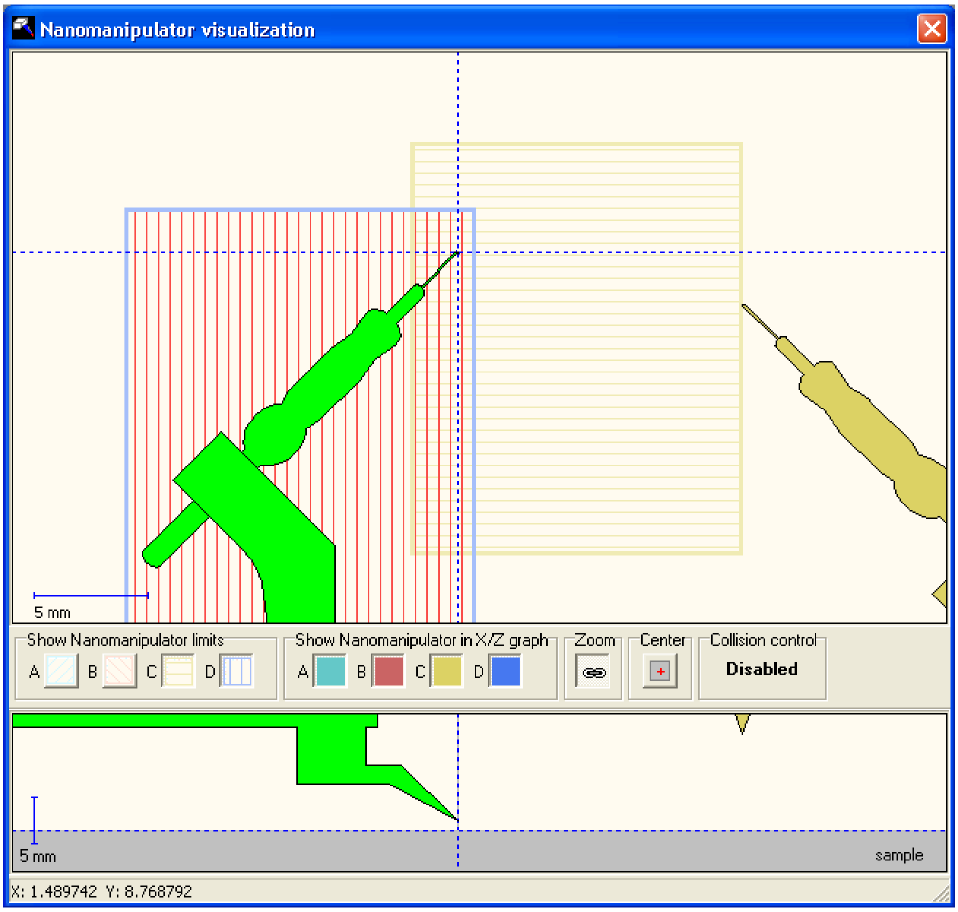


Figure 22-33: **Nanomanipulator visualization** dialog

The dialog is divided into three areas:

- The upper subset shows an XY view which corresponds to the view through the chamber lid. The intersection of the dashed lines (X = 0, Y = 0) corresponds to the center of the image.
- In the center subset, you can modify the display via some optional parameters (⇒ *Nanomanipulator visualization dialog – Center subset* on page 22-39).
- The lower subset shows an XZ view, which allows the distance between the nanomanipulators and the surface of the sample to be estimated. The intersection of the dashed lines (X = 0, Z = 0) corresponds to the center of the image with a working distance of 10 mm. The gray area corresponds to the area where the sample is located.

The display of the sample surface is only valid if the “Adjust W” procedure had previously been done (⇒ *Preparing the approach process (starting position)* on page 22-6).

The display can be shifted in the X and Y directions using the arrow keys.

The magnification can be changed by moving the mouse wheel whilst the cursor is within the view (zoom function). The XY view and the XZ view can be zoomed simultaneously if the **Zoom** button is enabled.

Clicking on a nanomanipulator with the mouse selects a nanomanipulator.

1.5.1 Nanomanipulator visualization dialog – Center subset

The parameters of the center subset are divided into five lower subsets with the following meanings:

Center subset	Function
Show Nanomanipulator limits	Enables and disables the display of the limits to the movement of the nanomanipulators in the X/Y view (hatched areas in (⇒ Figure 22-33 on page 22-38).
Show Nanomanipulator in X/Z graph	Enables and disables the display of the nanomanipulators in the XZ view.
Zoom	Allows the zoom in the XY and XZ views to be coupled.
Center	Centers the view.
Collision control	Shows whether the collision control is activated or not (⇒ *Working with the Collision Control* on page 18).

1.6 Nanomanipulator: Approach Tools dialog

The **Nanomanipulator: Approach Tools** dialog provides the calculated distances between the tip and the sample surface as well as the XYZ position of the piezos for each nanomanipulator.

Call up the dialog via **Nanomanipulator** dialog .

How to use the dialog for an approach process refer to (⇒ *Positioning a nanomanipulator (approach process)* on page 22-5).

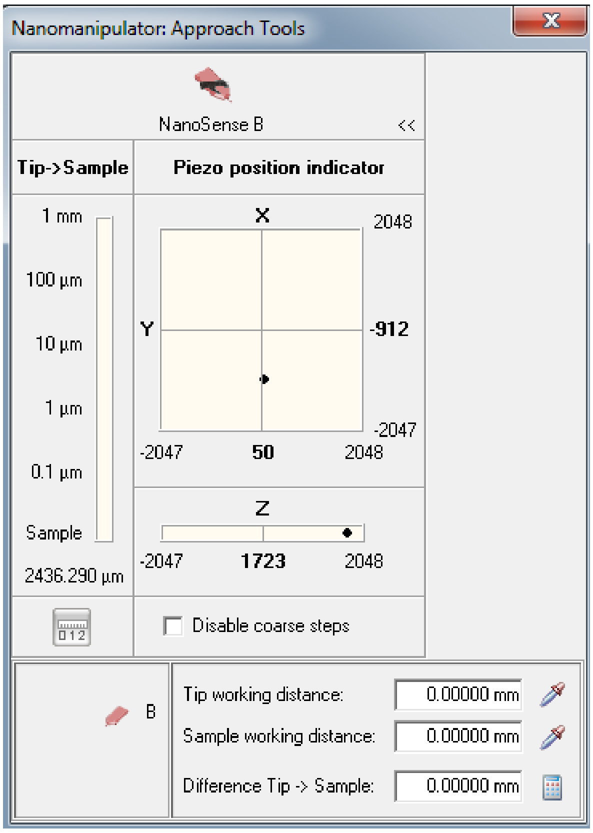


Figure 22-34: **Nanomanipulator: Approach Tools** dialog

The dialog is divided into two main subsets:

- **Nanomanipulator #** subset to display the calculated tip-to-sample distance and corresponding piezo position of the selected nanomanipulator (⇒ *Nanomanipulator: Approach Tools – Nanomanipulator # subset* on page 22-41).
- Calculation subset to calculate the real tip-to-sample distance measured by the electron or ion optics (⇒ *Nanomanipulator: Approach Tools – Calculation subset* on page 22-41).

1.6.1 Nanomanipulator: Approach Tools – Nanomanipulator # subset

The control elements in the both columns have the following functions:

Control element	Function
Nanomanipulator #	Indicates the current selected nanomanipulator by its button and name.
<< and >>	To swing in and out the **Piezo position indicator** column.
Tip->Sample column	Shows the current tip-to-sample distance graphically via the fill level of a column in the range of 1 mm to 0 (**Sample**) and digitally. **Note:** The scale is logarithmic.
	Overwrites the W-adjustment with the calculated value.
Piezo position indicator column	Indicates the current expansion tension of the nanomanipulator piezos in XYZ direction in bits. In this range, the piezo movement will be indicated smoothly in small steps. One step is about 1 to 2 nm. The cross indicates the center of the expansion range. The physical expansion of the piezos is limited to approximately 2.5 µm. If this expansion is reached – i.e. the indicator is at a border of the **Piezo position indicator** column – the corresponding piezo will jump on its axis and this movement will be indicated in more coarse steps. That means, the indicator in the column can e.g. jump from the borders directly to the center.
Disable coarse steps	To deactivate the coarse step mode – i.e. the piezo movement will be indicated always in small steps.

1.6.2 Nanomanipulator: Approach Tools – Calculation subset

The control elements have the following functions for each nanomanipulator:

Control element	Function
Tip working distance	To read the real tip working distance measured by the electron or ion optics with click on .
Sample working distance	To read the real sample working distance measured by the electron or ion optics with click on .

Control element	Function
Difference Tip->Sample	To calculate the real tip-sample distance with click on .

2 Nanomanipulator Control dialog

The **Nanomanipulator Control** dialog serves to move a selected nanomanipulator in XYZ with absolute or relative values or with a defined step size.

Call up the dialog via control bar .

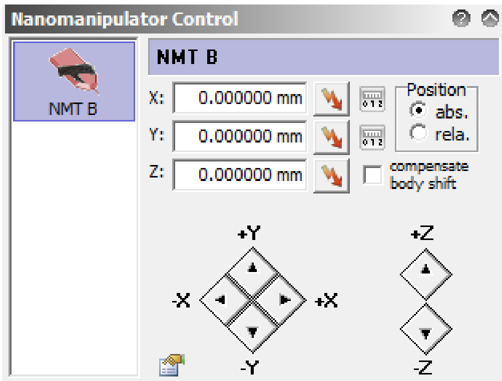


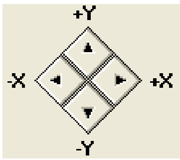
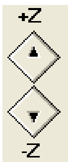
Figure 22-35: **Nanomanipulator Control** dialog

The dialog is divided into two subsets:

- Selection bar on the left hand side to display and select a nanomanipulator,
- **NMT #** subset to move the selected nanomanipulator.

The control elements in the **NMT #** subset have the following functions:

Control element	Function
X, Y, Z input fields	To enter a value for the target coordinates on the X, Y and Z axis.
	Moves the nanomanipulator on the corresponding axis to the defined position.
	Origin correction of X by value. The absolute X coordinates of the nanomanipulators or the NanoSense are set to the displayed value in the X text field.

Control element	Function
	Origin correction of Y by value. The absolute Y coordinates of the nanomanipulators or the NanoSense are set to the displayed value in the Y text field.
Position - absolute	To select the absolute motion mode: The nanomanipulator will be moved to the absolute coordinates defined in the **XYZ** input fields.
Position - relative	To select the relative motion mode: The nanomanipulator will be moved relative to its current position.
	Digital pads to move a nanomanipulator stepwise on its XY axis. The stepsize is defined in the **Stepsize** input field.
	Digital pads to move a nanomanipulator stepwise on its Z axis. The stepsize is defined in the **Stepsize** input field.
Compensate body shift	To compensate the body shift.
Edit Stepsize setting	Shows XYZ coordinates field where the stepsize can be entered.
Stepsize	To define the stepsize for the digital pads in mm.
Equal steps	To define an identic step size for the X and Y axes. If the checkbox is activated, the defined stepsize for one axis will be taken over to the other.

3 NanoWorkbench Control (optional)

NanoWorkbench Control is only available with the NanoSense option. It enables additional functionalities for the NanoSense, via an external interface.

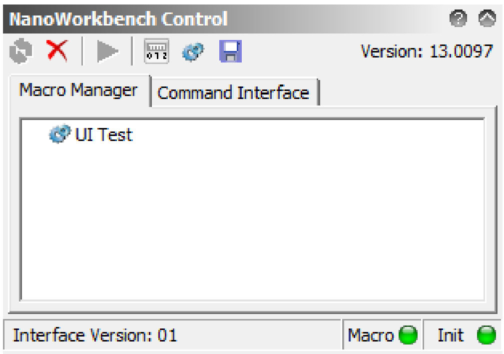


Figure 22-36: **NanoWorkbench** Control

The control elements in the **NanoWorkbench Control** subset have the following functions:

Control element	Function
	Synchronise with NCC (Nano Control Client). This will activate the functionality controls.
	Cancels the selected functionality.
	Activate the marked macros.
	Calibrate the sensor of the NanoSense tip for the signal using the Nanofinger calibration routine.
	Open the GUI interface of NCC. This opens an external program with extended functionalities.
	To save the results. This will save the results calculated using the macros.
Macro Manager tab	The Macro manager lists all available macros. They can be activated via Start Macro.
Command interface tab	Lists available macros, of which a command line can be selected.

3.1 Nanofinger calibration

When clicking on the 4th icon  of the NanoWorkbench Control, the Nanofinger calibration window opens. This allows viewing and calibration of the NanoSense working area prior to using the NanoSense. In this window. the NanoSense tip is not lowered to a sample surface. Instead, the NanoSense tip will remain stationary and different frequencies are applied to find the frequency at which the NanoSense tip is the most sensitive.

The NanoSense tip vibrates continuously and when the tip approaches the sample surface, the amplitude of vibration is decreased, or dampened by contact with the sample surface. When the amplitude of vibration of the tip is reduced, the signal will be reduced. This is the signal which is measured when using the NanoSense. In order to get the best sensitivity when measuring Z values, the NanoSense tip should be set to the highest sensitivity during the Nanofinger calibration described here.

The Nanofinger calibration should be performed immediately after the installation of the NanoSense tip. The aim is find the frequency at which the amplitude of vibration of the NanoSense tip is the largest, in order to have the greatest sensitivity when measuring the Z values. This will be the most sensitive frequency for that individual NanoSense tip.

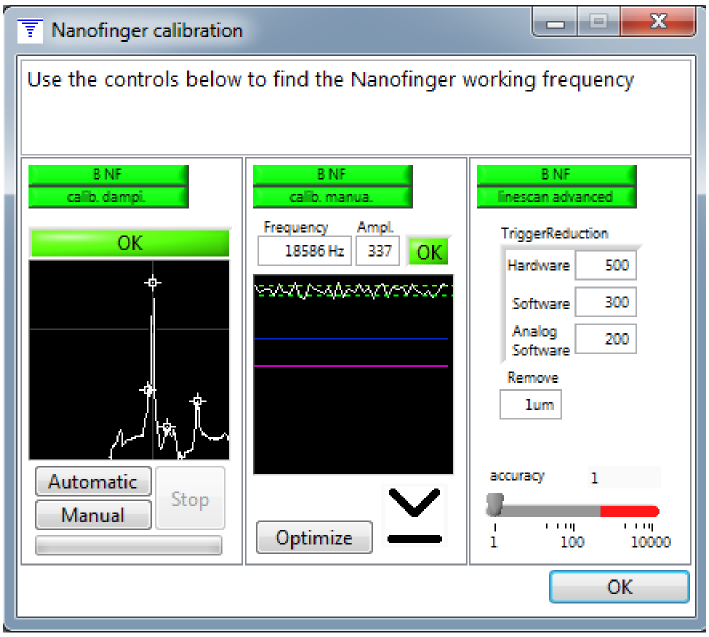


Figure 22-37: Nanofinger calibration

This window consists of three panes, **Scan**, **Signal** and **Editing** panes. The control elements in the **Nanofinger Calibration** subset have the following functions:

Control element	Function
Left pane	Scan window
BNF/Calib button	To set advanced parameters, only to be used by engineers.
OK button	Status indication: Indicates if the communication is active. (B = position of the sensor; F = Nanofinger (same as NanoSense).
Graph	Calibration damping, shows the working frequency of the sensor. Firstly, a scan is performed manually, followed by the display of the frequency scan. This controls the frequency of the sensor, with which an object is swept.
Automatic/ Manual button	To toggle between Automatic and Manual. When Automatic is selected, two different frequencies can be compared. A damped scan and an undamped frequency scan are displayed. Whichever looks best is then selected. Green status bar - the signal is in a pre-defined value range.
Stop button	To abort
Middle panel	Signal
BNF/Calib button	BNF/Calib will enable a change of the mode of this tile, do not change!
Frequency/ Amplifier, OK button	Displays the frequency and amplifier status (OK).

Control element	Function
Graph	This shows the actual signal of the sensor, showing how damped the signal is. The frequency is shown on the X-axis, whereas the Y-axis shows the output of the login amplifier. The dotted line shows the undamped signal of the tip. The band between the blue and red lines is the band where the working damping is in linescan mode. The damping is regulated between these two lines using the linescan, which means slight contact, but not full contact. The frequency and the damping value are indicated and are regulated automatically when a peak has been selected in the left hand pane. The frequency shown here is the new working frequency. This is the mechanical stimulation of the piezo of the sensor. If the signal is very low, a higher stimulation is required. If the signal is very high, a lower stimulation is required.
Optimize/ Arrow button	To perform the optimization.
Right pane	Editing parameters
BNF/Calib button	BNF/Calib will enable a change of the mode of this tile, do not change!
Trigger Reduction	To set the trigger reduction for Hardware, Software and Analog software. To calibrate the sensor of the NanoSense tip for the signal using the Nanofinger calibration routine.
Reduce	To reduce in um steps.
Accuracy	To define the accuracy.
Scale bar	To scale the results.

4 Nanomanipulator Coordinates dialog

In the **Nanomanipulator Coordinates** dialog, the current XYZ positions and status of the installed nanomanipulators can be monitored.

Call up the dialog via control bar .

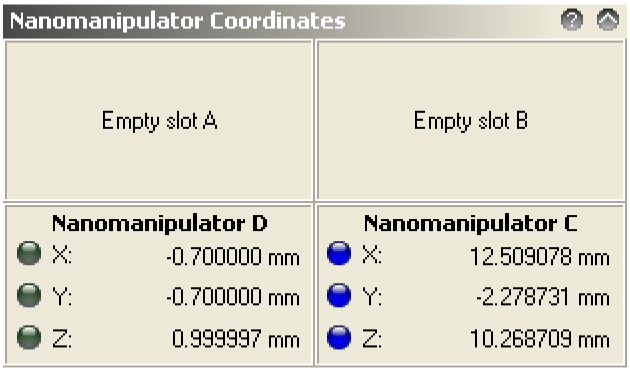


Figure 22-38: **Nanomanipulator Coordinates** dialog

The LEDs can have the following status for each axis:

LED color	Status
	Gray: Nanomanipulator is initialized and idle.
	Blue: Axis of nanomanipulator is in its parking position.
	Green: Axis of nanomanipulator is moving.
	Yellow: Axis of nanomanipulator is in the field of view.
	Red: Axis of nanomanipulator is locked. **Note**: If the axis is not initialized, **Not initialized** will be displayed instead of the coordinate value and the LED is red.

Chapter 23
RF Plasma Cleaning (option)

This chapter describes how to plasma clean the chamber interior surfaces from hydrocarbon contamination via the **Plasma Control Evactron** software module in combination with an Evactron De-Contaminator tool (EDC).

The **Plasma Control Evactron** module is an optional module of the Raith NANOSUITE software and only available, if an Evactron De-Contaminator tool (EDC) is installed to the chamber.

This chapter contains the following topics and tasks:

- Brief description of the *EDC*
(⇒ *Hardware* on page 23-2),
- Reasons for plasma cleaning the chamber and specimen
(⇒ *Plasma Control Evactron software module* on page 23-3),
- Principle and procedure of RF plasma cleaning with the EDC
(⇒ *RF plasma cleaning with the EDC* on page 23-4),
- Performance of the plasma cleaning procedure
(⇒ *Performing the plasma cleaning procedure* on page 23-6),
- Description of all functions and control elements of the **Plasma Control Evactron** module (⇒ *Software Reference* on page 23-9).

A Functional Description

This chapter gives an overview of the basic functions of the EDC and the RF plasma cleaning process.

1 Hardware

The EDC is an optional component for the Raith Lithography Systems RAITH150 Two and eLINE.

The EDC consists mainly of two components:

- Oxygen Radical Source (ORS), which is installed to the chamber. The ORS contains the plasma and generates the active species for cleaning.
- Controller unit, which is connected to the ORS and the PC. The controller unit delivers the RF power to build up the plasma.

(⇒ Figure 23-1 on page 23-2) shows the main components of the ORS:

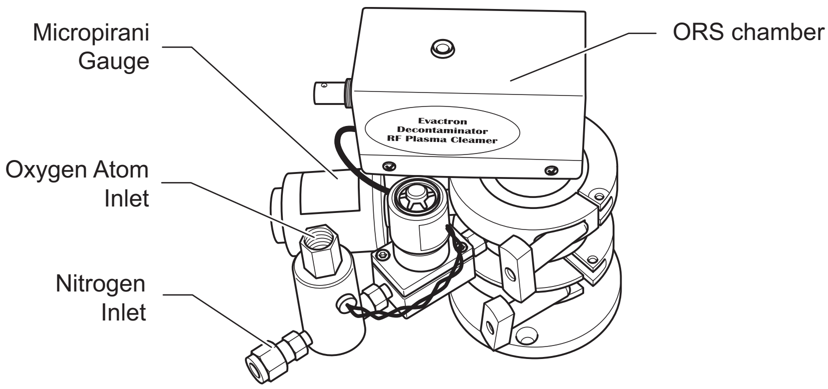


Figure 23-1: Main components of the Oxygen Radical Source (ORS)

2 Plasma Control Evactron software module

The control and operation of the EDC is carried out by the **Plasma Control Evactron** dialog. The dialog consists of three tabs and a command bar to control and monitor the plasma cleaning procedure.

(⇒ Figure 23-2 on page 23-3) shows the **Plasma Control Evactron** dialog:

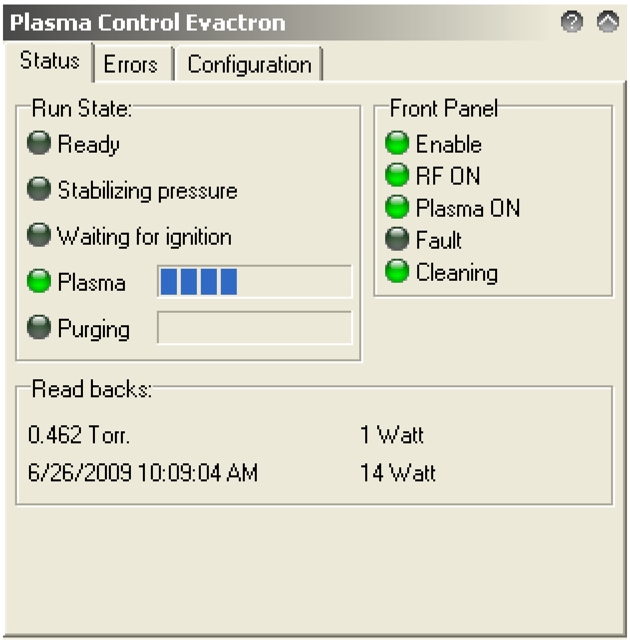


Figure 23-2: **Plasma Control Evactron** dialog

For a detailed description of all control elements of the dialog refer to (⇒ *Plasma Control Evactron dialog* on page 23-9).

3 RF plasma cleaning with the EDC

The following sections will explain the RF plasma cleaning procedure.

3.1 Principle of RF plasma cleaning

The cleaning principle of the EDC is based on chemical etch cleaning. In the ORS, oxygen atoms are created from air via an RF generated, low power plasma. The ORS generates the oxygen atoms as active species which flow into the chamber and react with the contaminants. As a result, volatile compounds like CO, CO_2 and water vapor are produced which can be removed by the vacuum pumps.

The surfaces are not subjected directly to plasma exposure.

After plasma cleaning is completed, it is often useful, to purge the chamber with nitrogen (N_2) to remove the remaining product gases and air. This will result in a faster pump down and a cleaner vacuum.

3.2 Plasma cleaning procedure

The plasma cleaning procedure consists of a sequence of states, which are performed automatically by the lithography system and the EDC. Once the plasma cleaning procedure is started, the following states will be carried out:

1. The EDC will be turned on and the lithography system will be put into its cleaning mode: The EHT will be switched off, the CCV will be closed and the chamber will be vented above a threshold of 2.0 torr.
2. The chamber will be evacuated to a pressure of 0.4 torr.
3. The RF power will be switched on and the plasma will be ignited at a pressure of 0.5 torr.
4. The cleaning state will be carried out: The ORS generates oxygen atoms and lets them flow into the chamber. During this state, the EDC regulates the chamber pressure continuously in order to maintain proper vacuum conditions. The RF power remains ON during this sequence
5. If enabled, the purging state will be carried out after the cleaning state is finished. N_2 will be let into the chamber. The RF power is switched off.
6. If the cleaning and purging states are finished, the chamber will be completely evacuated, the EDC will be switched off and the lithography system will be put back into its operating mode.

You can repeat one plasma cleaning procedure multiple times.

3.3 Settings and parameters

The right settings for an effective plasma cleaning procedure are highly dependent on the degree of contamination of the chamber and the specimen currently loaded, as well as the material of the specimen. Certain kinds of materials should not be used with the plasma cleaner, i.e. SiO_2 wafers.

To find the right settings for critical specimens, the user will need to experiment with the different parameters.

Control the plasma cleaning procedure via the following parameters:

- Set-points for the pressure:
  - at which the plasma will be initiated,
  - under which the cleaning state is to be run,
  - under which the purging state is to be run.
- Set-point for the forward RF power under which the cleaning state is to be run.
- Duration time, how long the cleaning and purging states are to be run.
- Number, how often the plasma cleaning procedure is to be repeated.

For a detailed description of the parameters refer to section (⇒ *Errors tab* on page 23-13).

B Tasks

This chapter describes how to perform a plasma cleaning procedure.

1 Performing the plasma cleaning procedure

STEP 1: Establishing settings

▶ Change to the **Configuration** tab:

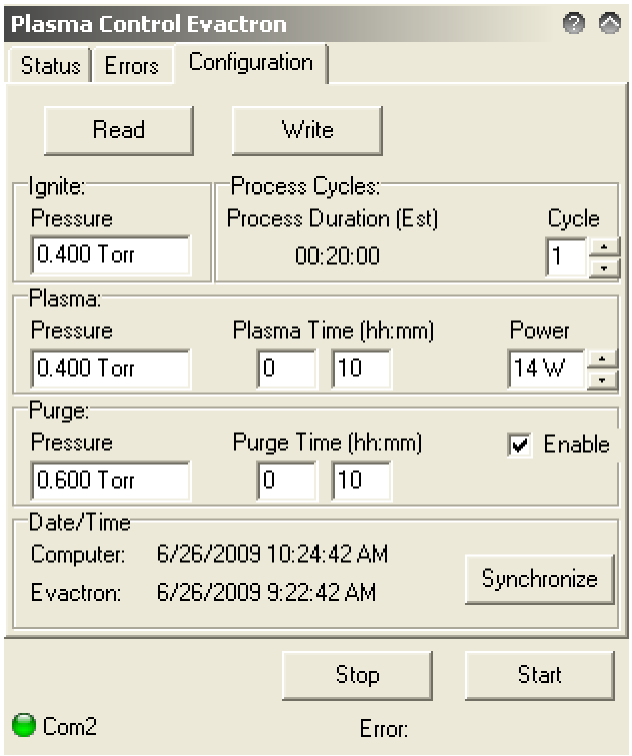


Figure 23-3: **Plasma Control Evactron** dialog – **Configuration** tab

▶ Start with the following default values for the plasma ignition, cleaning and purging state:

Subset	Parameter	Value
Ignite:	**Pressure**	0.400 torr
Plasma:	**Pressure**	0.400 torr
	Plasma Time	10 min
	Power	12 W
Purge:	**Pressure**	0.400 torr

- Choose in the list **Cycle**, how often the plasma cleaning procedure has to be repeated. Start with **Cycle** = **1**.
- Click on **Synchronize**, if you want to synchronize Date and Time of the PC and the EDC.
- Activate the checkbox **Enable**, if you want to carry out the purging state after the cleaning state.
- Click on **Write** to save the current settings.

Your settings will be saved and the plasma cleaning procedure will be carried out with these settings.

Alternatively, click on **Read**, if you want to perform the procedure with the most recently saved settings.

STEP 2: Starting the cleaning process

NOTICE
Risk of material damage for critical specimen!
Unload specimen which are not suitable for the plasma cleaning process.

- Change to the **Status** tab.
- Click on **Start**.

The system will prompt you to start the cleaning process:

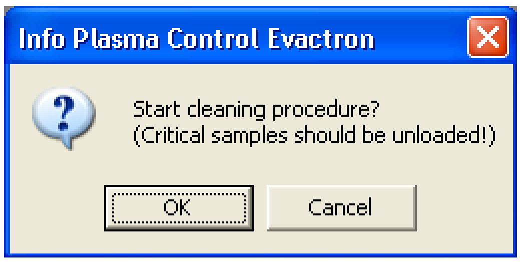


Figure 23-4: **Plasma Control Evactron** dialog – **Status** tab

- Remove a critical sample if necessary and click on **OK**.

The EDC will be turned on and the system will be automatically put into its cleaning mode. The plasma cleaning procedure will automatically be started. The LEDs in the **Status** tab will indicate green the state currently running and the corresponding operating status of the EDC:

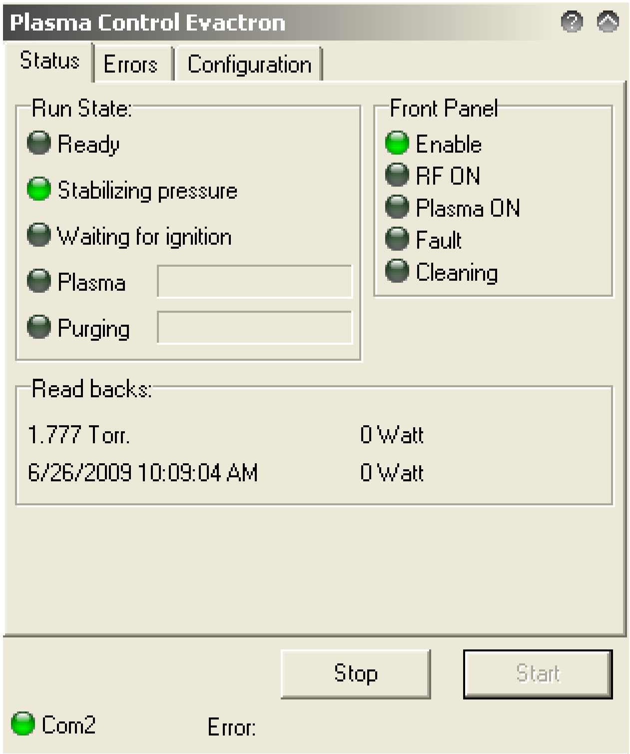


Figure 23-5: **Plasma Control Evactron** dialog – **Status** tab

During the **Stabilizing pressure** state, the system will be evacuated to a pressure of 0.4 Torr. This process takes about 10 minutes.

In the **Read backs** subset, you can monitor the development of the chamber pressure as well as the forward and reverse RF power between ORS and the controller unit.

If the plasma cleaning procedure is finished, the EDC will be turned off and the lithography system will be put into its operating mode. The system is ready for operation.

With click on **Stop** you can finish the cleaning and purging process at any time prematurely. The EDC will be turned off and the system will be put into its operating mode.

C Software Reference

This section describes all functions and control elements of the **Plasma Control Evactron** software module.

1 Plasma Control Evactron dialog

The **Plasma Control Evactron** dialog serves to control the EDC and perform a plasma cleaning procedure.

Call up the dialog by clicking on the icon in the control bar.
The **Plasma Control Evactron** dialog will be displayed right of the control bar. On the control bar **Service** will be displayed.

How to perform a plasma cleaning procedure refer to (⇒ *Performing the plasma cleaning procedure* on page 23-6).

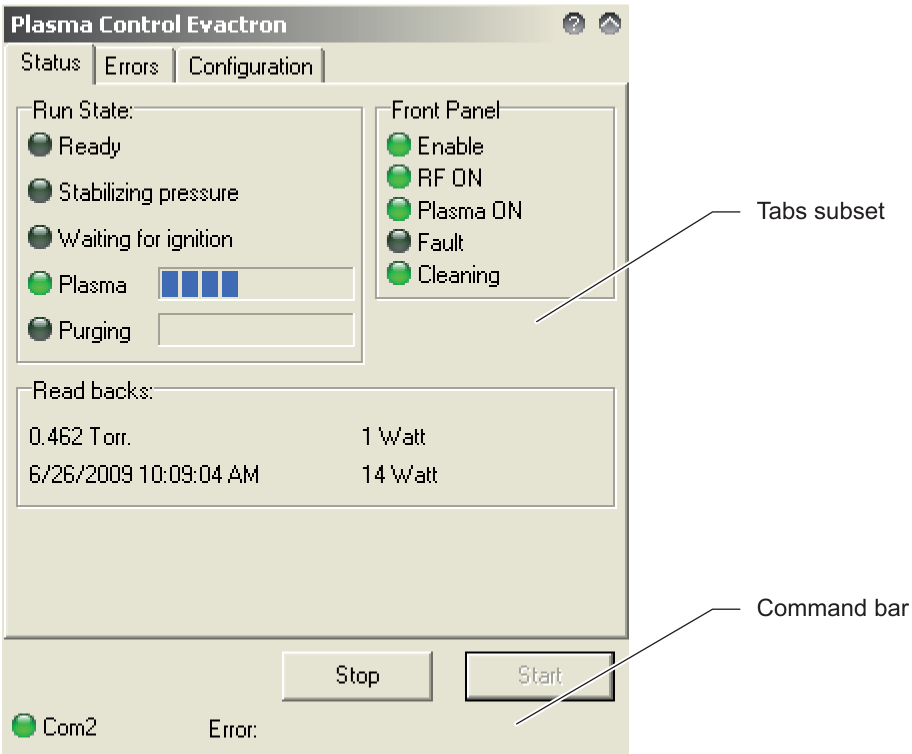


Figure 23-6: **Plasma Control Evactron** dialog

The dialog consists of two subsets:

- Tabs subset with the
  - **Status** tab for the monitoring of the plasma cleaning procedure and the EDC (⇒ *Status tab* on page 23-10),
  - **Errors** tab for indicating faults reported by the EDC (⇒ *Errors tab* on page 23-13),
  - **Configuration** tab for reading and editing the parameters used to carry out the plasma cleaning procedure (⇒ *Errors tab* on page 23-13).
- Command bar subset to start and finish the cleaning procedure (⇒ *Command bar subset* on page 23-17).

Throughout the dialog, LEDs are used to indicate the operating status of the EDC and the lithography system, as well as the current state of the plasma cleaning procedure. The LEDs can have the following status:

Status LED	Function
	Gray: Indicates that the corresponding function or state is disabled.
	Green: Indicates that the corresponding function or state is active.
	Red: Indicates a fault reported by the EDC.

The Plasma Control Evactron dialog offers a bubble-help function, which displays a brief description of all functions and control elements. The help will be displayed if you move the cursor above a field of the dialog.

1.1 Status tab

In the **Status** tab, you can monitor progress and the conditions of the plasma cleaning procedure as well as the operating state of the EDC.

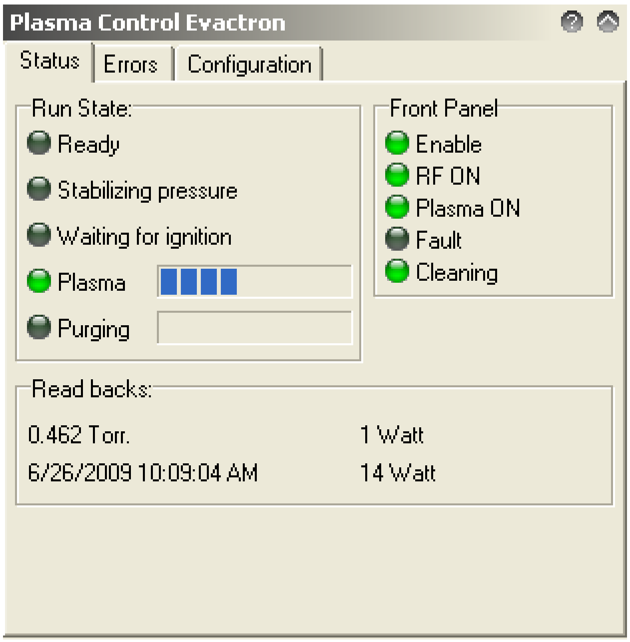


Figure 23-7: **Plasma Control Evactron** dialog – **Status** tab

The **Status** tab is divided into the following subsets:

- **Run State** (⇒ *Run State subset* on page 23-11),
- **Front Panel** (⇒ *Front Panel subset* on page 23-12),
- **Read backs** (⇒ *Read backs subset* on page 23-13).

1.1.1 Run State subset

The **Run State** subset indicates the states which are performed during one plasma cleaning procedure. The currently running state will be indicated by the corresponding LED illuminating or flashing green.

The indicators represent the following states:

Indicator	State
Ready	The lithography system is in the cleaning mode and the EDC is turned on. System and EDC are ready for the cleaning procedure.
Stabilizing pressure	The chamber will be evacuated to a pressure of 0.4 torr. During this process the LED is green flashing. **Note:** If this chamber pressure couldn‘t be reached, the process will be interrupted and an error message will be displayed in the command bar. In this case consult the locally named and Raith trained system expert.

Indicator	State
Waiting for ignition	The RF power will be switched on and the plasma will be ignited.
Plasma 	The plasma is on and the cleaning state is running. During this state the EDC is adjusting the chamber pressure to a proper value for the cleaning. The progress bar indicates the current status.
Purging 	The cleaning state has been finished and the chamber is purged with N_2. During this state the EDC is adjusting the chamber pressure to a proper value for the purging. The progress bar indicates the current status.

1.1.2 Front Panel subset

The **Front Panel** subset indicates the operating status of the EDC. The currently active status will be indicated by the corresponding LED illuminated green. The indicators correspond to the control elements of the controller unit.

The indicators represent the following operating status:

Indicator	Operating status
Enable/Disable	The EDC is turned on and ready for the cleaning procedure. The LED is illuminating, when the EDC is ready for cleaning and green flashing when the cleaning and purging states are running.
RF ON	The RF power is switched on and being sent from the EDC controller unit to the ORS.
PLASMA ON	The plasma in the ORS has been lit.
FAULT	A fault has been detected in the EDC (⟹ *Errors tab* on page 23-13).
Cleaning	The cleaning state is running.

1.1.3 Read backs subset

The **Read backs** subset displays the measured read backs from the EDC.

The indicators have the following meaning:

Indicator	Function
# Torr.	Displays the chamber pressure in Torr, which is measured by the EDC. **Note:** The EDC is equipped with a Micropirani gauge which measures the chamber pressure produced by the backing pump. The pressure is measured near the EDC flange. Therefore, the value indicated here may differ from the value for the system vacuum indicated in the Column Software ➡ **SEM Control** dialog ➡ **Vacuum** tab ➡ **System Vacuum**
# Watt Forward Power	Displays the amount of RF power being delivered from the controller unit to the ORS.
Date and Time	Displays the date and time when the last cleaning cycle was run by the EDC.
# Watt Reverse Power	Displays the amount of RF power being reflected by the ORS back to the controller unit.

1.2 Errors tab

In the **Errors** tab, faults which are reported by the EDC can be monitored.

NOTICE

Risk of system damage by inappropriate troubleshooting.

Measures for troubleshooting and clearing error messages have to be carried out exclusively by the locally named and Raith trained system expert.

The currently reported fault will be indicated by the corresponding LED illuminating red. Additionally, an error message will be displayed in the command bar of the dialog as clear text. Use this information when consulting the system expert.

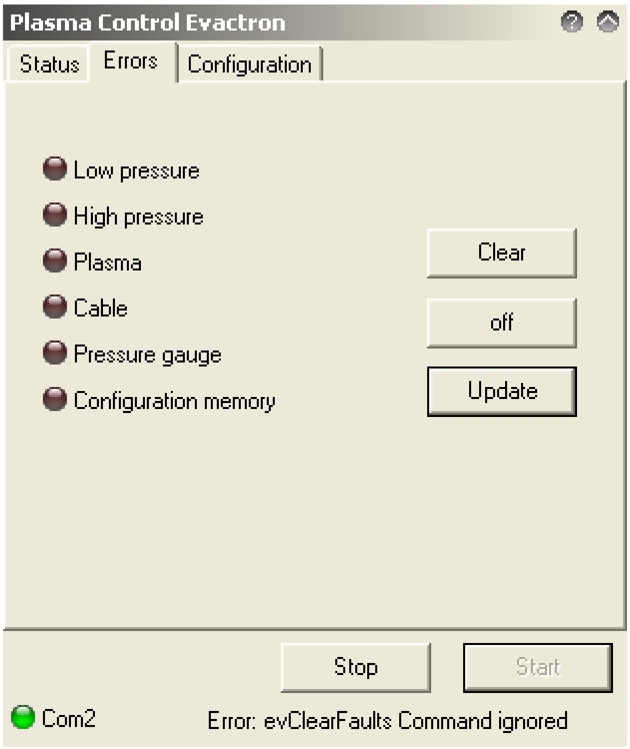


Figure 23-8: **Plasma Control Evactron** dialog - **Errors** tab

The indicators represent the following faults:

Indicator	Fault
Low pressure	Chamber pressure is significantly below the pressure set-point.
High pressure	Chamber pressure is above the high pressure threshold.
Plasma	Plasma is no longer lit.
Cable	Indicates a connection fault.
Pressure gauge	Indicates a connection fault.
Configuration memory	Indicates an EPROM fault.

1.3 Configuration tab

In the **Configuration** tab you can read and edit the parameters used to carry out the plasma cleaning procedure.

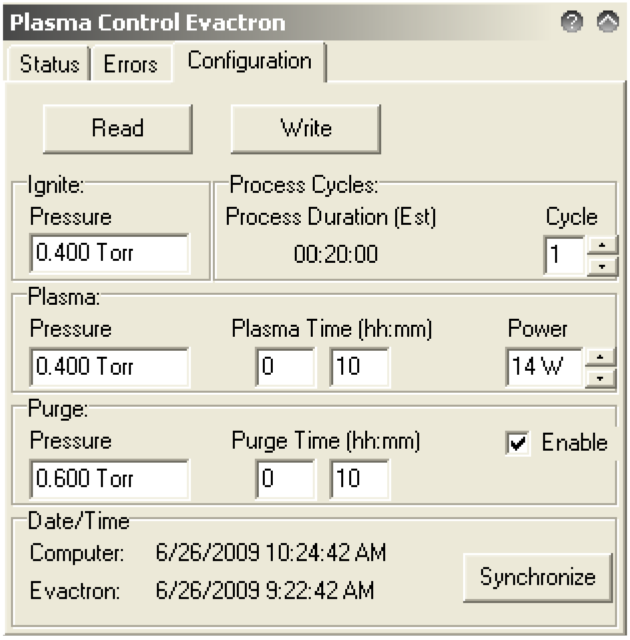


Figure 23-9: **Plasma Control Evactron** dialog - **Configuration** tab

The control elements and parameters have the following functions:

Subset/Button/ Parameter	Function
Read	Calls up the most recently saved settings from the EDC. The settings will be displayed in the input fields and can be used for the actual plasma cleaning procedure.
Write	Saves the currently edited settings. **Note:** The Button is only enabled when the EDC is turned off.
Ignite:	To define settings for ignition:
Pressure	To enter a pressure set-point for the plasma ignition in torr. **Recommendation:** Default value: 0.400 torr
Process Cycles:	To define the repetitions of the whole plasma cleaning procedure:

Subset/Button/ Parameter	Function
Process duration (Est)	Displays the estimated duration of the plasma cleaning procedure with the current settings displayed in the **Configuration** tab.
Cycle	To choose how many times the plasma cleaning procedure has to be repeated.
Plasma:	To define the cleaning state:
Pressure	To enter a pressure set-point by which the cleaning state is to be running in torr. **Recommendation:** Default value: 0.400 torr
Plasma Time (hh:mm)	To enter how long the cleaning state is to be running in hours and minutes.
Power	To enter a power set-point for the duration of the plasma run in W. **Recommendation:** Default value: 14 W
Purge:	To define the purging state:
Pressure	To enter a pressure set-point by which the purging state is to be running in torr. **Recommendation:** Default value: 0.600 torr
Purge Time (hh:mm)	To enter how long the purging state is to be running in hours and minutes.
Enable	To activate the purging state after the cleaning state is finished.
Date/Time:	For the synchronization of date and time.
Computer	Displays the current date and time reported by the PC.
Evactron	Displays the current date and time reported by the EDC.
	To synchronize the date and time information of EDC and PC.

1.4 Command bar subset

The command bar is static and attainable in all tabs. The buttons and indicators have the following function:

Button/Indicator	Function
Start	Turns the EDC on and starts the cleaning procedure. The button is disabled when the plasma cleaning procedure is running.
Stop	Finishes the cleaning process prematurely and turns off the EDC.
Com2	Indicates green the status of the communication port between the PC and the controller unit of the EDC.
Error	Displays a fault that has occurred, in clear text.