Raith

Innovation from a world leader in
e-beam lithography and
semiconductor navigation solutions

Software Operation Manual

e_LiNE

Version 5.0

Product:	*e_LiNE*
Document number:	05-10-9911-404-1.10
Date generated:	Tuesday, October 04, 2005

Statement

All information in this document has been carefully prepared and are believed to be accurate and complete. If doubts exist about any detail or additional information is necessary, please inquire Raith GmbH or their accredited representative.

Copyright

Trademarks

Microsoft, Windows, and Windows 2000	are the registered trademarks of Microsoft, Inc.
AutoCAD, DXF	are the registered trademarks of Autodesk, Inc.
Design CAD	is the registered trademark of American Small Business Computers, Inc.
GEMINI	is the registered trademark of LEO Elektronenmikroskopie GmbH.
GDSII	is the registered trademark of Calma GE.

Addresses

Raith GmbH	Tel.: +49 231 97 50 00 – 0
Hauert 18	Fax.: +49 231 97 50 00 – 5
44227 Dortmund	WWW: *www.raith.com*
Germany	Email: support@raith.com

Raith USA, Inc.	Tel.: +1 631 738 9500
2805 Veterans Hwy	Fax.: +1 631 738 2005
Suite 23	WWW: *www.raith.com*
Ronkonkoma, New York	Email: support@raithusa.com

Structure of software operation manual

The chapters are structured into different tasks, each task consists of several steps. Each chapter has an aim specified at the start of each chapter, and will guide you step-by-step through the process of achieving this aim.

Some of the tasks are optional and are designed to give additional useful information. These additional information sections are clearly marked with a 'Diversion Start' and a 'Diversion End' sign. Experienced users may choose to skip these sections and continue with the next task.


Diversion Start


Diversion End

To set up an exposure, you will need to carry out chapters 1-4 first before performing the exposure in chapter 5. It is important to study the chapters in the given order. Chapter 6 gives an overview of general pattern designs.

There are two computer systems supplied with the RAITH system. The Zeiss computer controls the SEM and is referred to as the column desktop. The Raith computer controls the lithography parameters and is referred to as the lithography desktop.

Getting Started

E-beam Optimization

Stage Adjustment

Write Field Alignment

Exposure

TASK 1: Familiarization with demo pattern
TASK 2: Measuring the beam current
TASK 3: Exposure
TASK 4: Developing the sample
TASK 5: Multiple exposure

General Pattern Design

TASK 1: Creating a design
TASK 2: Pattern design via toolbox
TASK 3: Modifying structures
TASK 4: Measuring a distance
TASK 5: Placing of elements in different layers
TASK 6: Saving deleting and copying of structures
TASK 7: Applying varying dose factors

Advanced Pattern Design

TASK 1: Design using hierarchy
TASK 2: Studying chessy.csf

Mix & Match Exposed

TASK 1: Defining UV positions of marks
TASK 2: Locating the first mark
TASK 3: 3-point adjustment
TASK 4: Semi-automated write field alignment
TASK 5: Automated write field alignment
TASK 6: Exposure

Getting Started

AIM

The aim of this chapter is to familiarize yourself with the basic functions of the *e_LiNE* system. The first task is to switch the system on , load the sample and to obtain an SEM image of your sample.

As the starting point for this chapter it is assumed that the system is on, but that no one is logged in.

Task 1 Start the system

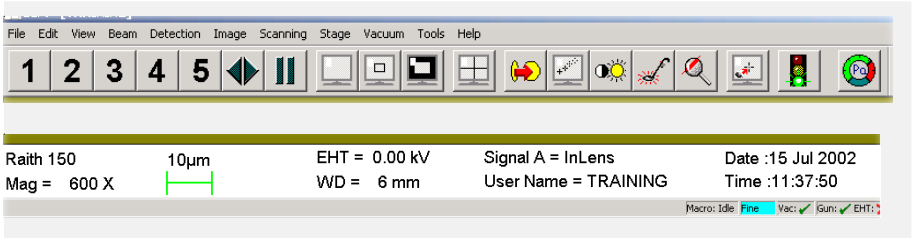
HINT

If the system has been left in another status, i.e. switched off completely, please contact a specialist for advice. For the operation of the *e_LiNE* system, both the column and lithography software have to be installed and in addition the RemCon32 at the column PC must be running in order to provide the connection between them.

STEP 1 ▶

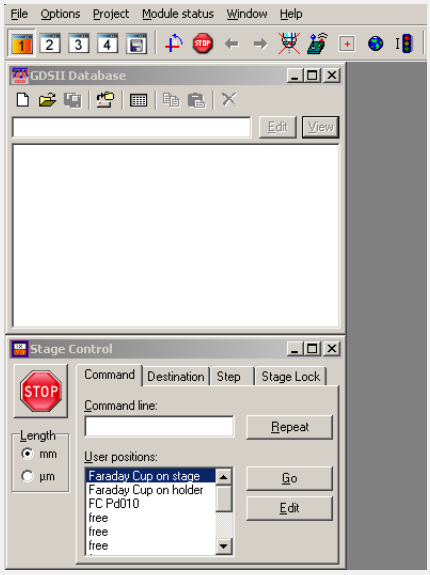
Start the column software and log in as user "training" and password "training". Ensure that the control program RemCom32 is running.

The column desktop displays the operation icons at the top and the image information, as well as the status controls at the bottom of the screen.



STEP 2 ▶

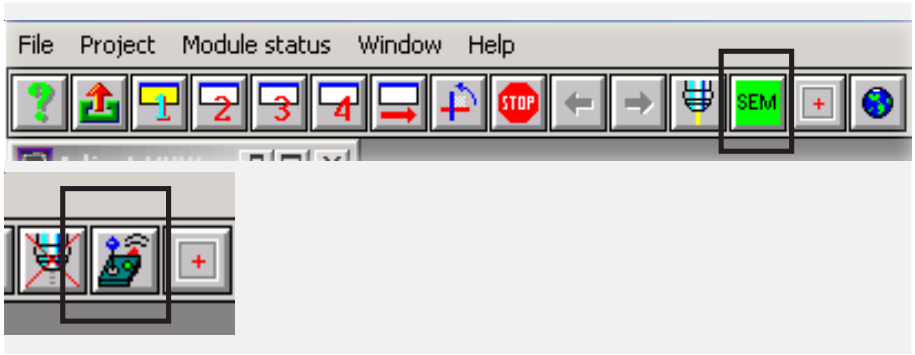
Start the RAITH lithography software and log in as user "training" and password "training". The desktop will show several windows opened.



STEP 3 ▶

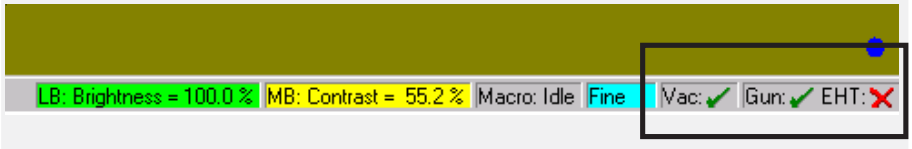
Check if the lithography software has control over the column software by clicking at the SEM icon in the lithography desktop. The icon has two modes; when showing SEM, the column is controlled through the column software, i.e. a scan is running. In the other mode the icon will display a remote control, in this case the column is controlled via the lithography software and the last scan will be frozen, therefore no running scans are shown.

As the icon is showing SEM, the column is under control of the column software. When the remote control icon is displayed, the SEM is under control of the lithography software.



STEP 4 ▶

Check for the status of the columns at the lower right corner of the column desktop, to see if the vacuum condition is OK, as shown in the lower right corner of the column desktop. We assume that the gun is running (green tick) and that the acceleration voltage EHT is switched off (red cross).



HINT

The toggle between Coarse and Fine control is a most useful feature. The Coarse and Fine control is always related to the currently selected parameter, such as Focus, Brightness, Alignment etc. All parameters which can be adjusted using the mouse can be either performed in Coarse or Fine mouse control. They also scale with the set magnification.

Task 2 Preparing a suitable sample

It is recommended that the sample should contain very small features suitable for imaging at high magnification with high contrast. For example, small metal particles can be added at the corner of a resist sample. Those particles will aid the SEM optimization which coincides automatically with the optimized beam conditions for exposure.

For this chapter we would recommend a small sample, for example a one square inch with a normal resist, e.g. PMMA. You will find this type of sample in the Starter-Kit provided with the instrument.

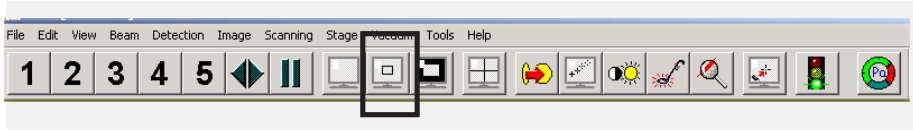
STEP 1 ▶ Use the latex spheres from your EBL Starter-Kit and dip it into the solution. Apply a small drop to the corners of your resist sample.

Although this method might not be adequate for the experienced lithography user, it will be most useful for a novice to gain some experience.

Task 3 Loading and unloading samples

STEP 1 ▶

We need to verify if a sample is loaded or not. To check this, use the CCD camera to view the inside of the vacuum chamber. Click on the Monitor icon in the columns desktop.

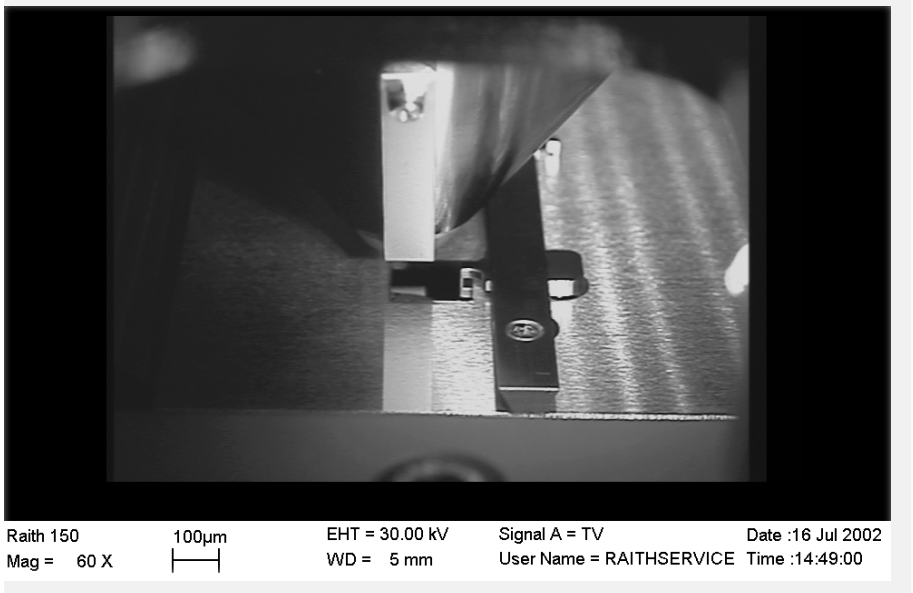


The CCD camera will now display an image. An example is given in the figure below. The image shows the system without sample holder.

A) If the sample holder is in the chamber, you have to unload it. This procedure is described in Step 3.

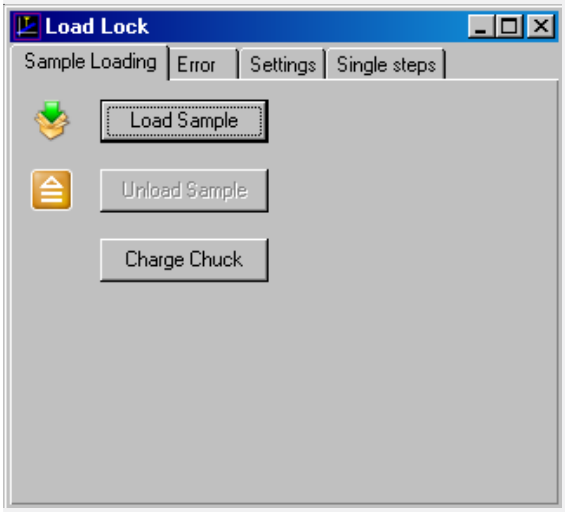
B) If there is no sample holder in the chamber, the following procedure will guide you to introduce one into the system:

Place the sample holder with your sample on the robot arm of the loadlock.



STEP 2 ▶

Check the current color of the Traffic Light icon number 1 in the lithography desktop. If it shows green, the Navigator Exchange is already open. Click on Sample Loading and then on the Load Sample button. The loading operation is marked red if a sample is already loaded.

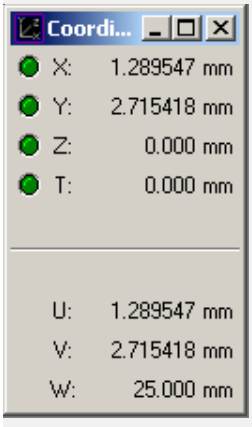


STEP 3 ▶

At the end of the automated loading procedure the user will be prompted whether the system should switch on the beam. Accelerating voltage and aperture can be typed in by the user. You should start with 10 kV acceleration voltage and 10 µm aperture. Then the system asks whether to reset the coordinates. It is safe to click OK.

To check whether the acceleration voltage is switched on, on the column desktop, there should be a green tick next to the EHT in the bottom right corner.

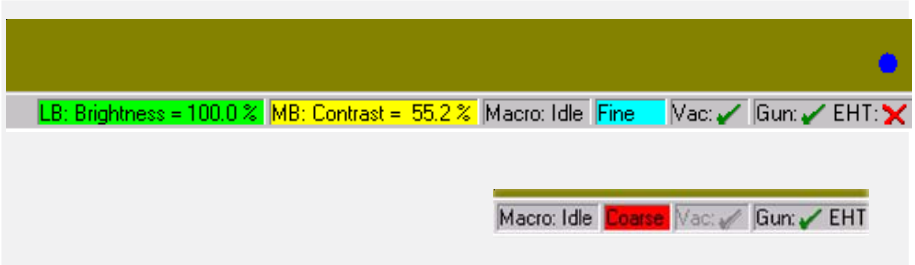
Check the home position. Use the lithography desktop, go to the Coordinates window and check if XYZ is displayed as zero.



Task 4 Obtaining an SEM image

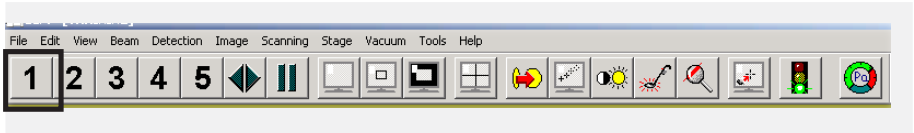
HINT

If the bottom line in the column desktop shows fine (light blue), change it to Coarse (red) by clicking on it once to widen the range available. At the start you might be a long way out of focus and you might therefore expect to see a noisy and grey picture. To obtain an image you need to adjust the column parameters as explained in the next steps.



STEP 1 ▶

Select Scan Speed '1' icon using the column desktop. A fast scan will be produced. During the fast scan, only noise can be seen as the acceleration voltage (EHT) is still switched off. Click on scan speed '1' to obtain a fast scan.

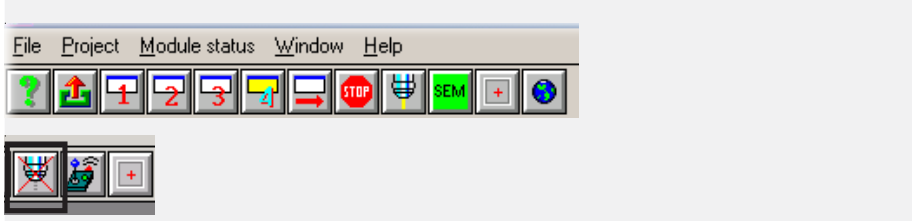


STEP 2 ▶

The EHT will be switched on automatically during sample loading. If it is switched off, click on the small EHT icon in the column desktop in the bottom right corner. A dropdown list box appears. Select EHT ON.

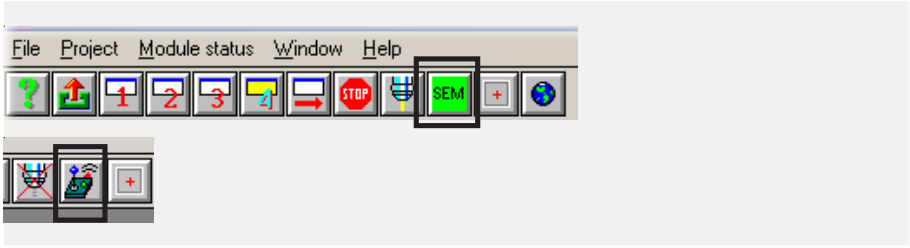
STEP 3 ▶

The beam blanker should be in the OFF state. To check this, click at the column icon left from the SEM icon in the lithography desktop and check if the beam blanker changes the signal during the scan. Leave the beam on.



STEP 4 ▶

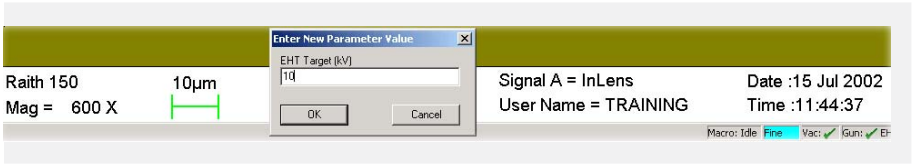
In addition, the column should be under internal control (SEM icon). If it is not, click on the remote control icon, the icon will then change to the SEM icon.



STEP 5 ▶

The next step is to check the acceleration voltage.

Double click on EHT in the data zone and a dialog window will open. Enter the required acceleration voltage, for example 10 kV

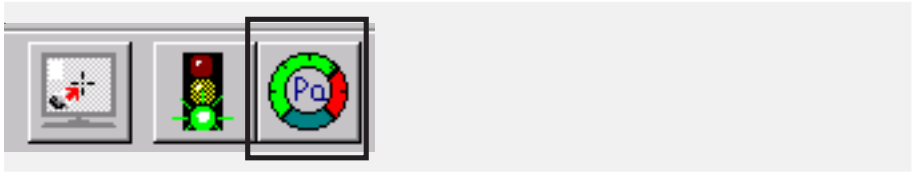


STEP 6 ▶

You now need to select a low magnification, as this makes it easier to obtain the first SEM image. Go to the Data Zone and double click on Mag. Enter a low magnification, for example 50. You should now be able to obtain a rough image of the sample holder. For better focusing follow the instructions on the next page.

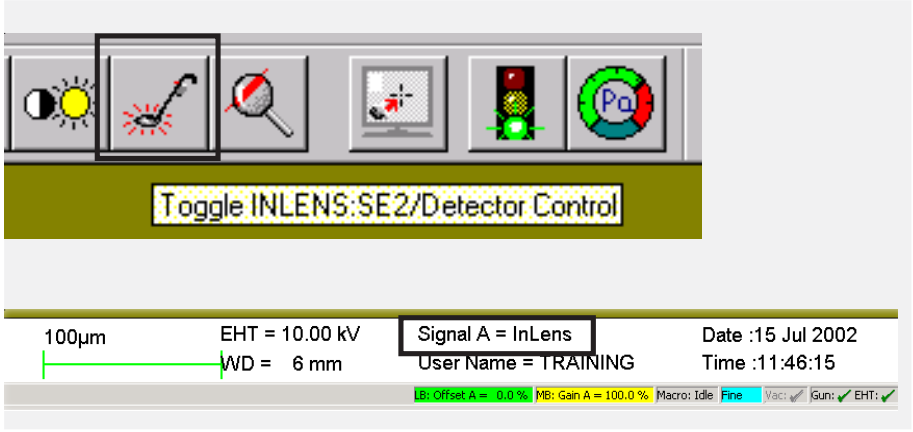
STEP 7 ▶

Go to the SEM Control dialog box and select Apertures. The Aperture tab will open. Select the required aperture, for example 10 µm from the drop-down list box. The SEM control can be opened by clicking on this icon. Alternatively, the SEM icon can be opened by pressing Ctrl G.



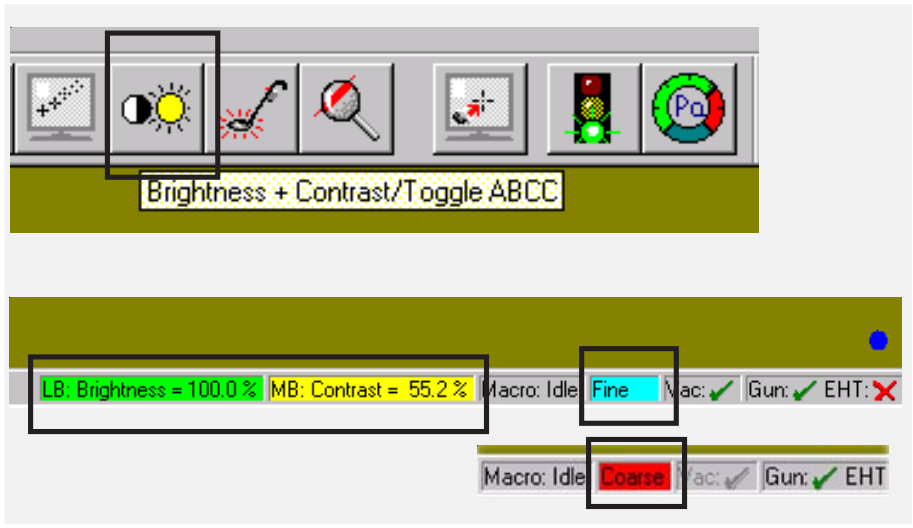
STEP 8 ▶

Click with the left mouse button on the Detector icon in the column desktop, which changes between INLENS detector and SE2 detector. Ensure, that the INLENS detector is used. Check in the data zone that the required detector is selected. Use the INLENS detector while you are still getting familiar with the system, as it provides a higher signal. The INLENS detector can be used for voltages up to 20 keV.



STEP 9 ▶

The next step is to adjust brightness and contrast. Click the icon for Brightness and Contrast. The left and middle mouse button will now be assigned for controlling brightness and contrast respectively by horizontal mouse movements. This assignment is shown on the bottom line. First, press the left mouse button and move it while pressing it down to adjust the brightness; then use the middle mouse button and the same movement to adjust the contrast. For getting first images a setting of Contrast=Brightness=50% will be sufficient.



The left mouse button LB is assigned to brightness control and the middle mouse button MB is assigned to contrast control.

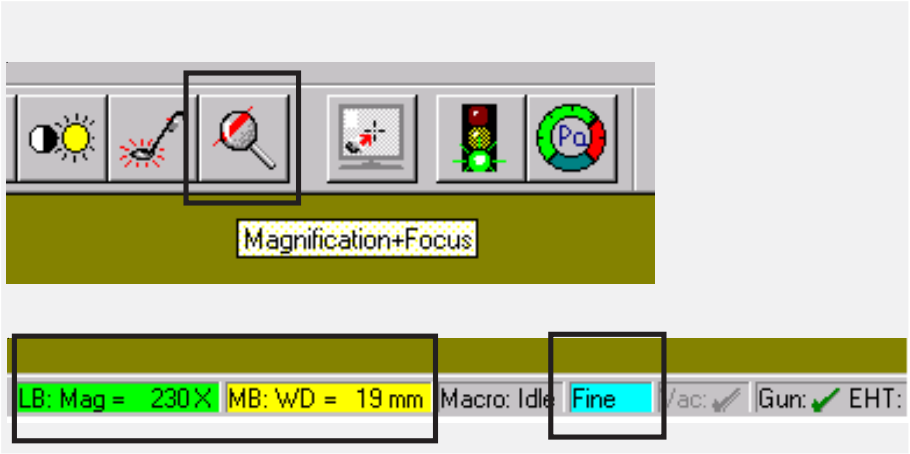
The mouse movement can be toggled between Fine and Coarse by clicking in this field once.

HINT

Click on the Brightness and Contrast icon using the middle mouse button in order to start an automatic Brightness and Contrast optimization. Afterwards click the icon again with the middle mouse button to switch off the automatic optimization.

STEP 10 ▶

Now that the Brightness and Contrast have been optimized, we can now start to focus onto a surface using a selected magnification of 50x . Click on the Magnifying Glass icon using the left mouse button and assign the left and middle mouse button to magnification and focus control during horizontal mouse movements. Now you can optimize the focus by pressing the middle mouse button and moving the mouse from left to right or vice versa.



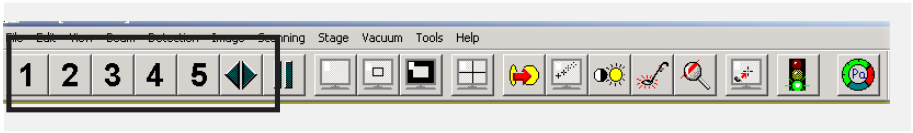
The left mouse button LB is now assigned to Magnification control and the middle mouse button MB is assigned to Magnification control. The mouse movement can be toggled between Fine and Coarse control.

HINT

Please note that focus is related to working distance.

STEP 11 ▶

As the sample is now in focus, a higher quality image (lower noise) can be obtained by changing the scan speed to a higher value, as this reduces the scan speed. Select a slower scan speed in order to reduce the noise by clicking the left numbered icons or freeze an image by clicking the icon with the double bar.



The scan speed can be changed using these icons. The higher the number, the slower the scan speed, the higher the image quality (lower noise).

Clicking on these icons with the middle mouse button will switch imaging to continuous averaging. To get started, middle mouse click on icon “2”.

Task 5 Finding your sample

STEP 1 ▶

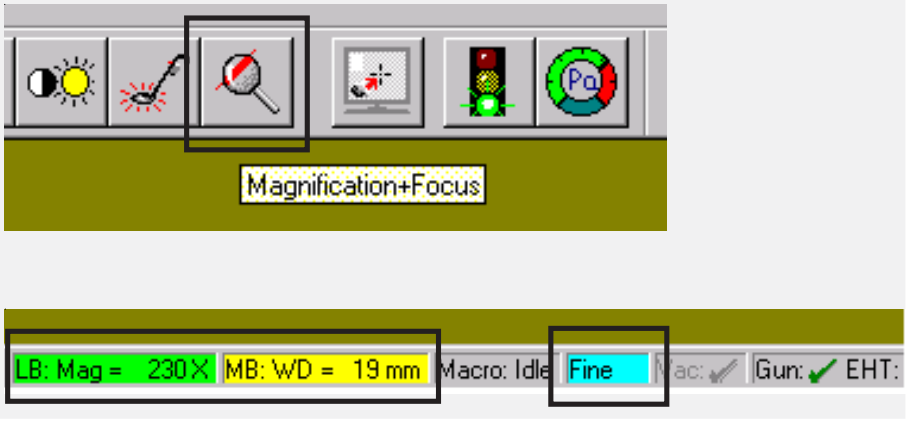
You can use the joystick to drive the stage to the desired position.

Switch on the X and Y buttons in order to illuminate corresponding LEDs. You can now move the stage at variable speed, depending on joystick inclination. The LED on the joystick indicates the corresponding axes, which are now under joystick control.

Move close to your sample but do not move over your sample, otherwise you would start the exposure when you switch back to the SEM image.

STEP 2 ▶

We can now start to focus onto the sample holder using our selected magnification of 50x. Click on the Magnifying Glass icon using the left mouse button to assign the left and middle mouse button to magnification and focus. Now you can optimize focus by pressing the middle mouse button and moving the mouse. Mouse movement can be toggled between Fine and Coarse control.



HINT

In addition the speed of stage movement can be doubled by pressing the first left button on the joystick.

HINT

The joystick axes are aligned to the SEM image, which is rotated by 90° with respect to the CCD camera. The stage movement on the CCD camera will therefore appear different to the way you may expect. To get started, just rotate the joystick by 90° counterclockwise.

Since the CCD camera views the sample at a slight angle, the image will vary in appearance depending on the working distance (pole piece to sample distance). Some practise may be required at first when working close to the sample. The only way to check, whether the sample is located under the beam, is to observe the SEM image.

STEP 3 ▶

Now that you have optimized the focus, you need to locate the sample at low magnification. Click on the Raster icon to switch back to the SEM image. Move the lower left corner of your sample into the center of the field view.

STEP 4 ▶

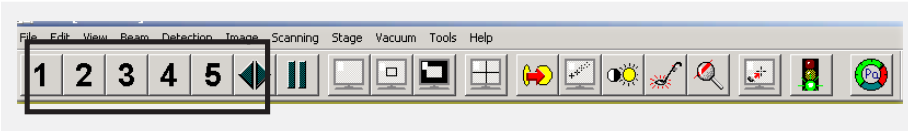
Now that you have optimized the focus, you need to locate the sample at low magnification. Click on the Raster icon to switch back to the SEM image. Move the lower left corner of your sample into the center of the field of view.

HINT

You can turn on a crosshair, indicating the centre of your screen, by clicking on the monitor icon with the centered cross.

STEP 5 ▶

As the sample is now in focus, a higher quality image (lower noise) can be obtained by changing the scan speed to a higher number, as this is reduces the scan speed. Change the scan speed to slower scan speeds in order to reduce the noise by clicking the left numbered icons or freeze an image by clicking the icon with the double bar. The scan speed can be changed by these icons. The higher the number, the slower the scan speed, the higher the image quality (lower noise).



E-beam Optimization

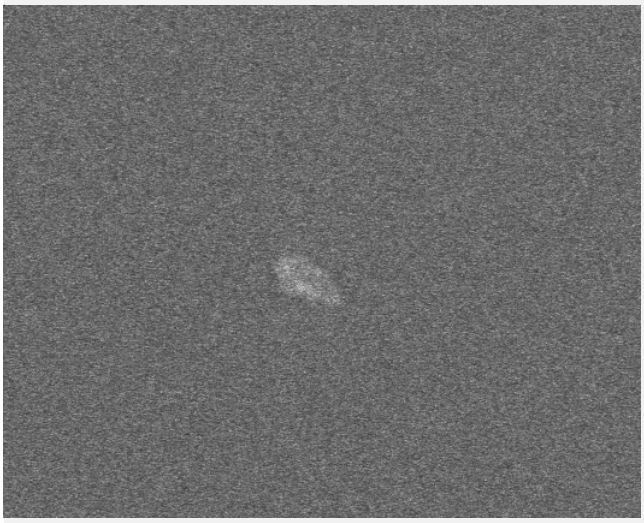
AIM

This chapter explains how to optimize the column setting in order to get a good exposure by selecting the correct parameters.

Task 1 Focusing on the sample

STEP 1 ▶ It is assumed that you have loaded a 1 square inch sample into the system as described in the first chapter. Select a small particle of less than 1 μm on your sample.

STEP 2 ▶ Move the particle into the center of the field by using the joystick.



STEP 3 ▶ Zoom onto the particle until you seem to lose the focus. Remember that the zoom is assigned to the left mouse button after the magnification icon has been selected, as described in detail in chapter 1.

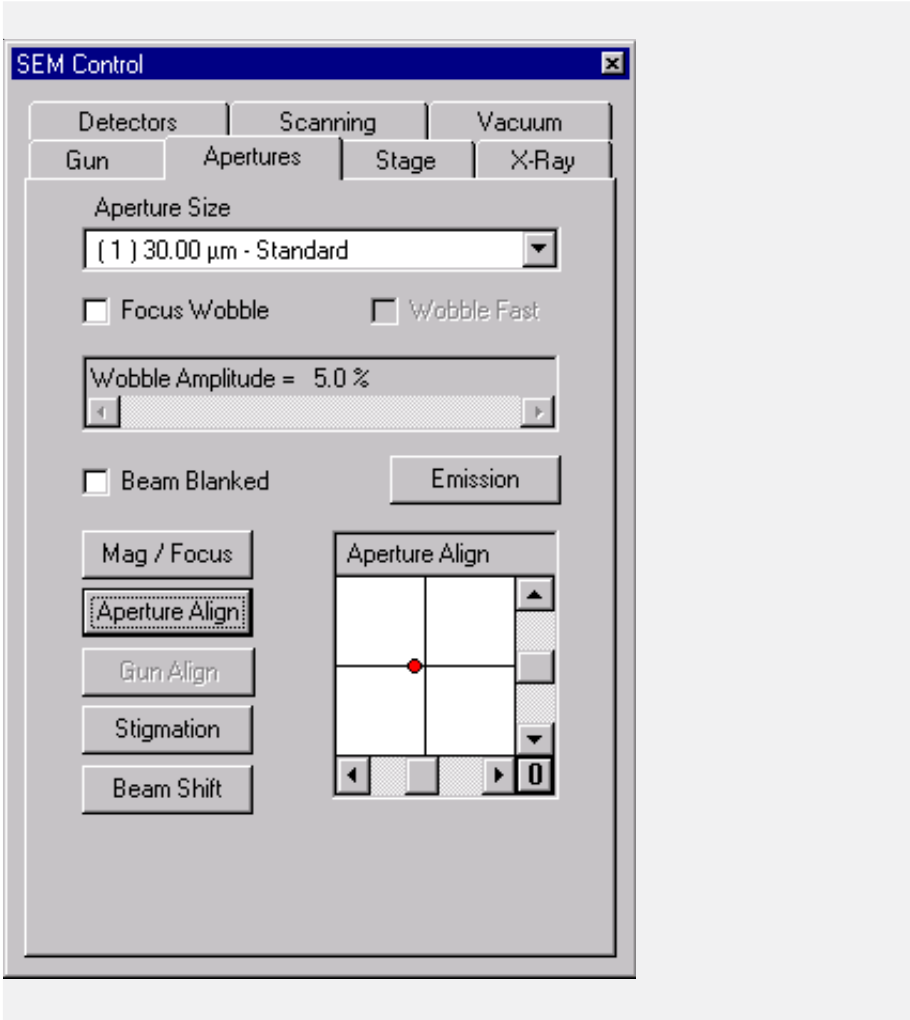
STEP 4 ▶ Refocus onto the particle. Remember that the focus is assigned to the middle mouse button.

STEP 5 ▶ Zoom in further and readjust the focus.

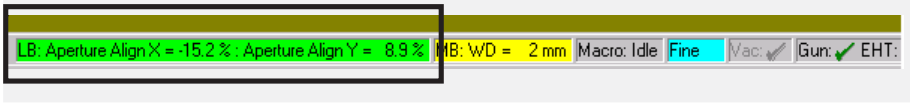
STEP 6 ▶ Repeat the zoom and refocus procedure until no further improvement in focus can be achieved.

Task 2 Aperture alignment

STEP 1 ▶ Open the SEM control panel Tools > Go to Control Panel (Ctrl-G) and select the Apertures tab.



STEP 2 ▶ Click on Aperture Align, which assigns the left mouse button to the aperture alignment in XY by moving the mouse in X and Y directions. The assignment is displayed in the status bar at the bottom of the screen.

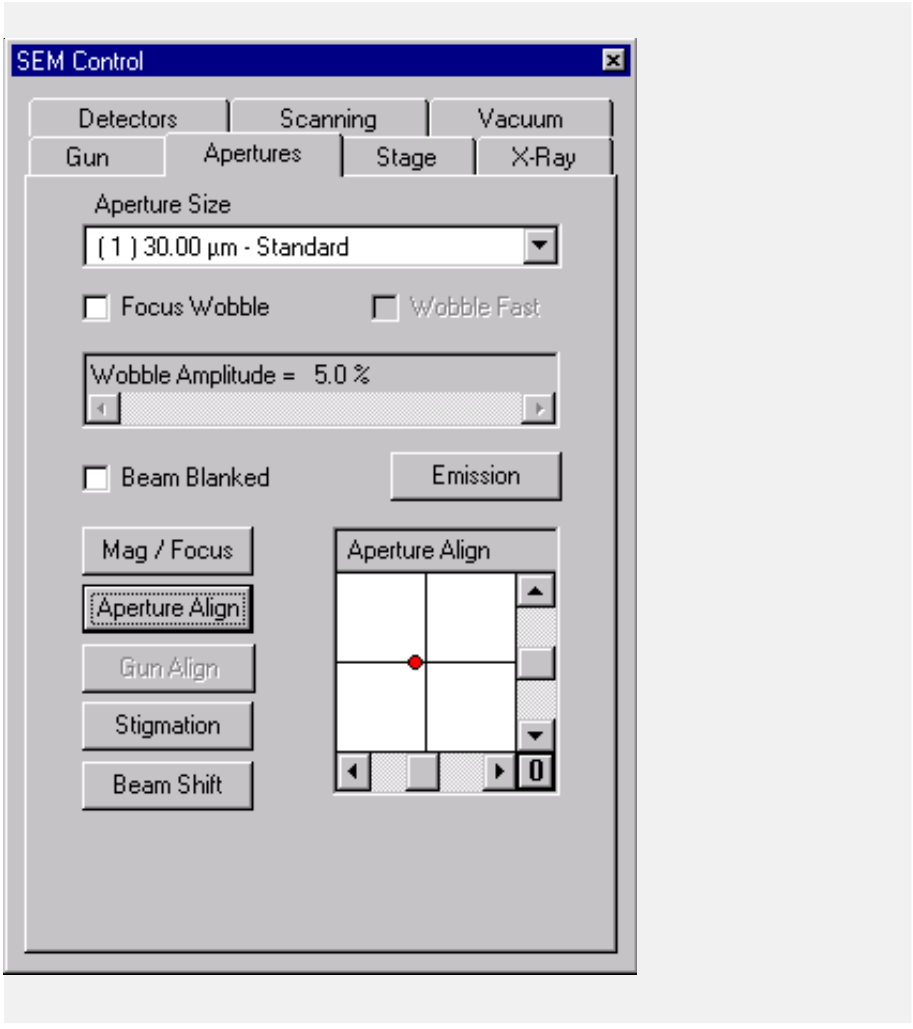


STEP 3 ▶

Keep the left mouse button pressed and move the mouse in X and Y directions. You can observe the changes by viewing the image and a corresponding movement of the red point in the window. In addition you can place the cursor on the red point and drag it around while keeping the left mouse button pressed. A third alternative for adjustment is using the scrollbars.

STEP 4 ▶

The next step is to switch on the Focus Wobble in the Apertures tab of the SEM Control by marking the corresponding field and selecting a useful amplitude for the current magnification. This will cause a periodic change of focus and during this cycle the image will sometimes move.
Select a suitable Wobble Amplitude by using the slider bar.
Observe the movement of the red point in the Stigmation window.



STEP 5 ▶ The key of the aperture alignment is to minimize the image shift during the wobble sequence. To achieve this, move the mouse in the X and Y directions while keeping the left mouse button pressed and optimize for lowest image movement.

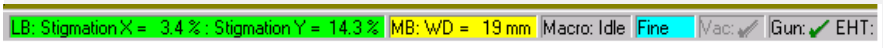
STEP 6 ▶ You might be able to improve the aperture alignment even further by repeating the same procedure at higher magnification and reduced wobble amplitude.

HINT

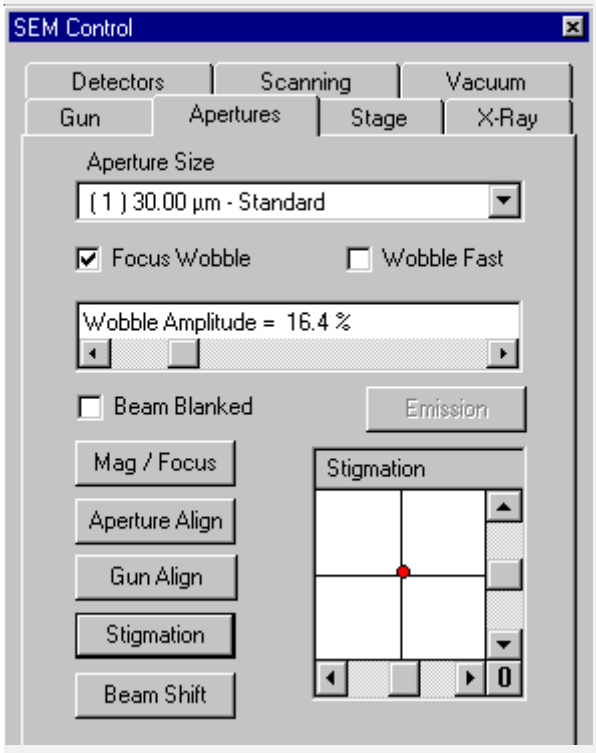
If the dust particle is becoming too large at high magnification, move to a smaller dust particle and continue the optimization. In order to change the magnification, click on the button Mag/Foc. Do not forget to switch back to Aperture Align once finished.

Task 3 Astigmatism correction

STEP 1 ▶ Click on Stigmation, which assigns the left mouse button to the stigmation alignment. The adjustments are carried out in the same manner as the aperture alignments.



STEP 2 ▶ Switch on the Focus Wobble in the Apertures tab of the SEM Control by clicking on the corresponding field and select a useful amplitude for the current magnification. During the wobble sequence the particle will be stretched first in one direction and then in the perpendicular direction.



STEP 3 ▶ Optimize for lowest shape changing of the particle.

Task 4 Further E-beam optimization

For the final optimization of the E-beam, you need to change between Aperture Alignment and Astigmatism Correction several times in order to optimize the setting for high image quality at high magnifications. The final result should be a well resolved image of the particle at a magnification of 300.000x or higher. If not, create a contamination dot as described in the next task.

HINT

Please note that during the aperture alignment we concentrate on the image movement while during the stigmation optimization we will concentrate on the shape changes.

STEP 1 ▶ Perform the Aperture alignment again at higher magnification and reduced wobble amplitude. In order to change the magnification, click on the button Mag/Foc and use the mouse.

STEP 2 ▶ Perform the Astigmatism correction again at a higher magnification.

STEP 3 ▶ Continue the alignment optimization without the use of the automatic focus wobble (deselect Wobble) and use instead alternating Aperture Alignment (left mouse button) and the manual Focus (middle mouse button). The aim is an aperture alignment which avoids image shift during defocusing. This method allows a more precise adjustment than the automatic wobble and is recommended for the final optimization steps.

STEP 4 ▶ Repeat the same procedure between the optimization of Aperture Alignment and Astigmatism correction until no further improvement can be achieved.

Task 5 Creating a contamination dot

After completing the e-beam optimization using the latex particles, it is recommended to burn a contamination dot for the final optimization of the aperture alignment and astigmastism correction.

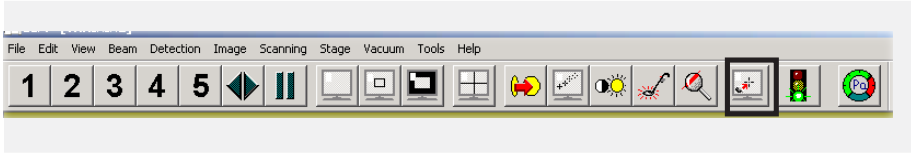
STEP 1 ▶

To carry out the final optimization, move the stage slightly away from the latex particles to ensure that a free area of the sample is visible in order to burn the contamination dot.

STEP 2 ▶

Click on the Dot icon on the column desktop using the left mouse button to burn a dot for a duration of 3 s. The software will automatically switch to the reduced scan area.

If you were not able to burn a visible contamination dot, click on the middle mouse button which will start the dot exposure. Wait for 1 minute while the dot is burned into the sample and click the middle mouse button again to end the dot exposure.



STEP 3 ▶

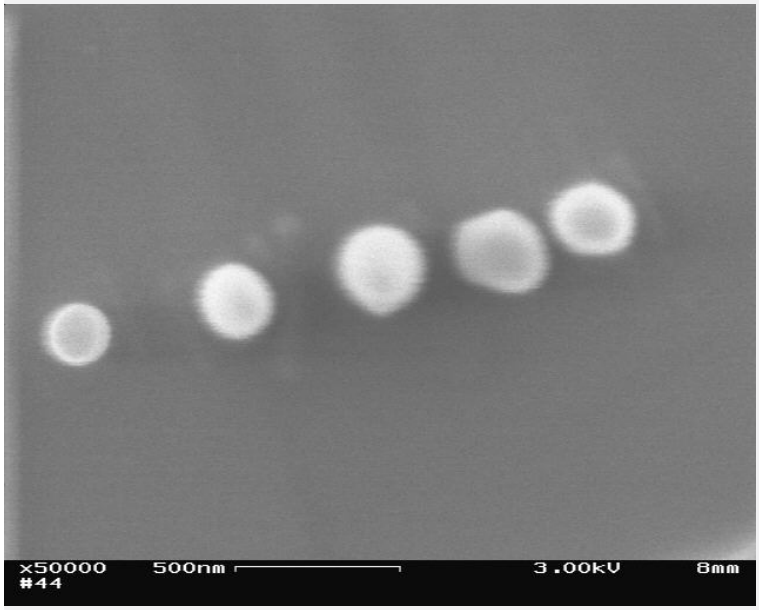
Focus now on the contamination dot, move the stage and burn another contamination dot. The new dot should be smaller since the focus has been improved.

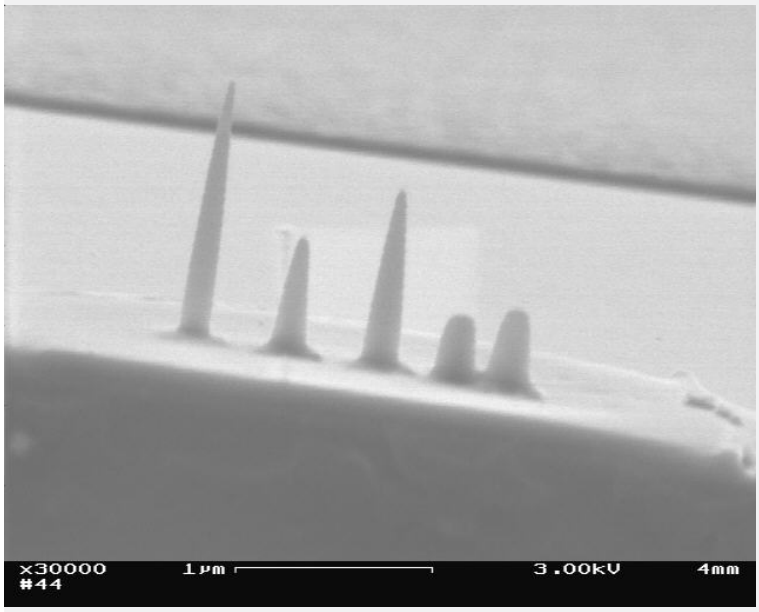
HINT

If the dot is not round, apply the aperture alignment and then the astigmatism correction again, using this dot. Using such alternating routines, it is possible to achieve an ideal round dot, which grows within a few seconds of exposure time and shows perfect alignment. The optimization on this dot now provides the optimized conditions for a real exposure nearby.

HINT

An example of contamination dots is shown in the images below to illustrate top and side views.





Task 6 Checking the leveling limits

It is likely that your sample surface is tilted to the beam. This can be checked by the following tasks, but this task is not necessarily required prior to an exposure.

STEP 1 ▶ Switch to a lower magnification and move the stage for a relatively long distance, i.e. 1 mm. Ensure that you notice the direction of movement in order to relocate the previous contamination dots.

STEP 2 ▶ Burn another dot and view the result. This dot now is likely to be larger than the previous one, but this time the focus adjustment should be sufficient for the optimization. It should not be necessary to perform the Aperture Alignment and Astigmatism correction again.

STEP 3 ▶ Perform some experiments to establish the stage travel distance, at which you need to refocus the sample surface.

HINT

To improve the leveling of small samples, it is recommended to use two opposite clamps. In addition, routines to solve the leveling problems will be explained in later chapters.

Stage Adjustment

AIM

This chapter describes stage adjustment, which allows navigation with a blanked beam on the sample in order to find a new exposure area without pre-exposing or to find an already exposed and processed area for inspection or multi-layer exposure. The two coordinate systems (XY for the stage and UV for the sample) will be explained in detail, thus permitting the determination of the correct UV sample coordinates independent of how the sample has been mounted on the stage.

The aim of stage adjustment is to find the relationship between XY and UV with respect to shift, scaling and rotation in order to perform a permanent coordinate transformation between both systems.

In this chapter we will explain in tasks 1 and 2 how to set up a coordinate system on an sample. In task 3 we will explain how to navigate on this.

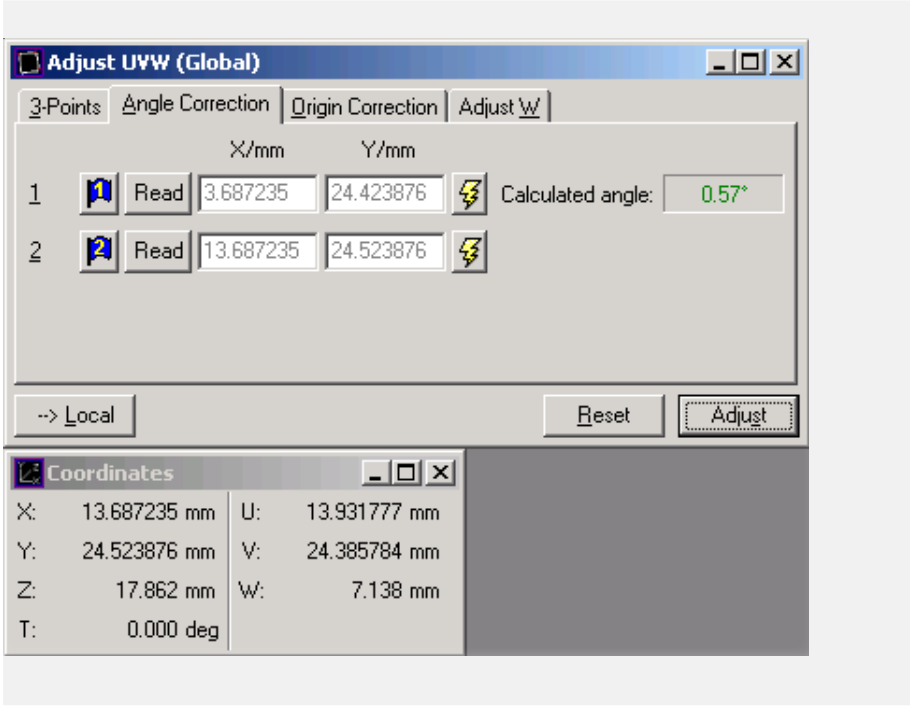
Task 1 Angle correction

Normally the axes of the sample surface will not be parallel to the axes of the stage. An angle correction can be carried out to compensate for this difference.

STEP 1 ▶ To carry out the angle correction, image the sample at medium magnification, approximately 100x. Ensure that the crosshair is switched on (View > Crosshairs) in the column desktop. Identify the lower edge of the sample and follow this edge to the lower left corner. The crosshair is now situated above the lower left corner.

STEP 2 ▶ On the lithography desktop, open the window Adjust UVW. Ensure that it is in mode Global; if it is in mode LOCAL, click on the button once to change it. Click on the Angle Correction tab.

STEP 3 ▶ In the coordinate window the actual XY coordinates are displayed. Click on Read at the first position of Flag 1 in Adjust UVW and the coordinates will be displayed in the window.



STEP 4 ▶ Once the coordinates are displayed, switch back to low magnification and move the stage a few millimeters along the sample edge to the lower right corner. Move the stage so that the cross hair is situated above the lower right corner. Click on Read at the second position of Flag 2 and the second set of coordinates will be displayed in the window.

STEP 5 ▶ Click on Adjust to calculate the angle.

This angle will now compensate the difference between the sample surface and the stage axes.

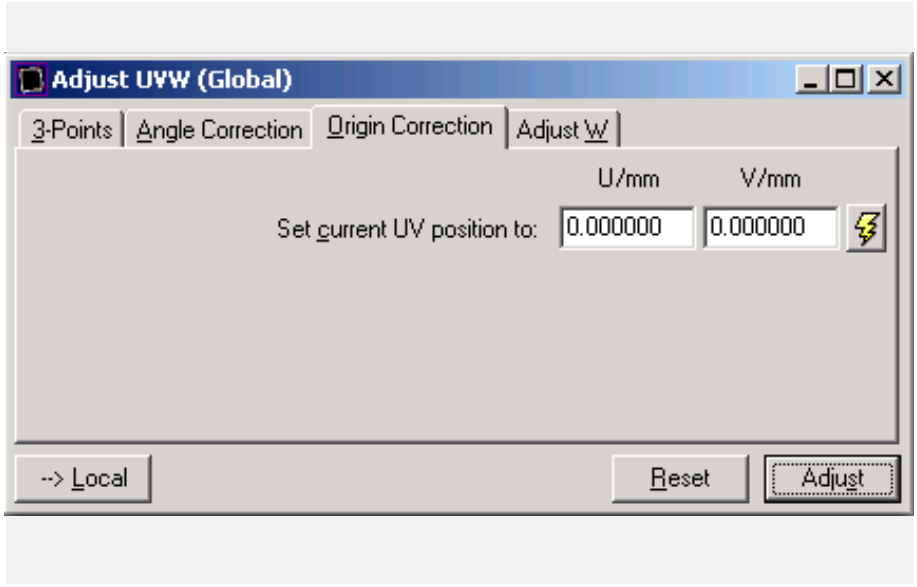
Task 2 Origin correction

The sample can be placed at any location on the sample holder. To compensate for the different origins of XY and UV, the origin correction can be applied.

STEP 1 ▶ Ensure that the beam is blanked.

STEP 2 ▶ Click on the lightning button of the first coordinates pair to move back to the lower left corner.

STEP 3 ▶ Click on the tab Origin Correction and enter 0 for both the U and V values, then click on Adjust. The lower left corner is now defined as the origin of this UV coordinate. It is now possible to move the stage to any point on the sample using UV coordinates.



Task 3 Adjust W

STEP 1 ▶ Make sure that your sample is still in focus by burning a new contamination dot.

STEP 2 ▶ Click on the Adjust W tab in the Adjust UVW (Global) window. Press the Read button and confirm. Then click on Adjust.

Task 4 Digital addressing

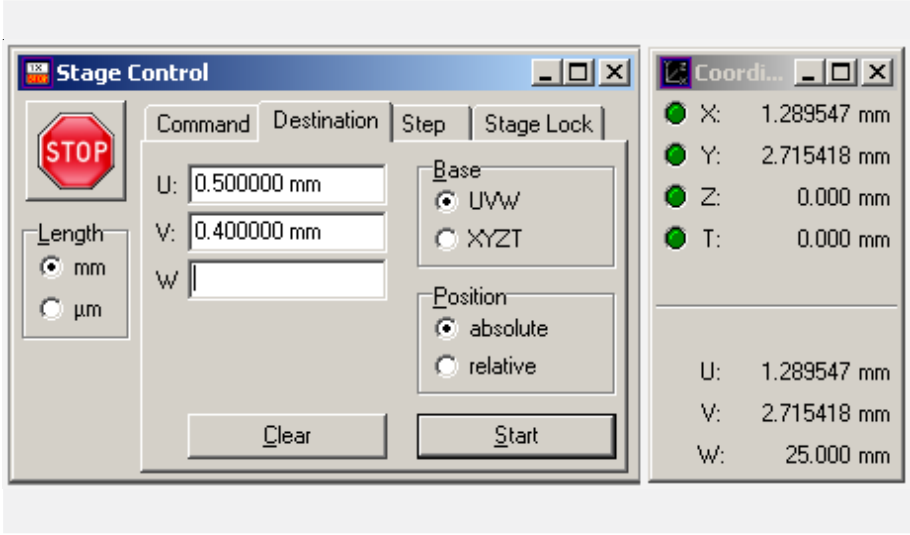
Digital addressing aids navigation on the sample. Digital addressing means that the user can enter a digital location as coordinates and the stage will drive to this location. This task is not vital for the exposure sequence.

In tasks 1-3 we have established a coordinate system in UVW, which we can now use to address certain points on the sample. This will be explained in this task.

Please note that it is not required to perform this task.

STEP 1 ▶

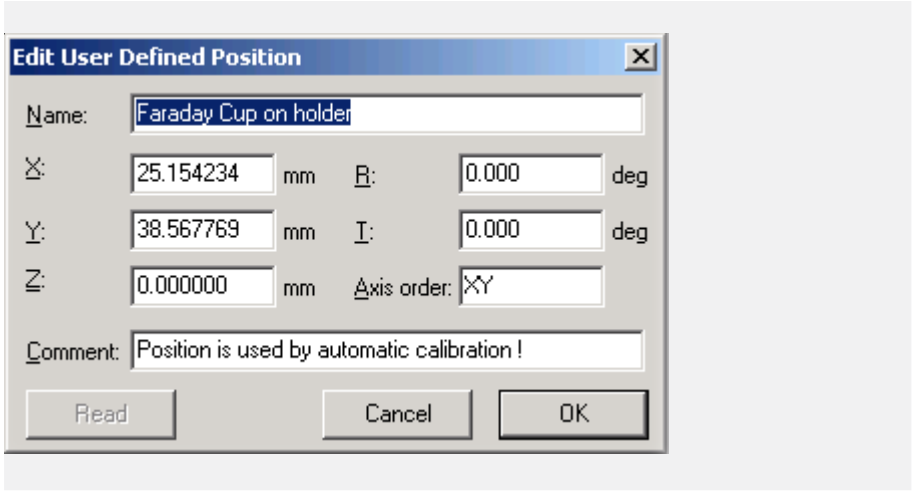
Locate the stage control window within Destination tab. Click on Base UVW and Position absolute. Now you can address the stage to any position in UV. W describes the working distance, which is directly related to the stage height Z. If you do not want to change the stage height (working distance) leave the corresponding line blank. After clicking Start, the stage will move to the sample position entered. In the coordinates window you will see the addressed sample position and the corresponding position in XYZ.



STEP 5 ▶

It is also possible to go to a stored position. In this example, the stored position is the Faraday cup. To edit a position, you can either enter the required position or you can read the actual position, if the stage is already at the selected point.

If the stage is already at the specified position, click on Read to store the coordinates.



Write Field Alignment

AIM

This chapter explains the alignment procedure for an exact writing field. In the previous chapters the image scan has been under the control of the column software. In order to perform lithography, the beam has to be controlled via the lithography software. For this a write field alignment has to be performed. The procedure described in this chapter via Align Write Field is required for stitching and for any exposures on a bare sample. The alignment of the field size to the previously written marks for multi-level lithography will be explained in a later chapter.

The Align Write Field is a very important task, as it aligns the write field to the sample coordinates UV. In chapter 3, we performed a point navigation in UV, but the image via the column software was still parallel to XY at a certain point and non parallel to UV. For pattern stitching it is essential that the write field is exactly parallel to UV and this can be achieved with the Align Write procedures.

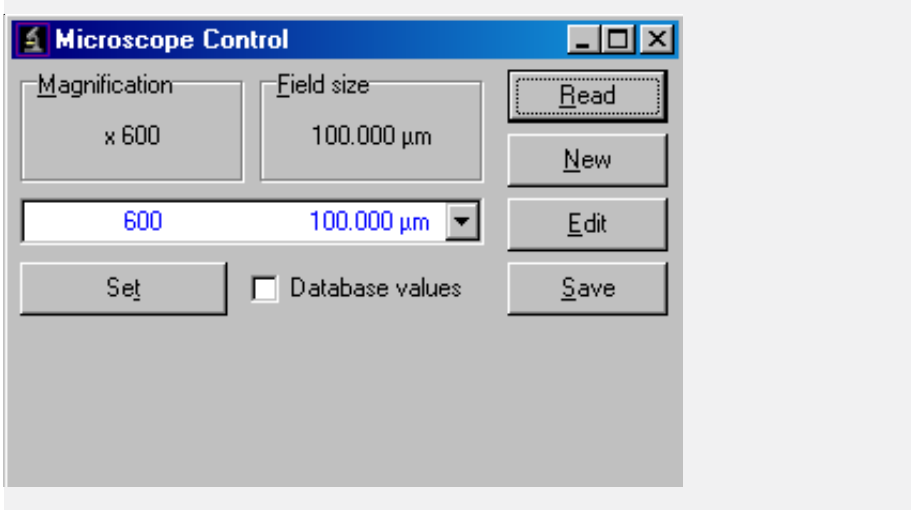
Task 1 Locating a mark or particle
Task 2 Defining the alignment procedure
Task 3 Executing the alignment procedure

Task 1 Locating a mark or particle

STEP 1 ▶ Move the stage back to the lower left corner of the sample. Please note that you can use the lightning icon in the Adjust UV window on the origin correction.

STEP 2 ▶ Locate a small particle which can be used as a mark for the following tasks.

STEP 3 ▶ Choose the window Microscope Control from the desktop. Select the write field size, in this case 100 µm. If initial correction values should be taken from a database, select option Database values. Finally, press Set to send correction values to pattern generator as well as column.

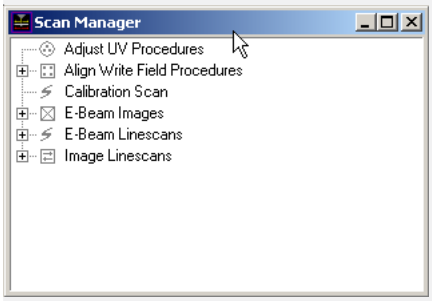


STEP 4 ▶ Click on File and select New Image.
Scan this image by choosing the menu command Slowscan, Image.
The particle is located near the center.

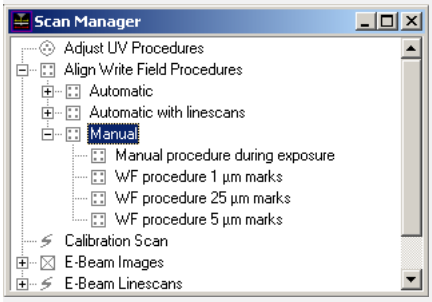
Task 2 Defining the alignment procedure

The write field needs to be calibrated and turned. This procedure is called write field alignment. The idea of this procedure is to scan some marks on the chessy sample. From the difference between the detected position in comparison to the ideal position, it is possible to calculate the scaling, shift and rotation of the write field. Within the scan manager, all the parameters of such a procedure are stored and can be recalled for later usage.

STEP 1 ▶ Choose Scan Manager on the desktop window.



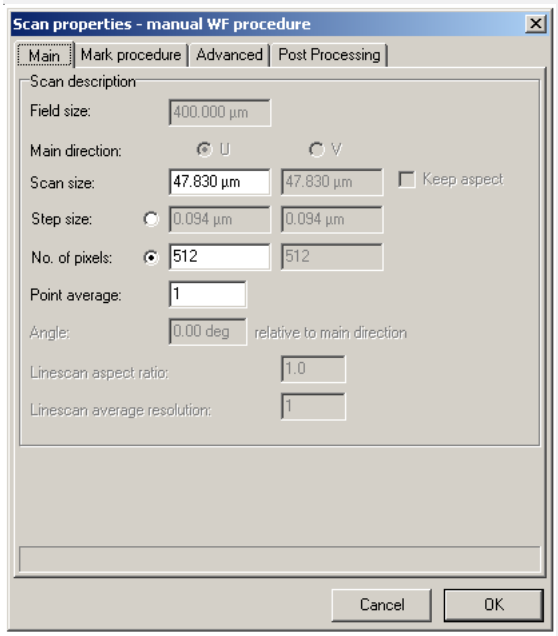
STEP 2 ▶ Choose Align Write Field procedure and Manual from the sub-procedure menu.



STEP 3 ▶ If a suitable sub-procedure is already available, drag and drop it onto the mark in the image. You can continue with the next task.

STEP 4 ▶

If no suitable sub-procedure is available, go to the menu command Edit and select New, which will create a new sub-procedure for Manual. The property window for the chosen procedure will automatically open.



HINT

Finding the suitable write field size is dependant on several factors such as:

- For a newly defined write field it is recommended to start with scan sizes as large as possible. We have selected a 100 µm write field. The scan size should be of the order of 50 µm.
- For a write field which has been used successfully beforehand, a smaller scan size can be used. Suitable scan sizes can be in the range of several µm, e.g. 5 µm.

HINT

It is recommended to rename the manual Writefield procedure to show the write field size in the title. Right mouse click opens the dialog window. Select rename and enter the new name. This will distinguish this procedure from other Writefield procedures of different sizes.

We will assume that no sub-procedure is available to explain the steps.

Otherwise please continue with Task 3.

STEP 5 ▶

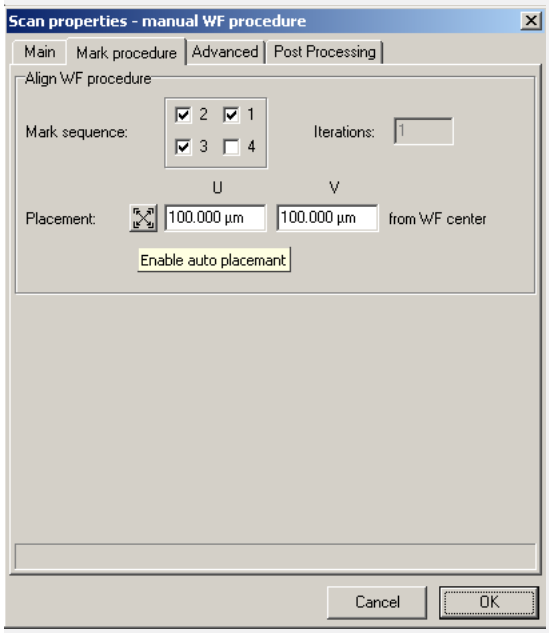
Toggle on the radio button for the number of pixels and enter 300 for U and V. Enter the scan size for U and V, in this case 40 µm. If you exit this input field the software automatically adjusts the scan size according to the number of pixels.
Use 1 for point average.

HINT

If a non-valid value for mark size or number of pixels is entered, the software will give a warning prompt message. If the placement is invalid, the software will also give a warning prompt message. The OK button will be disabled and greyed out.

STEP 6 ▶

Choose the Mark procedure tab. Check the Mark sequence 1, 2 and 3 for example. For Placement parameter, enter 25 µm in U and V.

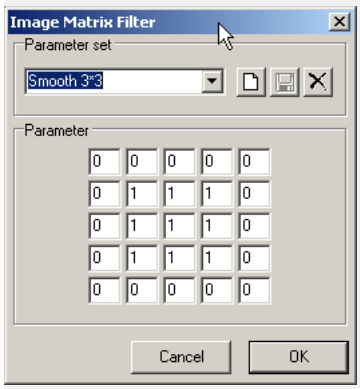


HINT

You can enable the Autoplacement function by pressing the auto placement icon. The software will now place the marks automatically as far as possible into the far corners of the write field.

STEP 7 ▶

If you have obtained a noisy image, select the Post Processing tab. Choose the Edit icon which opens up an Image Matrix Filter dialog. Select a Filter from the dropdown list or create a new one (see reference menu). Confirm with OK.



STEP 8 ▶

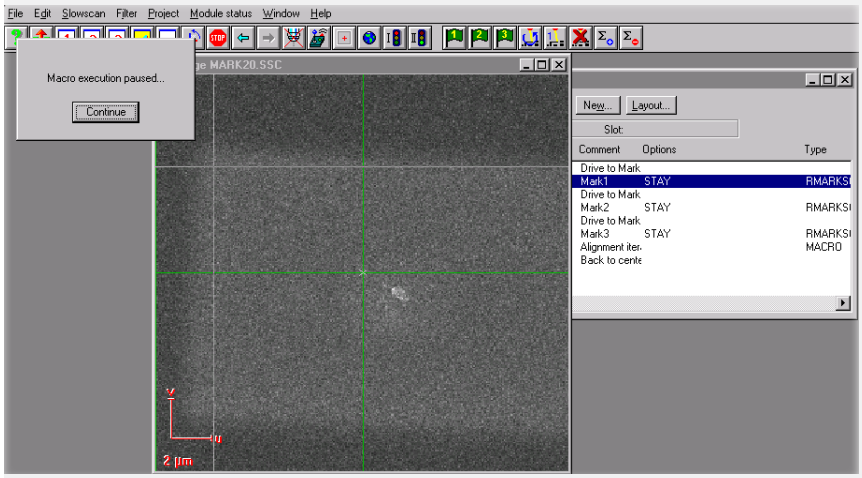
Drag and drop this new scan onto the mark in the image. A position list will open up automatically.

Task 3 Executing the alignment procedure

We will now execute the alignment procedure, which will scan the three mark areas to determine the difference between the real and ideal positions.

STEP 1 ▶ Ensure that the position list, where the procedure has been stored, is active. Choose from the menu Scan > Selection.

STEP 2 ▶ Click on Scan > Select All from the dropdown list box. This will initiate the alignment process. First, the stage will move 25 µm in U and V towards the first corner and an image will be scanned at the reference point. The image will cover a 49 µm x 49 µm square. The Mark window will be automatically opened, where the particle should be visible.
The green cross shows the position where the mark is expected.

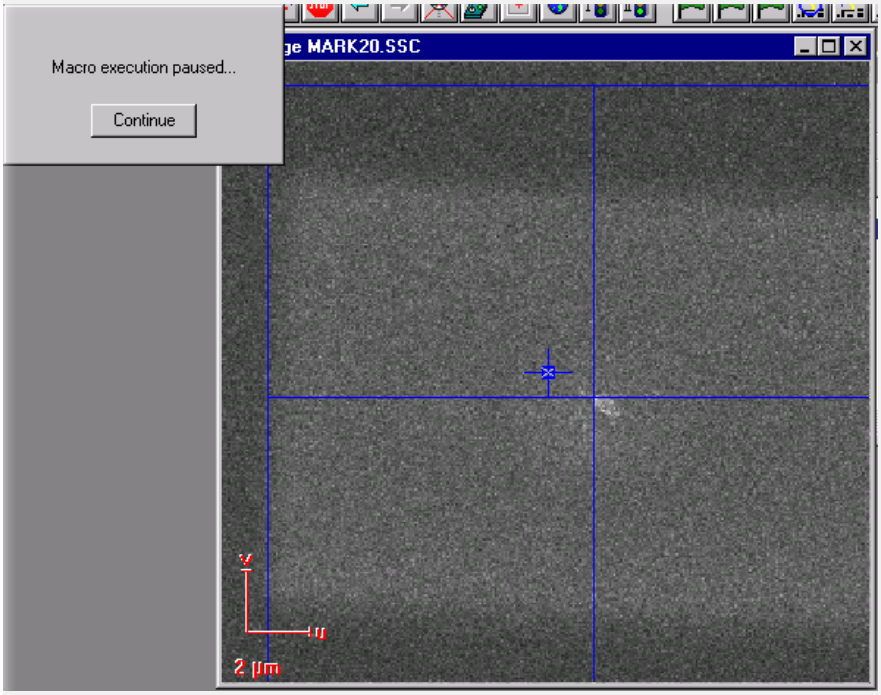


HINT

If no mark is shows up, highlight the position list, click the cancel icon (red cross) and confirm the Continue prompt. Repeat the task, now choosing a larger mark size.

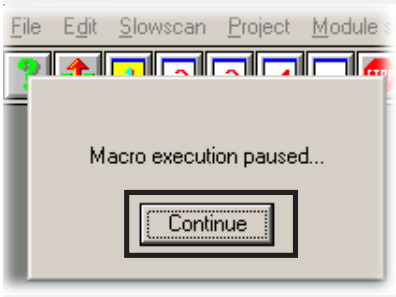
STEP 3 ▶

The green cross, displayed in the center of the image, defines where the special mark feature is expected. At this stage, the mark will probably not be at the center, but it can now be defined manually. To define the position of the mark, keep the Ctrl key and the left mouse button pressed while moving the mouse cursor to the required position. Once you have reached the new position, release the Ctrl button and a blue cross will be displayed at the selected position.



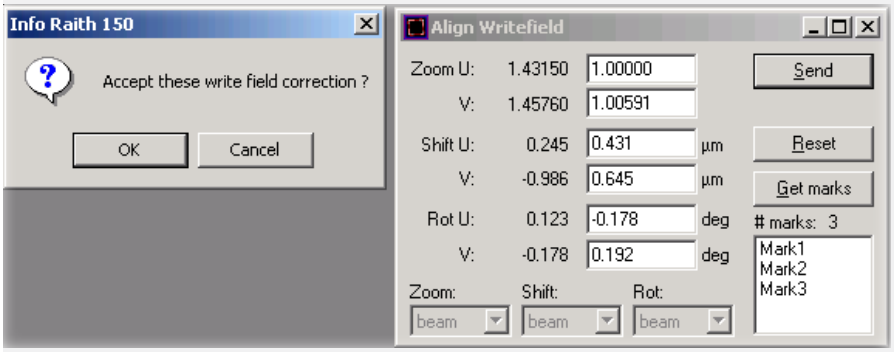
STEP 4 ▶

Click on Continue and the stage will move into the next corner to perform the same mark alignment. These steps must be repeated for each mark.



STEP 5 ▶

At the end of the position list a dialog window opens and write field correction has to be confirmed. Note the values of the Align Write field for Zoom, Shift and Rotation in UV and confirm if the values are acceptable.



HINT

The left column of numbers shows the alignment parameters before alignment. Here the scaling factors are around 1.4. Due to the alignment procedure, new alignment parameters have been calculated as shown in the center column. By accepting, these values will then be sent to the pattern generator and displayed on the left side.

HINT

If an alignment has already been carried out beforehand, the new values will be multiplied or added to the values on the left hand side.

STEP 6 ▶

Go back to the Align Write field and repeat this procedure several times by using smaller mark fields from iteration to iteration. In addition, the placement should be moved closer to the corner of the write field, e.g. 45 µm. The previous alignment parameters will now be used for the imaging, therefore the marks will be already positioned close to the center of the images. The final correction parameters in the Align Write field window should be very small or close to 1 for the scaling.

STEP 7 ▶ Activate the Microscope Control window and click on Save. The alignment parameters will be saved together with the magnification and the field size. Whenever you wish to call up this setting again, the correct field alignment will have been stored and you only need to perform the final optimization steps for the alignment.

Exposure

AIM

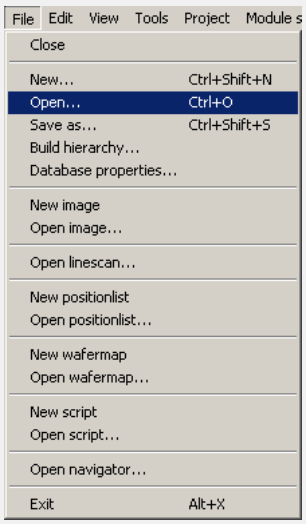
The aim of this chapter is to guide the user through the steps needed to carry out an exposure.

Task 1 Familiarization with demo pattern

Please note that you can go directly to Task 2, if you are already familiar with the demo pattern.

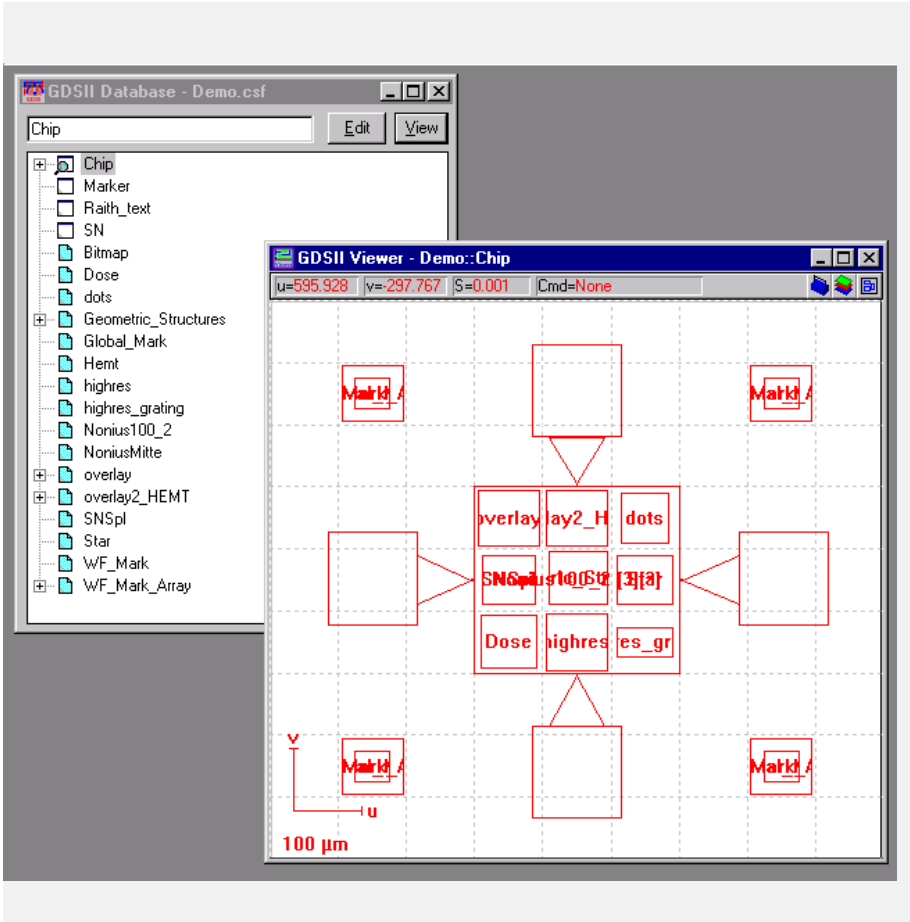
STEP 1 ▶

Click on the GDSII Database window to activate it. It may show a pattern already or it may still be empty. After activating the GDSII Database window, choose File > Open from the menu bar. A dialog box opens with a list of file names and folder options. Select Demo.csf.

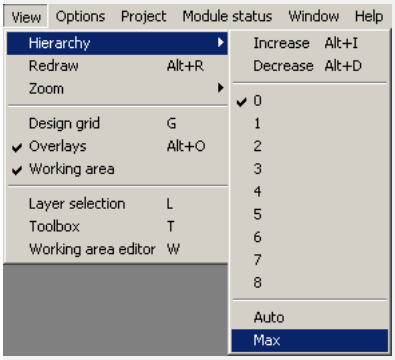


STEP 2 ▶

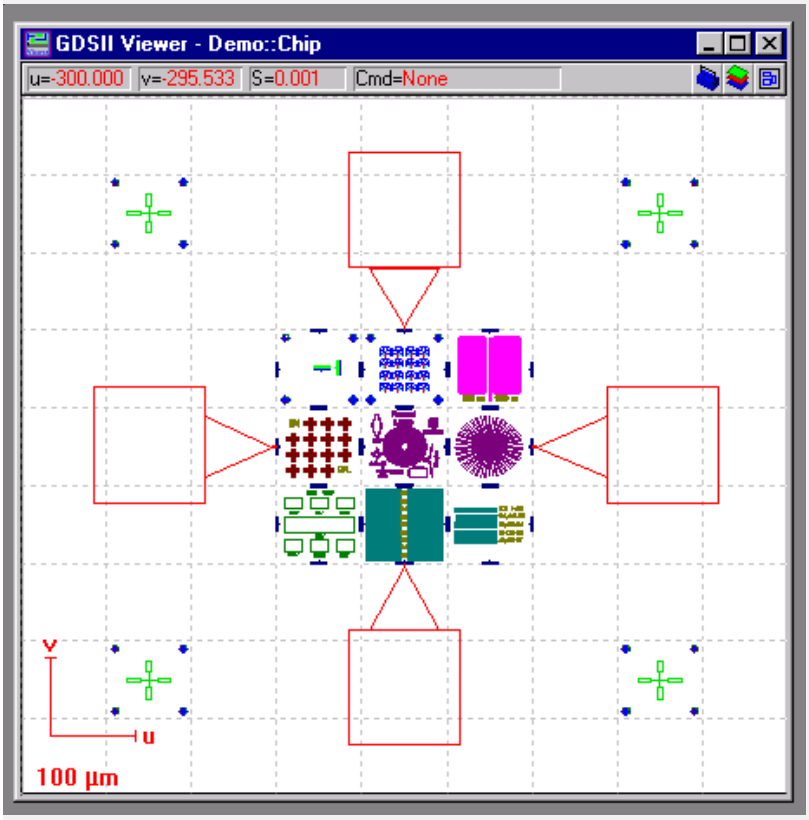
Select the pattern Chip, then choose View. Alternatively, you can double click on Chip. The GDSII Viewer will now display the hierarchical structure of the selected pattern.



STEP 3 ▶ While the viewer is activated, choose View > Hierarchy from the menu bar.

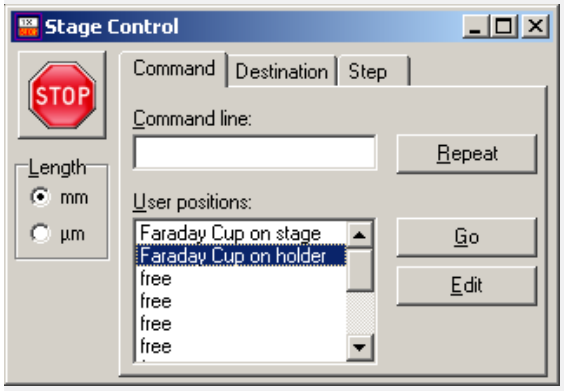


STEP 4 ▶ The full structure is now displayed, showing various line grids as well as an exposure test pattern in the upper left corner including the current date and time.



Task 2 Measuring the beam current

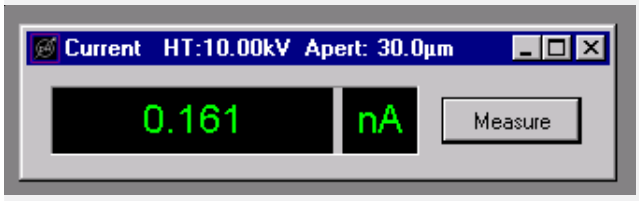
STEP 1 ▶ Open the Stage Control window and drive to one of the Faraday cups. Its position may already be stored as one of the User positions.



STEP 2 ▶ When the stage is at the Faraday cup, toggle the beamblanker to switch on the beam. In the SEM software make sure that the Faraday cup is in the center of the image. If necessary, fine tune the position manually, by using the joystick.

STEP 3 ▶ Ensure that scanning is controlled via the lithography software. The icon must display EXT. This will turn the system into spot-mode, so all electrons will go into the Faraday cup.

STEP 4 ▶ If available, select the current window and measure the current. Otherwise write it down.

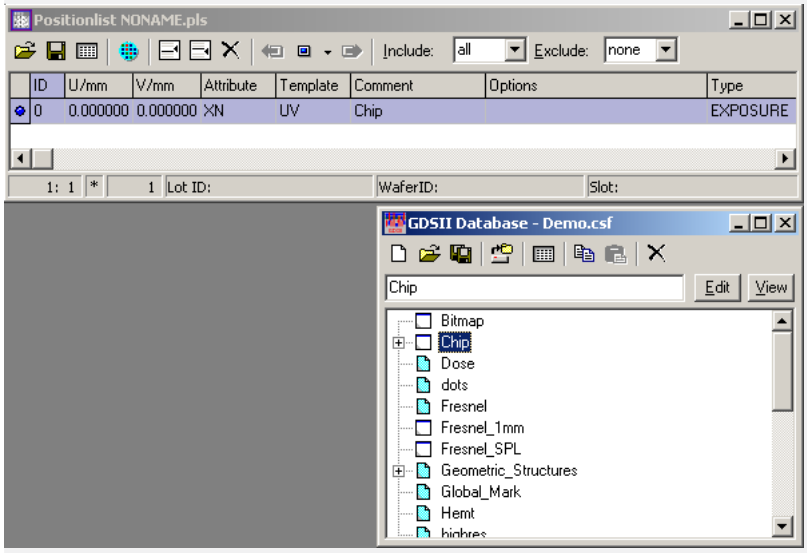


HINT

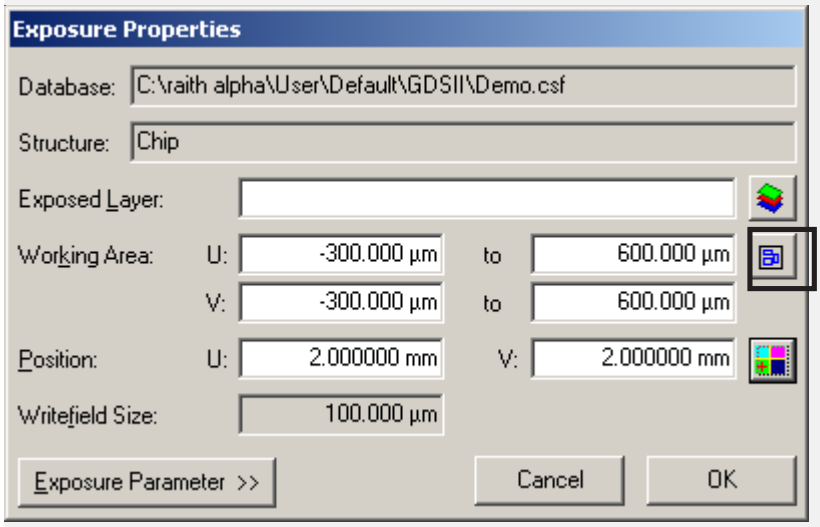
The beam current should be measured just prior to the actual exposure. To avoid complication with defocussing, the beam current should also be measured using the same working distance as the exposure.

Task 3 Exposure

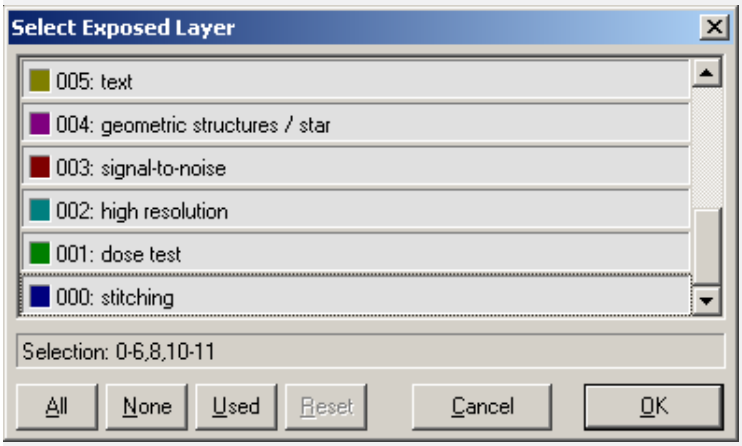
STEP 1 ▶ Open a New Position list. Drag and drop Design Chip into the position list.

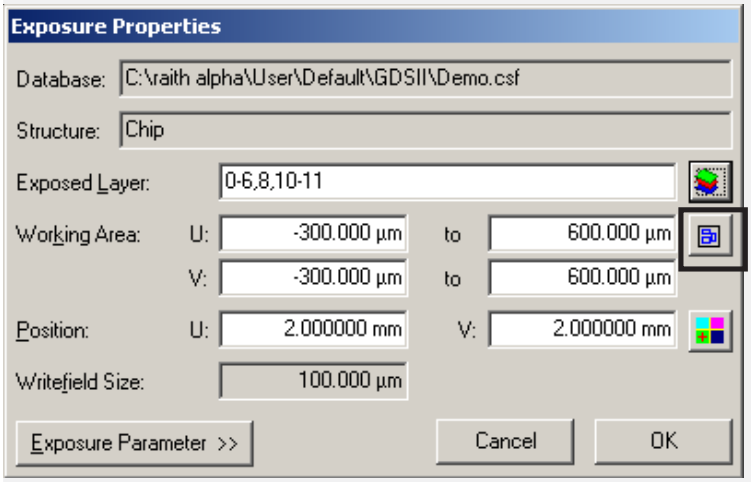


STEP 2 ▶ By default, the exposure is scheduled for the current sample position. The next step is to change the exposure position to the required location. Assuming that your sample has a UV coordinate range between U=V=0 and U=V=10 mm, the first exposure could be set at U=2 and V=2 mm. To set the new UV coordinates, click once with the right hand mouse button at the corresponding line and a cascading menu will be displayed. Click on Properties. Enter the position to U=V=2 mm.



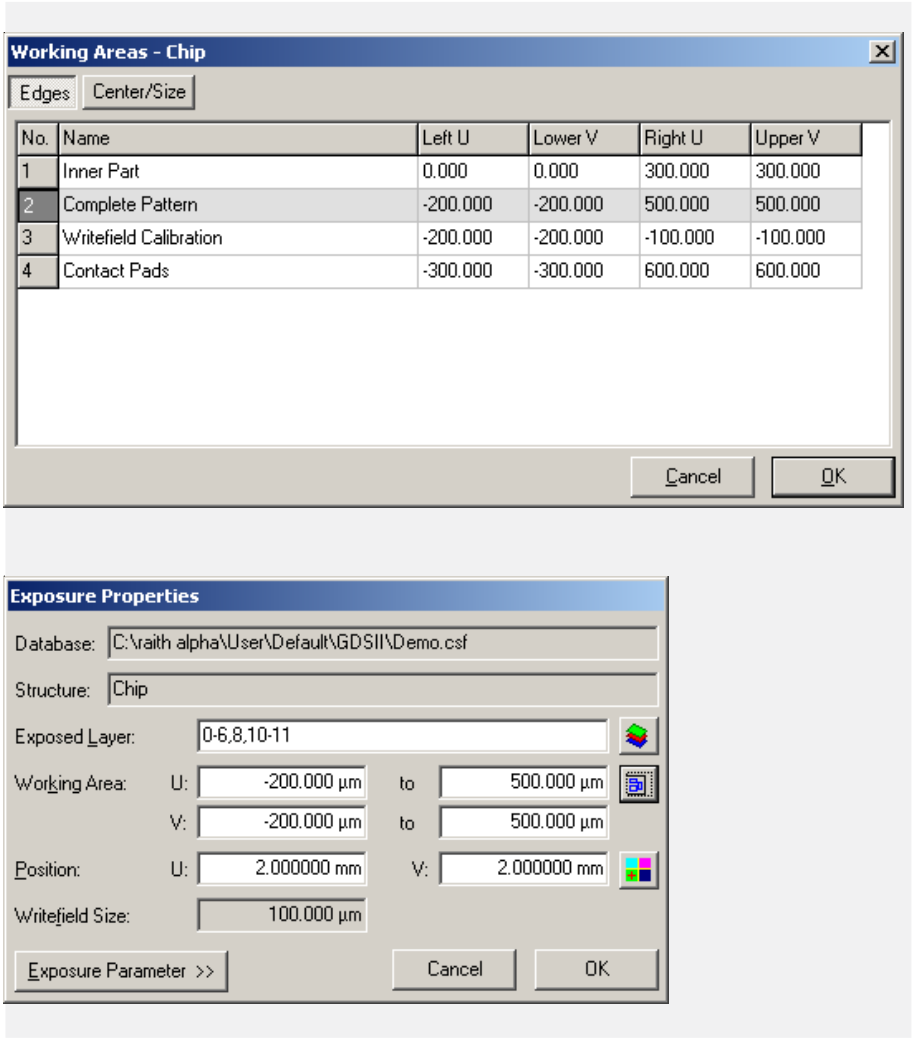
STEP 3 ▶ In the Exposure Properties dialog box, click on the layer button and select layers 0-6 as well as layers 8, 10 and 11. Confirm with OK.





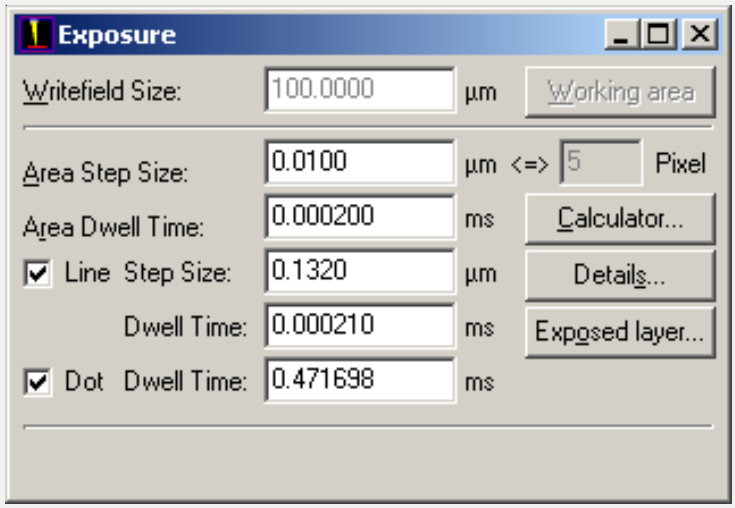
STEP 4 ▶

In the same dialog box click on the Working Area button. Define the working area named Complete Pattern by checking the corresponding row. Confirm both windows with OK.



STEP 5 ▶

Open the Exposure window and check Lines and Dots. Click the Calculator button.



HINT

The Beam Current in the Exposure Parameter Calculation window should be the same value as it was measured before. If measurement is not automated, type in the previously noted value. There is a different tab assigned for Areas, Lines and Dots. At the bottom of the window the formula used for area, line or dot is given. On the right hand side of each parameter a Calculator button is shown in order to recalculate the corresponding parameter.

STEP 6 ▶

Select the Area tab. Enter the Area Dose, which depends on your resist. For example, if you use PMMA, 950 k molecular weight, thickness 100 nm, as provided with the starter kit, and beam voltage of 10 keV, the area dose is 100 μAs/cm^2. Click on the Line tab and enter the corresponding Line dose of 300 pAs/cm. Then click on the Dot tab and enter 0.08 pAs for the Dose.

HINT

After you have entered the appropriate dose the corresponding tab title (Area, Line or Dot) will normally be shown in red. In addition the corresponding formula is shown in red and the OK button is disabled and shown in grey, as the parameters are not consistent anymore.

STEP 7 ▶

Switch back to Area tab and enter the step size of 0.020 μm. Click the Calculator button next to the Dwell time. This will recalculate the corresponding Area dwell time according to the formula shown at the bottom.

HINT

After you have recalculated the Area dwell time, the parameters are consistent and therefore the tab title as well as the formula are now shown in black.

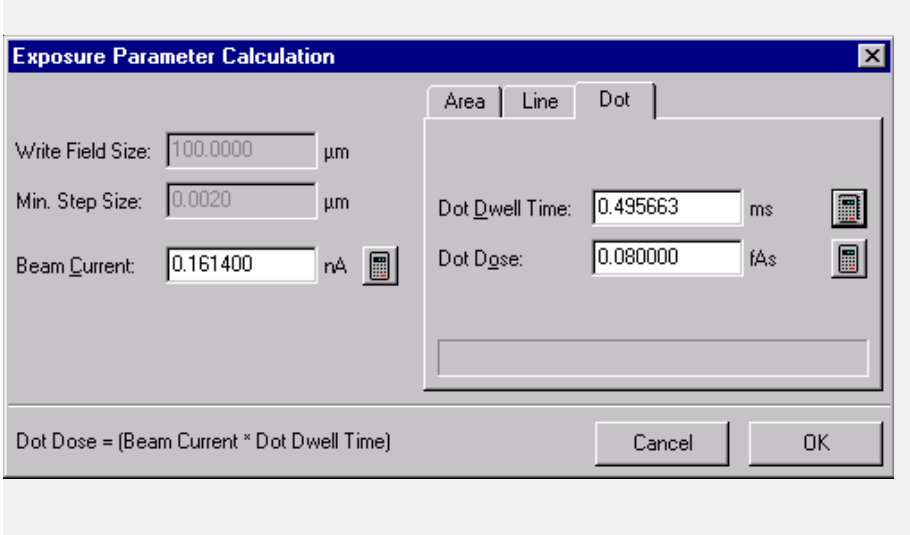
STEP 8 ▶

Select the Line tab and enter 0.010 μm for the Line Step Size and click the Calculator button next to the Dwell time. After the recalculation the tab title as well as the formula will change again to black, as the parameter set is now consistent.

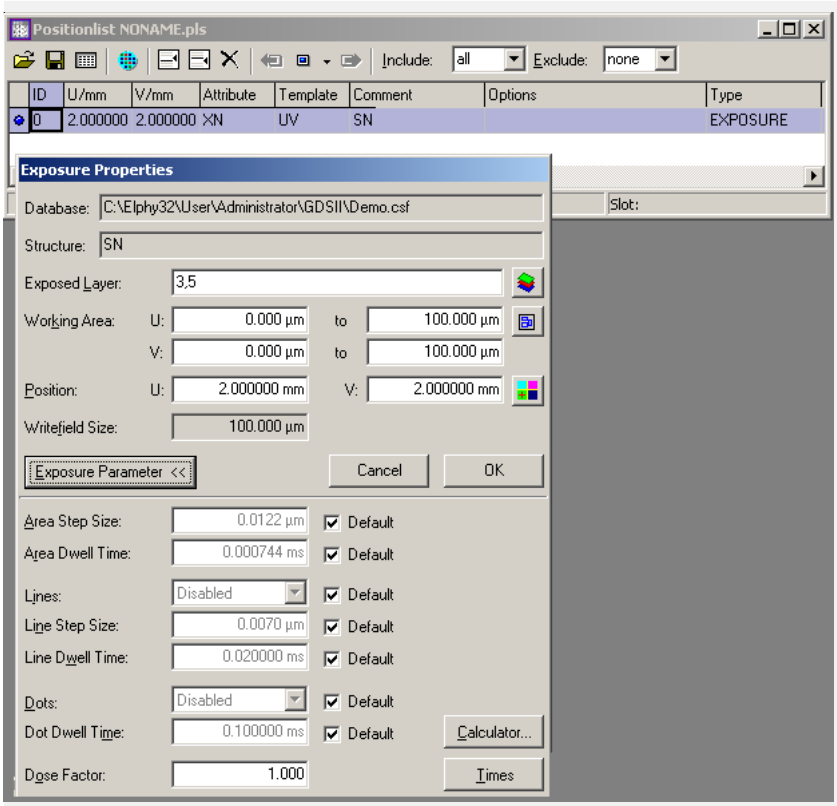
STEP 9 ▶

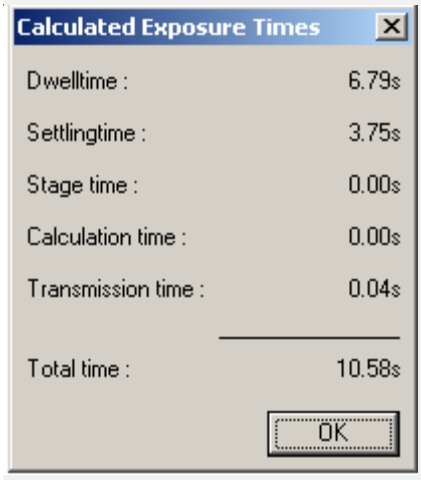
Select the Dot tab. In this case no Step Size is required. Just click the Calculator button next to Dwell time.

STEP 10 ▶ Now all three tab titles (Area, Line and Dot) should be in black and the OK button is now enabled. Click on OK.



STEP 11 ▶ Go to the Position list window. Highlight the corresponding line with the right mouse button, select Properties > Exposure Parameters and click on Times.





STEP 12 ▶ Activate the position list. Go to the Menu Bar > Scan > All. The stage will now drive to the position to expose the pattern.

Task 4 Developing the sample

STEP 1 ▶ Unload the sample.

STEP 2 ▶ Develop the resist according to its type. For example, if you have used the PMMA sample type described earlier, it should be dipped into the developer MIBK:ISO=1 : 3 for 30 seconds and immediately afterward for 15 seconds in pure isopropanol. To ensure a clean surface, the sample should be blown dry using nitrogen.

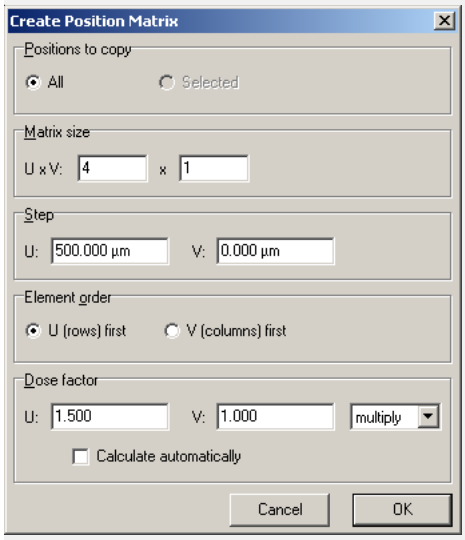
STEP 3 ▶ After you have completed the first inspection using the optical microscope, you can insert the sample into the RAITH system. Perform the stage alignment and address the corresponding sample positions. In our example U = V = 2 mm , for imaging the pattern.

Task 5 Multiple exposure

If the correct dose is known, it is not necessary to follow this task. If the correct dose is not known, a method is explained demonstrating exposure of the same pattern with different doses to reduce the dose parameter range values.

STEP 1 ▶

We will expose the same structures several times with different doses, as it has no dose variation. Highlight the line in the position list, select Filter > Matrix Copy and enter values or Matrix size, step size and dose scaling.



STEP 2 ▶

The structure will be exposed 4 times with a different dose, always increasing by 50%. To check the individual dose factors, highlight the corresponding line with the right mouse button, select Properties > Exposure Parameters.

General Pattern Design

AIM

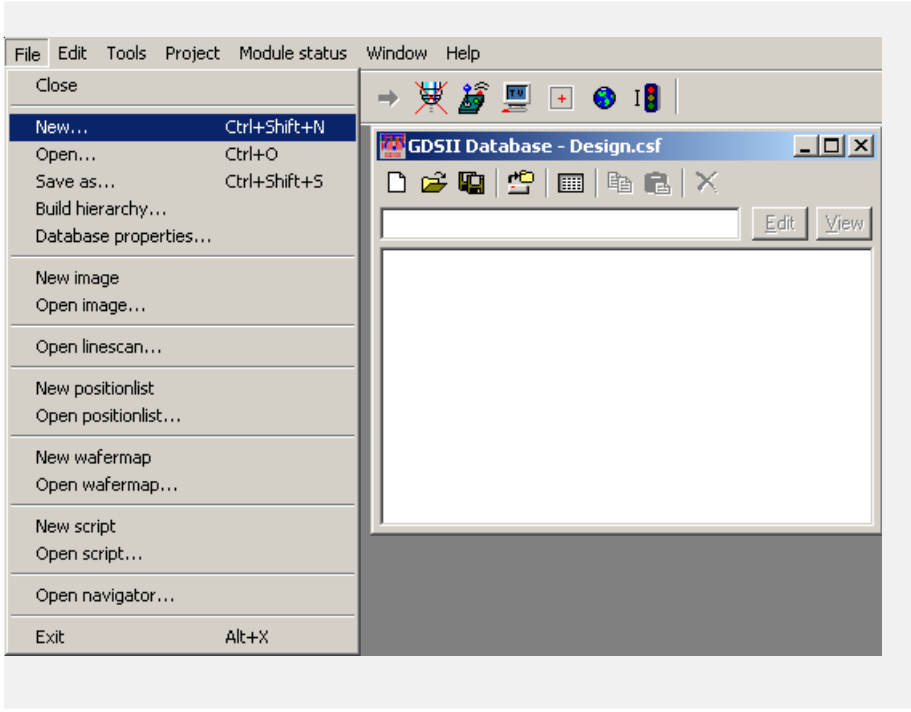
This chapter gives an overview of the different design features by using the internal GDSII editor. It is also possible to import a pattern from other editors such as AutoCAD™, but it is recommended to use the internal editor, mainly because it allows you to assign a different dose to each feature in each GDSII layer.

Task 1 Creating a design

STEP 1 ▶

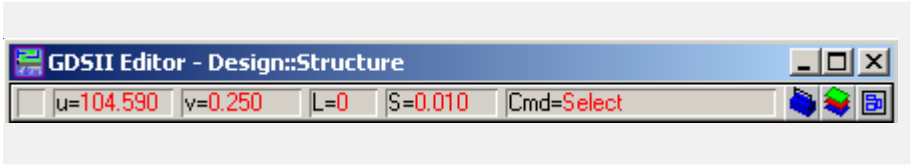
Activate the GDSII database window, click on File > New. A dialog box will open in which you can create a new database within any folder, e.g. with the name "Design". After saving, you will get an empty GDSII Database with the name "Design.csf".

Activate the GDSII database before opening the File menu.



STEP 2 ▶

Choose Edit > New. Now another dialog box will open, in which you can define the name of the first structure, e.g. "Structure1". After confirming, this first structure will appear in the database. At the same time the GDSII Editor will open a default size of 100 µm square.



At the top of the Editor a number of pieces of information are given in the status bar:

*: A star in the first field highlights unconfirmed changes.

UV: The actual coordinates of the cursor position in U,V.

L: The layer chosen for design is displayed. The layer can be changed via Add > Preset > Layer.

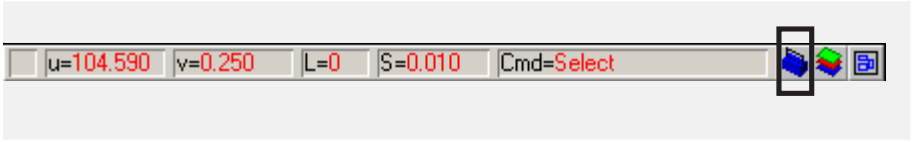
S: The selected step size is displayed. The cursor step size can be changed via / and * button. At the moment the step size is 1 µm, which means that the cursor can only be located at positions with integer microns, leading to a corresponding invisible design grid.

Cmd: The currently used command is displayed. For example, after clicking on Add > Box, the command will show Add box.

Task 2 Pattern design via toolbox

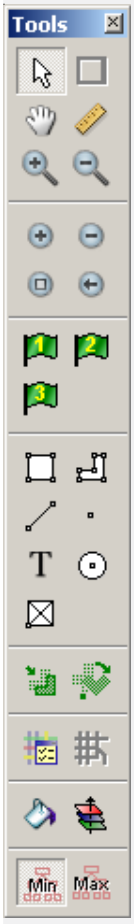
STEP 1 ▶

Open the GDSII Toolbox via the small blue icon in the top right corner of the design field (illustrating a toolbox).



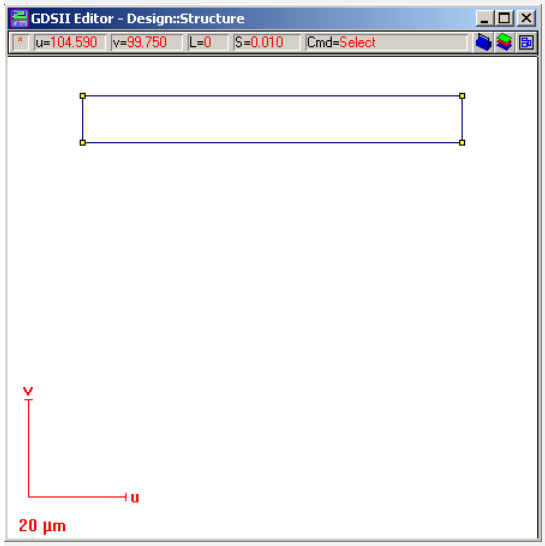
STEP 2 ▶

The icons of the tool box give easy access to the main design functions, which are alternatively accessible via the dropdown list box View > Options > Modify > Add.



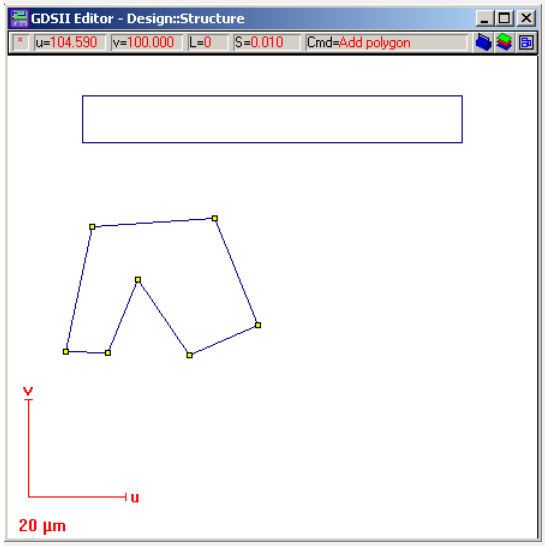
STEP 3 ▶

Select the rectangle icon. Two mouse clicks always define two opposite corners of a rectangle. Once the rectangles are completed, choose the red cross icon or press Esc key to cancel.



STEP 4 ▶

Draw polygons by activating the corresponding icon. Each corner will be defined by a mouse click. During the drawing process the pattern is always displayed by a click of the left mouse, assuming the next mouse click would be the final one. Use the right mouse button or the Return key for the last corner. Double click at a finished polygon allows you to change all parameters digitally.

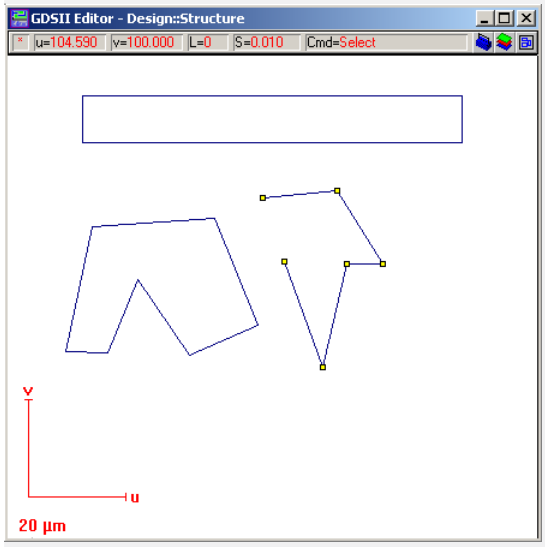


HINT

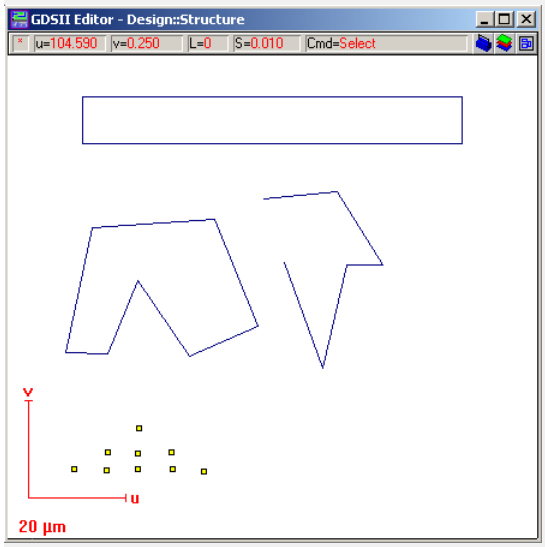
During precise pattern designs you may like to work in a zoomed area. You can zoom in and out during the design by using the + and - key or by using the mouse wheel.

STEP 5 ▶

Draw open paths in the same way. An open path could be a "single pixel line", i.e. no area, or it could have a width defining an area. A double click into any designed structure opens a window with all details. In case of an open path you can change all corner locations digitally, add or delete points, define the dose and the layer and finally you can define the width. A line width of zero defines a single pixel line.



STEP 6 ▶ Place dots after clicking the corresponding icon, one with each mouse click.



STEP 7 ▶ Use all remaining icons of the toolbox to familiarize yourself with the functions. The icons are mostly self explanatory.

STEP 8 ▶ Save the pattern via File > Save and Close. During the work you can use Save or press Ctrl S from time to time. Any unsaved work is highlighted by the red star in the upper left corner of the GDSII Editor window.

HINT

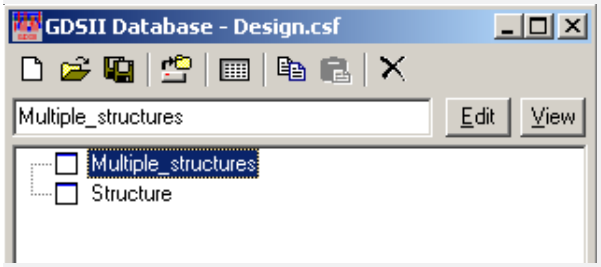
You can Undo/Redo the last changes by using the corresponding commands in the Edit menu or Ctrl Z and Ctrl Y respectively.

Task 3 Modifying structures

STEP 1 ▶

The next step is to create a new structure in the same database called "Test2" (click on GDSII database and then Edit > New) to define the size of the working area. Save and close this window.

To edit a pattern, select it from the list, as shown here for Test2 and then click the Edit button.



STEP 2 ▶

Open the former pattern "Structure1" via the button Edit and click once at the polygon. Once it is selected, the corners are marked by tiny yellow squares.

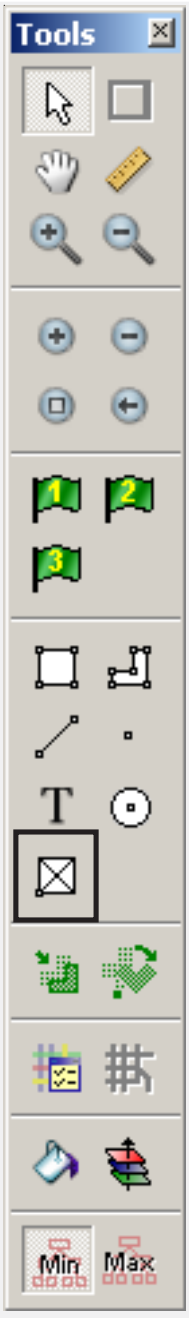
STEP 3 ▶

Choose Edit > Copy. Now open the new design field in the Editor 'Multiply Structures.' Choose Edit > Paste.

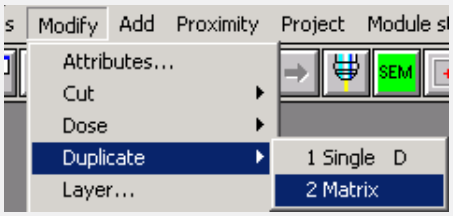
HINT

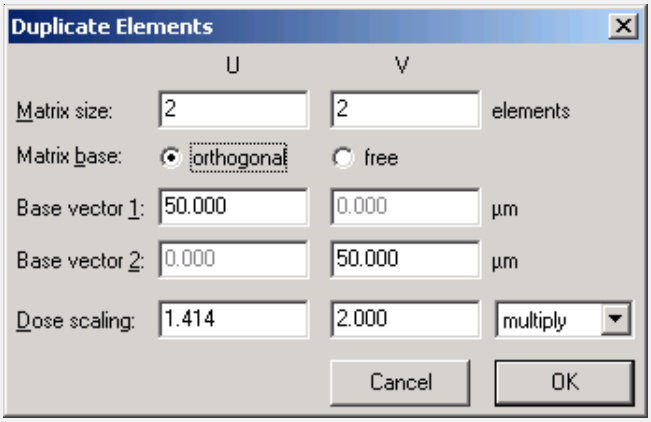
It is also possible to copy groups of elements from one structure into another structure by using Copy and Paste via the menu Edit. To select more than one element, go to Edit, Select or Unselect and choose one of the commands from the cascading menu.

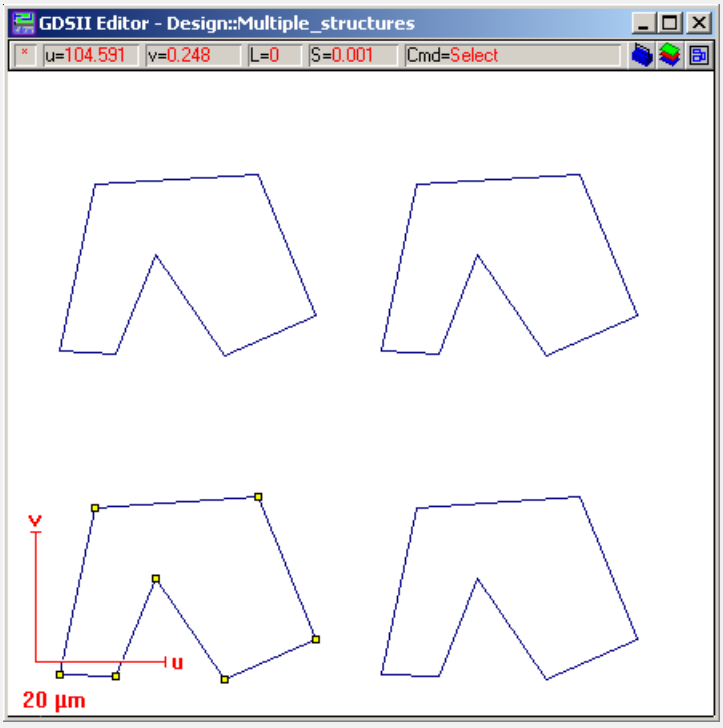
Use the corresponding tool button for moving this structure into the center of the lower left quadrant.



STEP 5 ▶ Choose Modify > Duplicate > Matrix, which will open up the following dialog box. Enter the values as shown.

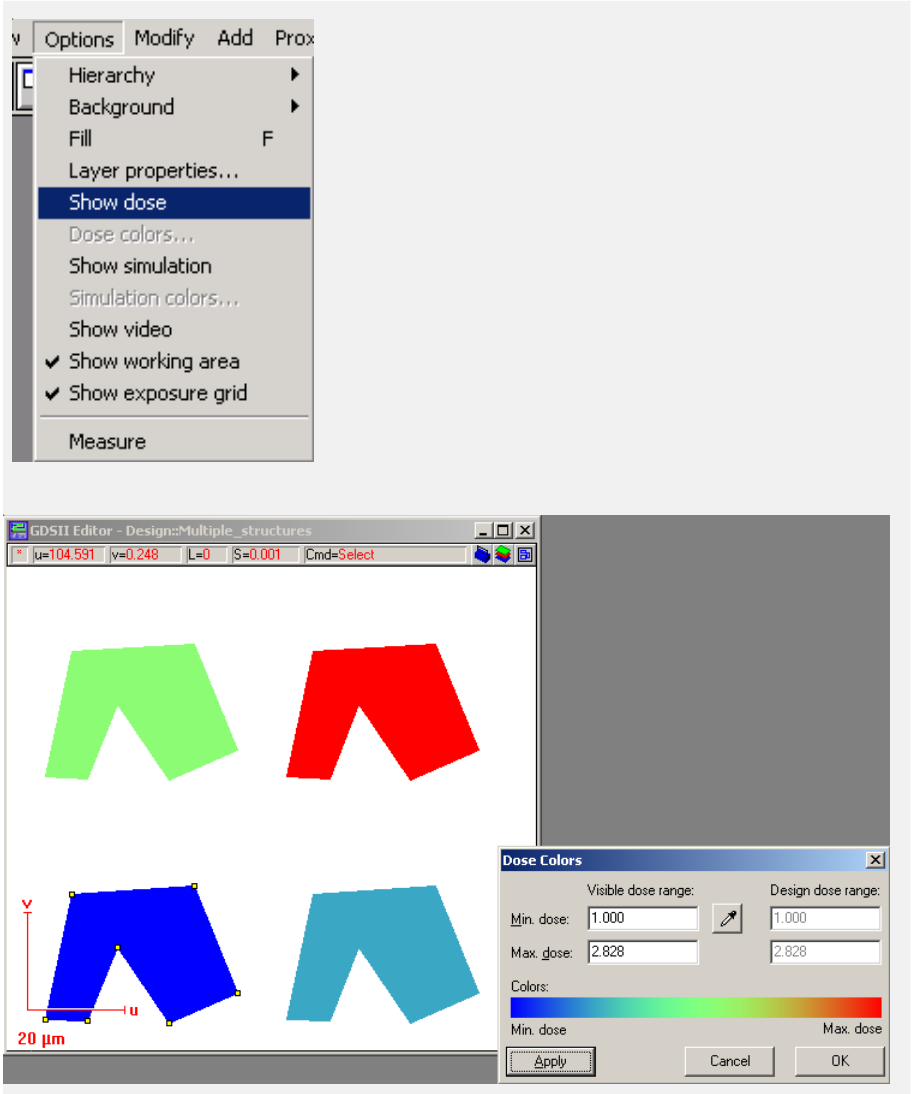






STEP 6 ▶

The result is shown in the figure below. To inspect the dose choose Options > Show dose. You will find that all patterns have the same color. To change the relationship between dose and color, choose Options > Dose colors and a dialog window will open. Choose the pipette icon. This will update the visible dose range. Choose Apply to update the GDSII window and confirm with OK.
The Show Dose option is a useful tool to check the exposure doses prior to the actual exposure test.

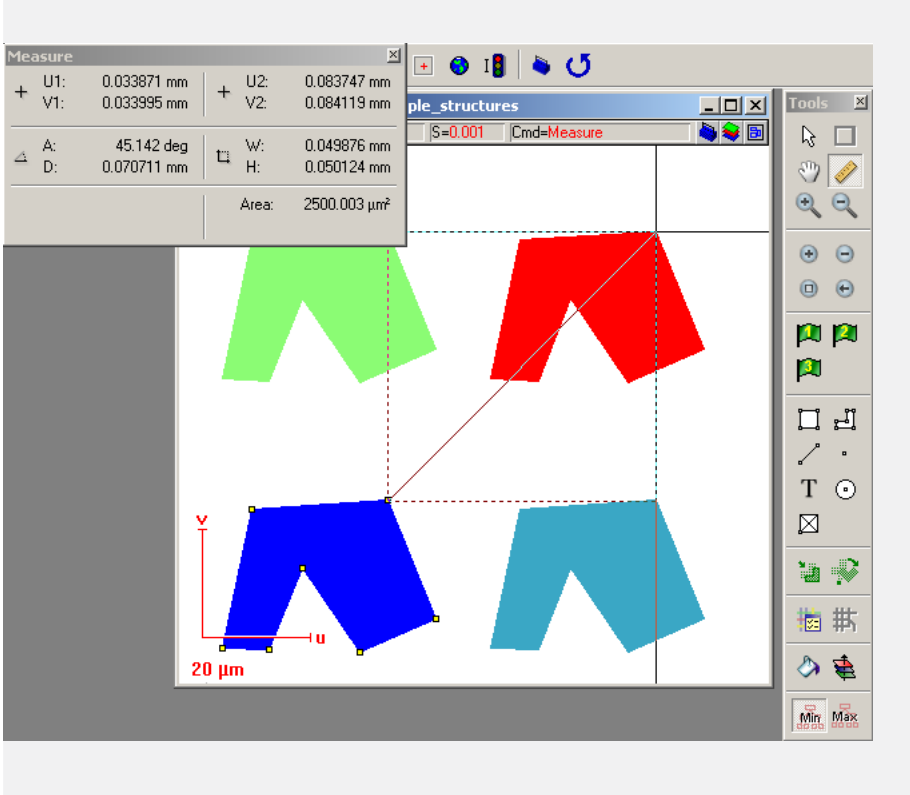


Each individual structure can be edited by a double click, which opens a dialog box, where the UV coordinates, the layer and the dose can be viewed and edited.

Task 4 Measuring a distance

STEP 1 ▶

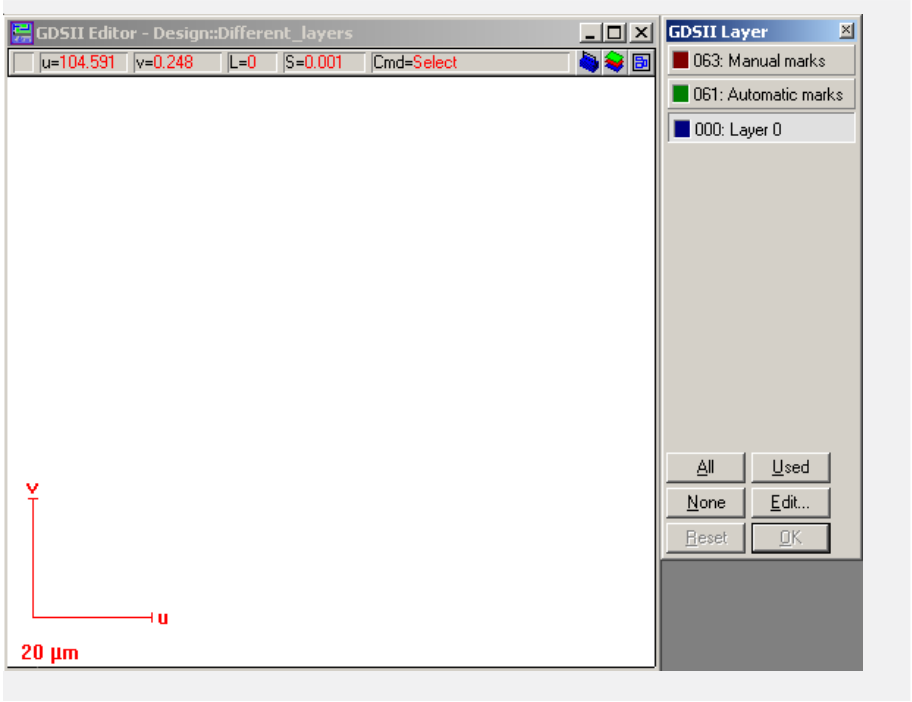
To measure any distance within the design field, click on the corresponding icon in the toolbox and move, while keeping the mouse button pressed, to the other, opposite corner. An information window will appear, in which some dimensions are displayed digitally.



Task 5 Placing of elements in different layers

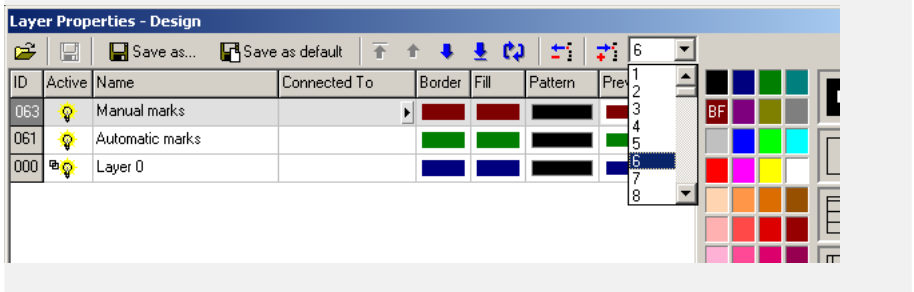
STEP 1 ▶

Create a new structure, we have named it here 'Different' layers. Press the layer icon next to the toolbox. A dialog window will open, showing the existing layers. Press Edit and a new dialog window will open.



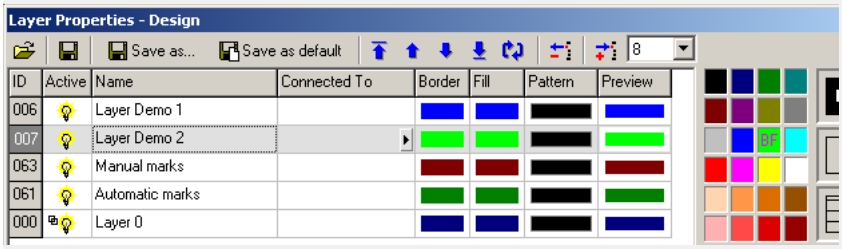
STEP 2 ▶

To add a layer, choose the layer number from the dropdown list box on the right hand side and choose the Add a Layer icon next to it, which will update the table in the Layer Properties window.



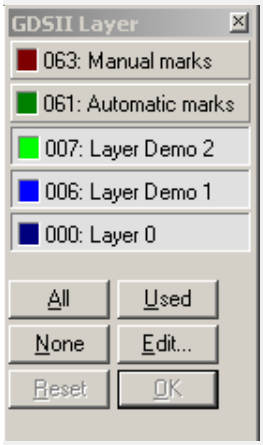
STEP 3 ▶ Enter a name for the new layer, e.g. Layer Demo1. You should now define further properties of this layer. You can change the color of the Border and Fill by moving the mouse to the new color and pressing the left and then right mouse buttons respectively.

STEP 4 ▶ Repeat the last two steps and add Layer 7. In our example we have modified the pattern as shown.



STEP 5 ▶ Choose OK and confirm with Yes to save the changes.

STEP 6 ▶ Make these layers visible by selecting them in the GDSII layer window.

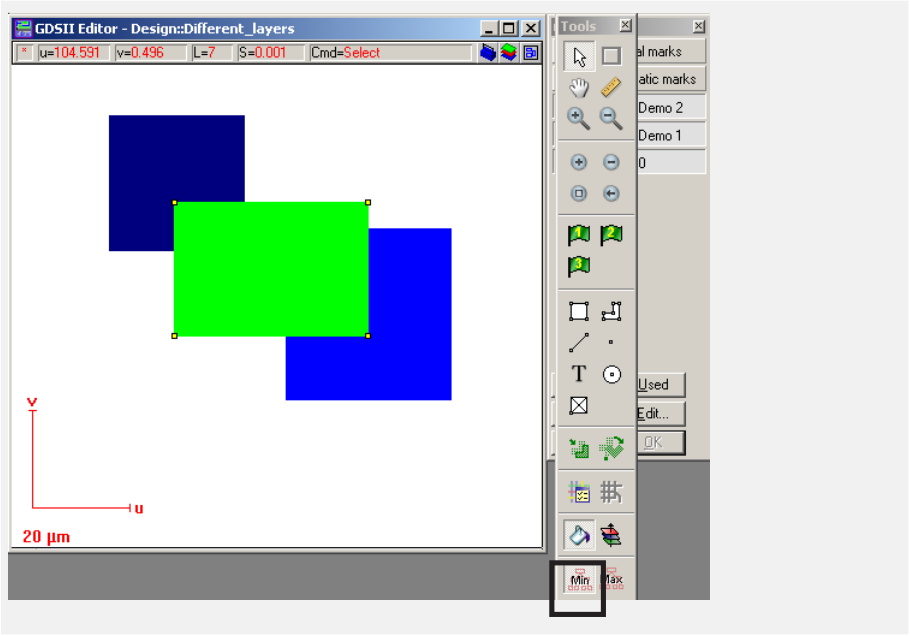


STEP 7 ▶ The active layer is displayed in the top of the Editor window, in our example layer 0. Place a rectangle in this layer.

STEP 8 ▶ Choose Add > Preset > Layer > Show All from the dialog window. Choose Layer 6 and confirm with OK. Layer 6 is now the active layer and will be displayed in the status bar.

STEP 9 ▶ Place another a rectangle in the active layer.

STEP 10 ▶ Make Layer 7 the active layer and place another rectangle in layer 7. Click on the Filling icon in the toolbox to show the result.



STEP 11 ▶ Save and close the test structure.

Task 6 Saving, deleting and copying of structures

STEP 1 ▶ Saving of structures is possible via File > Save or Save and Close as before.

STEP 2 ▶ An existing structure within a database can be deleted while highlighted via Edit > Delete.

STEP 3 ▶ A structure can be copied within the same database while highlighted via Edit > Copy, which is useful for various modifications.

STEP 4 ▶ It is also possible to Rename a structure.

STEP 5 ▶ Sometimes it is also useful to make a copy of the total database, which can be done via File > Save as.

Task 7 Applying varying dose factors

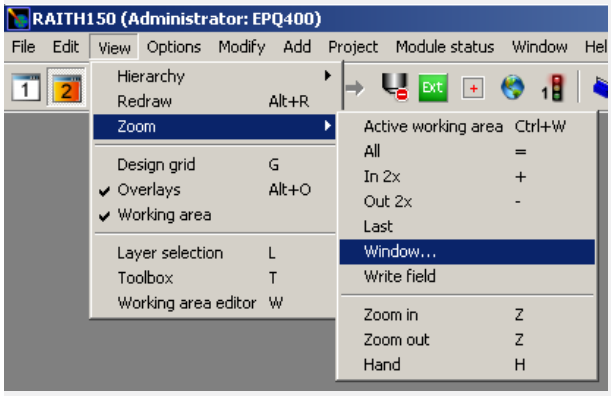
Optimum resolution requires optimum exposure dose. The next steps will explain the design of a resolution test pattern, which will cover a wide range of doses. Please note that a similar structure is already designed and saved within the demo structure.

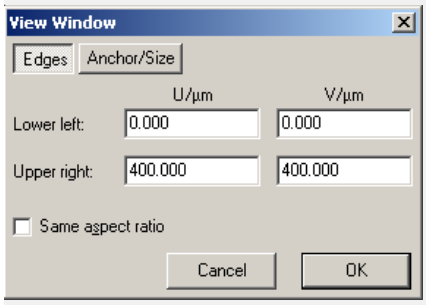
STEP 1 ▶ Select GDSII Database, choose File > New and enter the filename ResTest.

STEP 2 ▶ To create a new structure, choose Edit > New, enter filename RES and click on OK.
The GDSII Editor window opens automatically.

STEP 3 ▶ To select a working area, click on the working area icon. Select a working area of 400 µm for both U and V and save it.

STEP 4 ▶ Choose View > Zoom > Window from the menu bar.

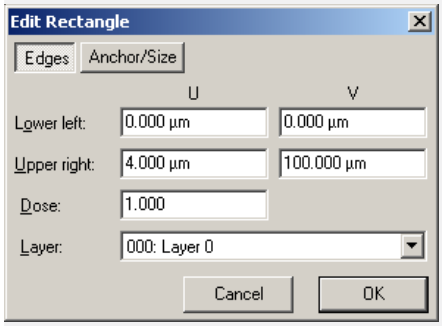




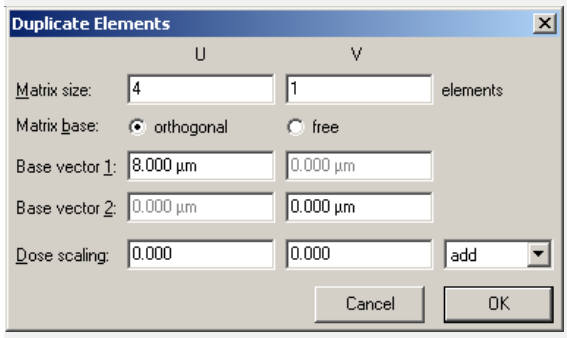
STEP 5 ▶ Select the rectangle icon in the toolbox and draw one rectangle.

STEP 6 ▶ Cancel the repeating command by pressing the Esc key.

STEP 7 ▶ Double click inside the square to edit the parameters. Enter the coordinates 0 and 4 for U and 0 and 100 for V. This creates a rectangle with a length of 100 µm and a width of 4 µm.



STEP 8 ▶ To create a matrix from this rectangle, we need to design four rectangles with 4 µm widths. Choose Modify > Duplicate > Matrix from the menu bar. Enter 4 for Matrix size U and 1 for V, as we only want to duplicate the structure in U direction. Choose the matrix base orthogonal. Enter a stepsize of 8 µm for base vector 1.

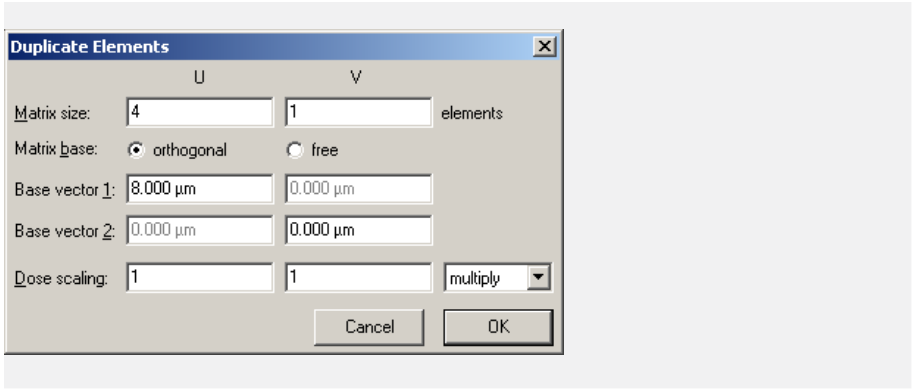


STEP 9 ▶

The rectangle has now been repeated 4 times leading to a grid of equal rectangles and spaces with 4 µm width.

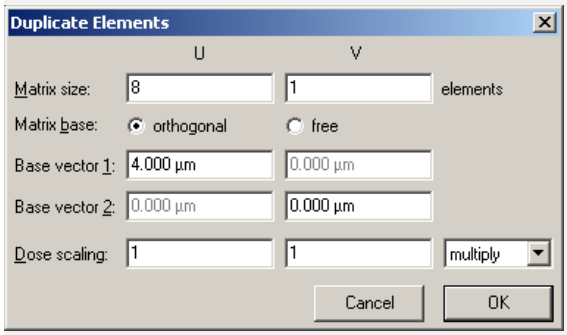
STEP 10 ▶

Select the rectangle icon in the GDSII toolbox and draw another rectangle.
Click on the red cross icon (or Esc key) to cancel the repeat command.
Double click inside the rectangle and enter the following coordinates:
U 0 and 2 µm
V 150 and 250 µm
layer 0 and dose 1.
Click on OK.



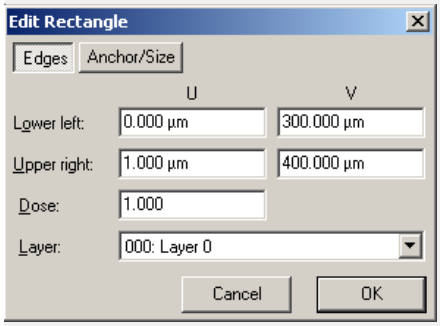
STEP 11 ▶

Choose Modify > Duplicate > Matrix. Matrix size is 8 for U and 1 for V, stepsize 4 for U and dose scaling is 1.
The width of the lines as well as the distance between them is now only half compared to the previous grid.



STEP 12 ▶

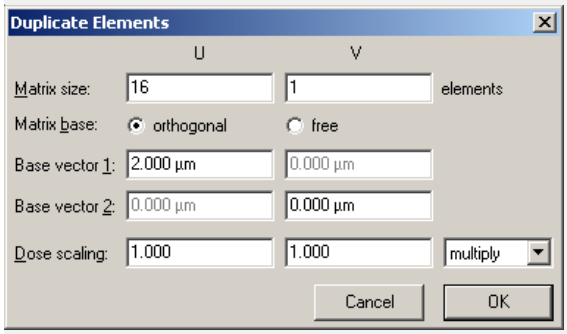
Select the rectangle icon in the GDSII toolbox and draw another rectangle.
Click on the red cross icon (or Esc key) to cancel the repeat command.
Double click inside the rectangle and enter the following coordinates:
U 0 and 1 µm
V 300 and 400 µm
layer 0 and dose 1.



STEP 13 ▶

Choose Modify > Duplicate > Matrix. Matrix size is 16 for U and 1 for V, stepsize 2 for U and dose scaling is 1.
The periodicity of the grid is now only half compared to the previous grid and only a quarter, compared to the first grid.

We have now designed three grids each with rectangles of equal width and spaces. In each of the three grids the width of the rectangles and gaps has been selected to be 4 µm, 2 µm and 1 µm respectively.

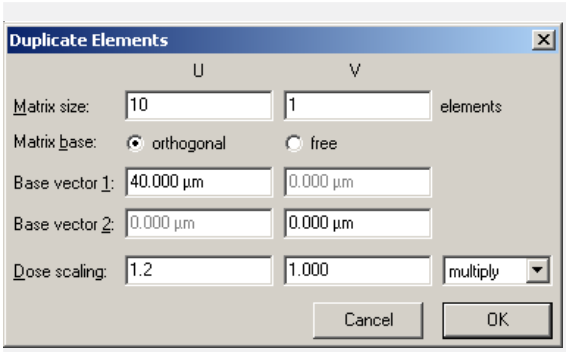


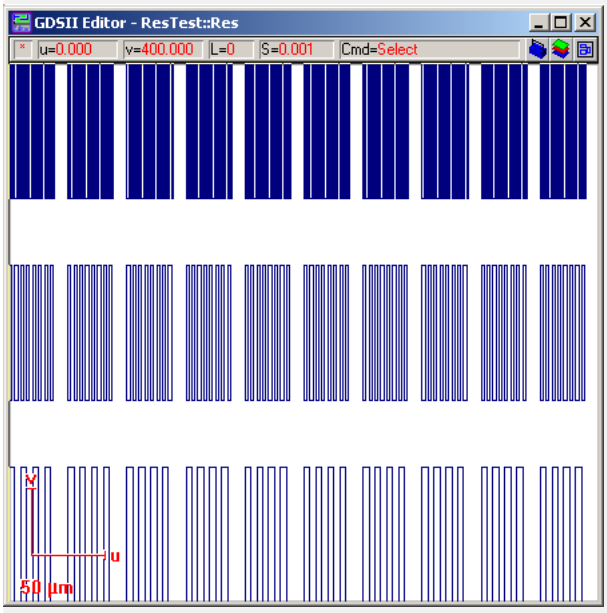
STEP 14 ▶

Choose Edit > Select > All, from the menu bar.

STEP 15 ▶

Choose Modify > Duplicate > Matrix. Matrix size is 10 for U and 1 for V, stepsize 40 for U and 1 for V, dose scaling is 1.2 and select multiply. Click on OK.



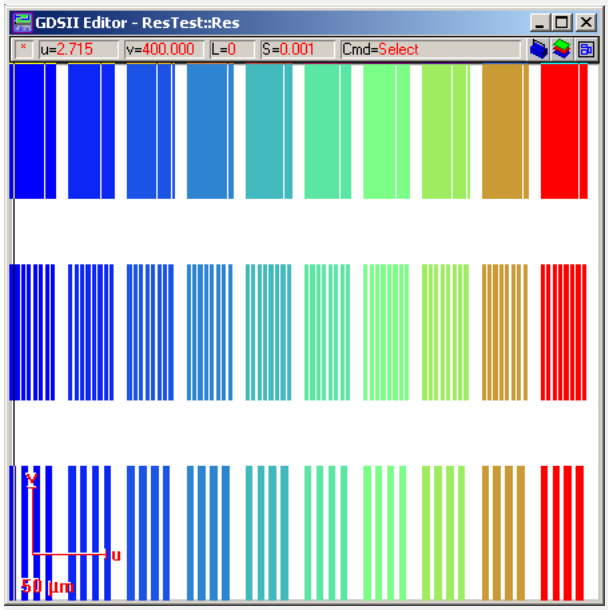


The line structure has now been duplicated, filling the complete write field. The different spacing from row to row can easily be observed.

STEP 16 ▶

Choose Options > Show Dose. The doses applied are now displayed in different color codings. Choose Options > Dose Colors and update using the pipette icon, then select Apply.
The design of the resolution pattern is now completed within a 400 µm field.

STEP 17 ▶ Choose File > Save and Close.



Advanced Pattern Design

AIM

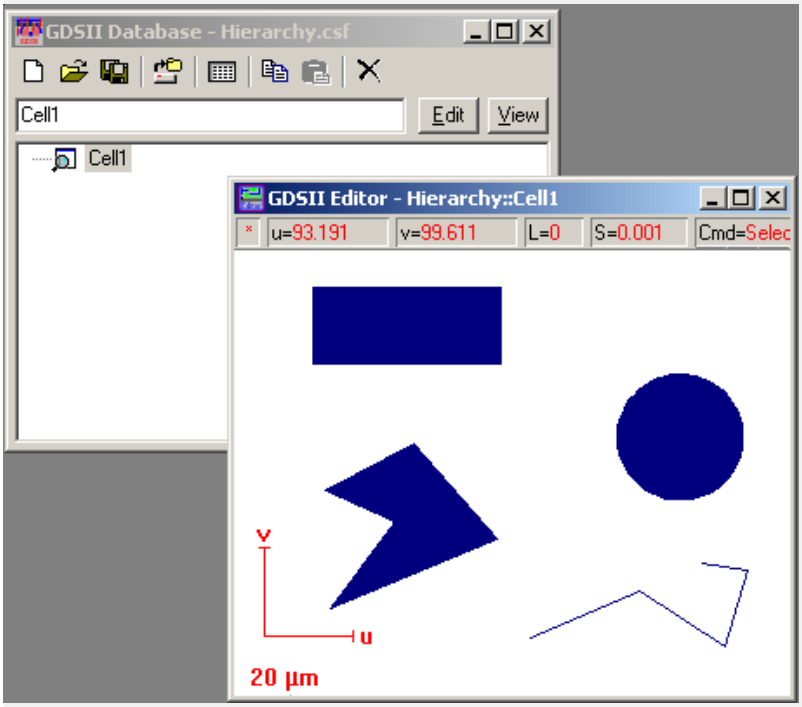
In the previous chapter, we learned how to multiply structures within a matrix when each structure could be assigned another dose. This method can lead to patterns of a large file size. Using the hierarchy function, the pattern file size will remain small and it also simplifies the creation of multiple structures.

Task 1 Design using hierarchy
Task 2 Studying chessy.csf

Task 1 Design using hierarchy

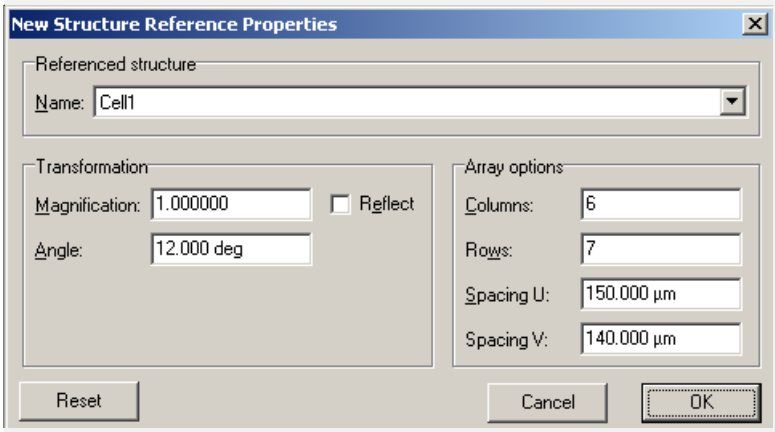
STEP 1 ▶

Create a new database and name it Hierarchy. Design one structure and name it Cell1. The GDSII Editor will now open automatically, so that you can place several structures within the field of approximately. 100 µm. Save the structure and close the Editor.



STEP 2 ▶

Create a structure Matrix1. From the menu, choose Add > Structure Reference. The following dialog window will be displayed.

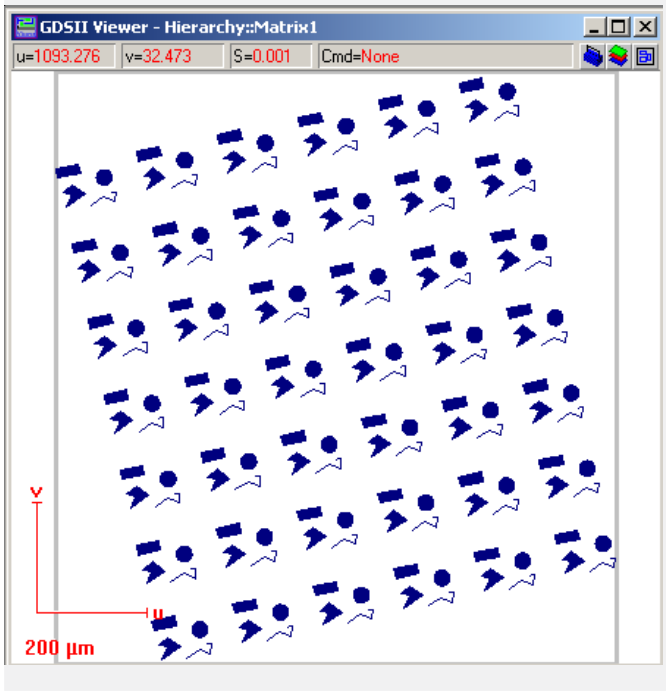


STEP 3 ▶

Select from Name, one of the structures listed. In our example there is only Cell1 available. At the bottom of the window you can enter the Magnification, Angle, Column, Rows as well as the Spacing in U and V. Once you have entered all parameters, as shown in the example, click on OK to create a new Structure Reference. You can now place this new structure anywhere in the pattern. Press Escape to place the structure only once. Save the structure and then close it. Reopen the structure Cell2 using the GDSII viewer.

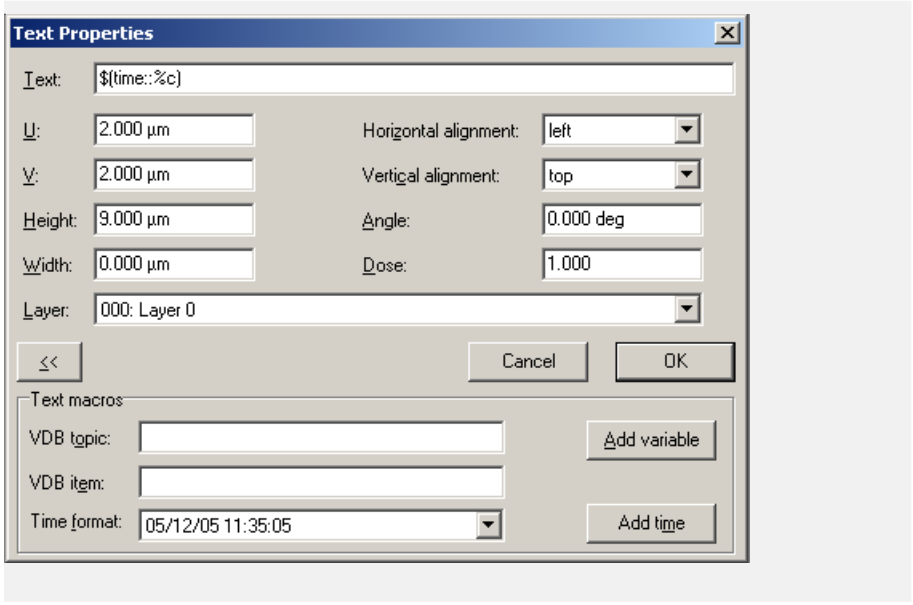
STEP 4 ▶

By opening the pattern again via the GDSII viewer, no pattern will be displayed. Instead it shows a red box with the name Cell1[6][7]. This naming structure indicates that Cell1 has been repeated in 6 columns and 7 rows. To view the full pattern, go to View > Hierarchy and select level 1 or higher. A structure similar to the figure will be shown.



STEP 5 ▶

Create a new structure within the GDSII database with the name Time. Go to Add and select Text. You can now insert any text. By pressing >>, additional parameters are available. You can call current variables such as the time or any other variables from the VDB files used for the current exposure. Select Time format and press Add time button, which will display a special command string in the text field.

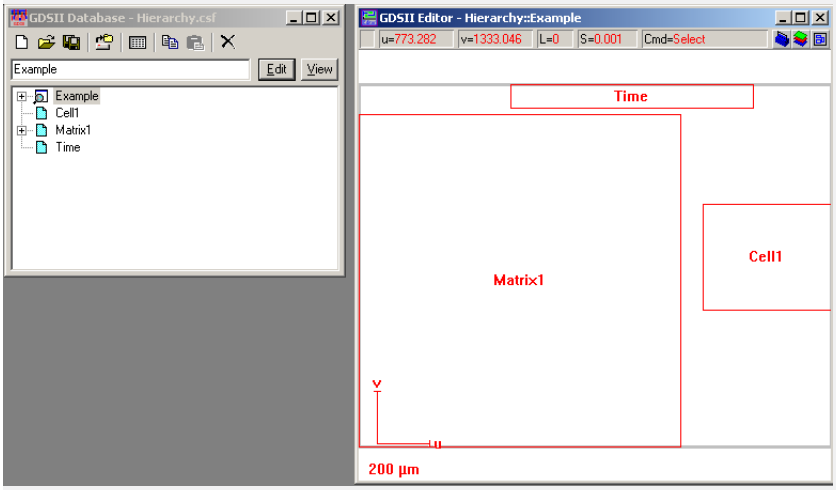


STEP 6 ▶

There is a wider variety of command strings available for other formats or variables, which are described in more detail in the manual. In addition you can enter further Parameters for the Text such as the Position in U and V, the Layer, the Height, Width and Dose. After you have entered your parameters, click on OK and the current time will be displayed. Save the structure and close the Editor.

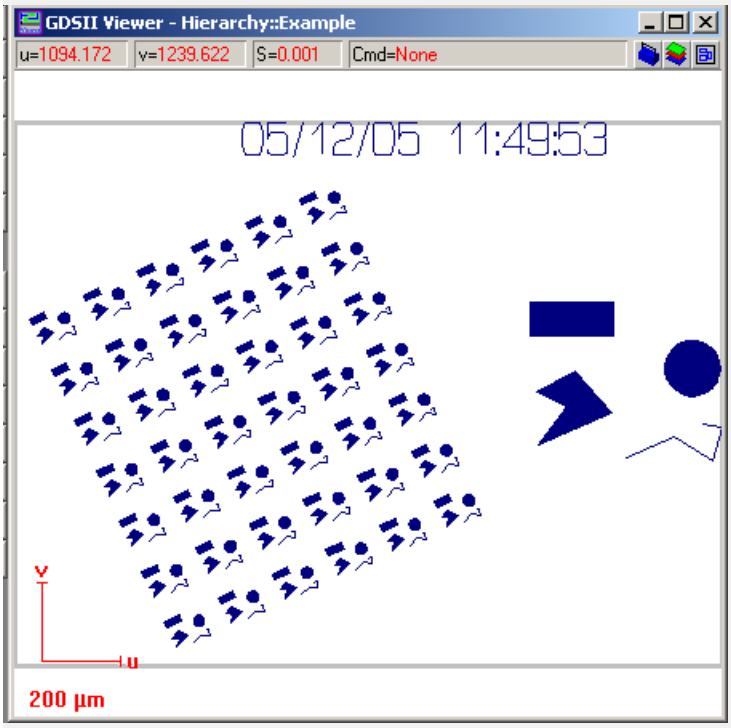
STEP 7 ▶

Create a new structure with the name ‘Example’. In this structure we will insert the structures designed earlier.



STEP 8 ▶

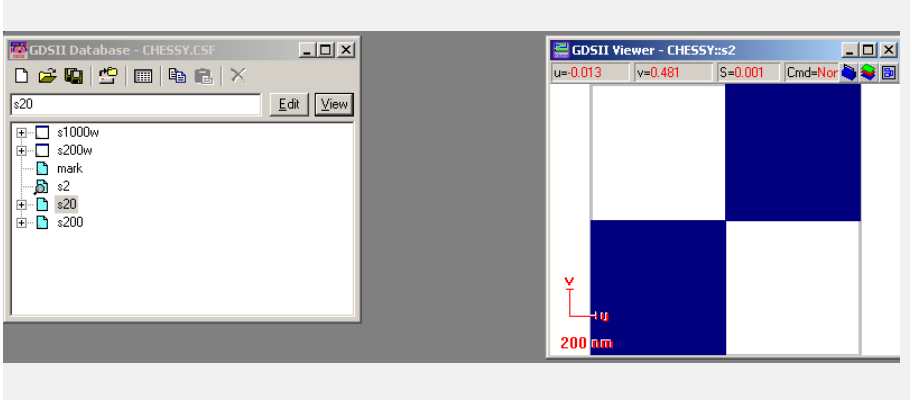
Click on Add and Structure Reference and insert ‘Matrix1’, ‘Cell1’ (5 times enlarged) and Time (10 times enlarged) within the structure ‘Example’. Make sure to set Columns and Rows to 1. Save and close the structure and reopen it with the GDSII Viewer in order to resolve the hierarchy. Select hierarchy level 2 or higher to resolve the pattern containing the current date and time.



Task 2 Studying chessy.csf

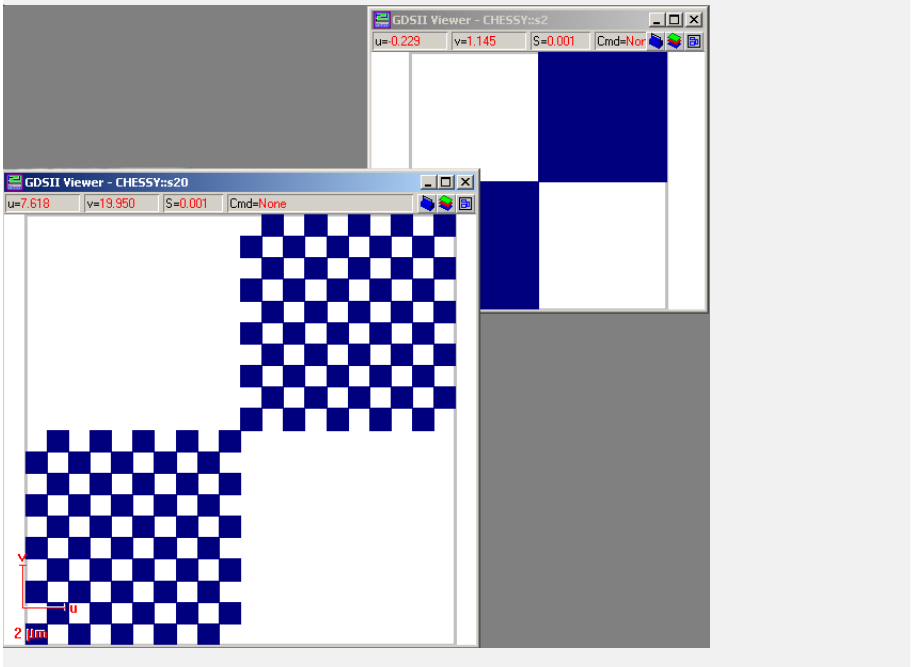
STEP 1 ▶

Chessy is an ideal example to study the design at various hierarchy levels. Open the structure S2 using the GDSII Viewer and select Fill from the Menu options. The GDSII Viewer will now display the design within a 2 µm field covering just 2 squares of 1 µm size.



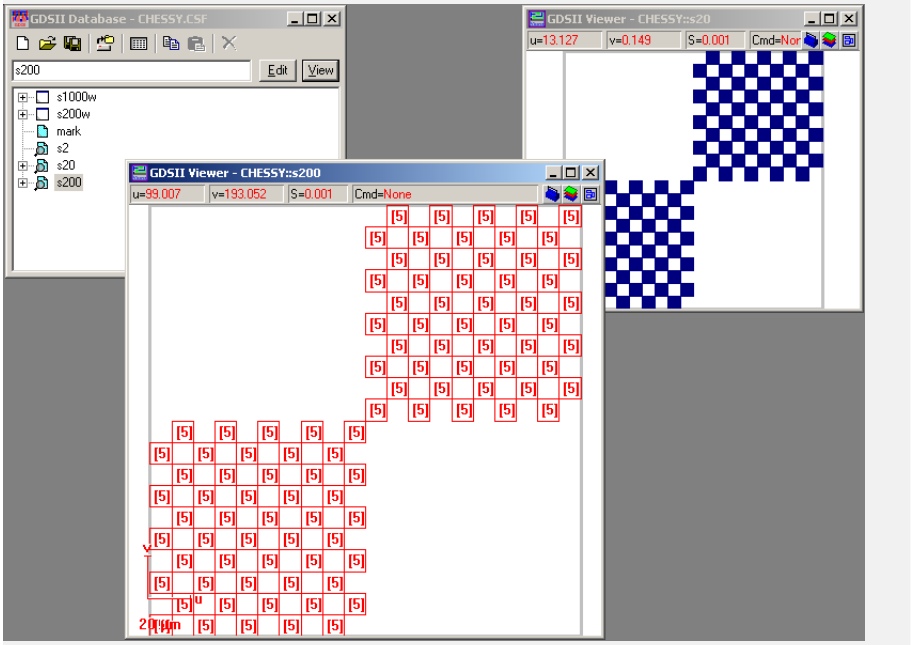
STEP 2 ▶

Now open the pattern S20 using the GDSII Viewer and select Fill. This pattern shows the next hierarchy level, where two matrices are shown. Each matrix contains a 5x5 pattern S2. Select the hierarchy level 1 to resolve the pattern in order to view the single squares.



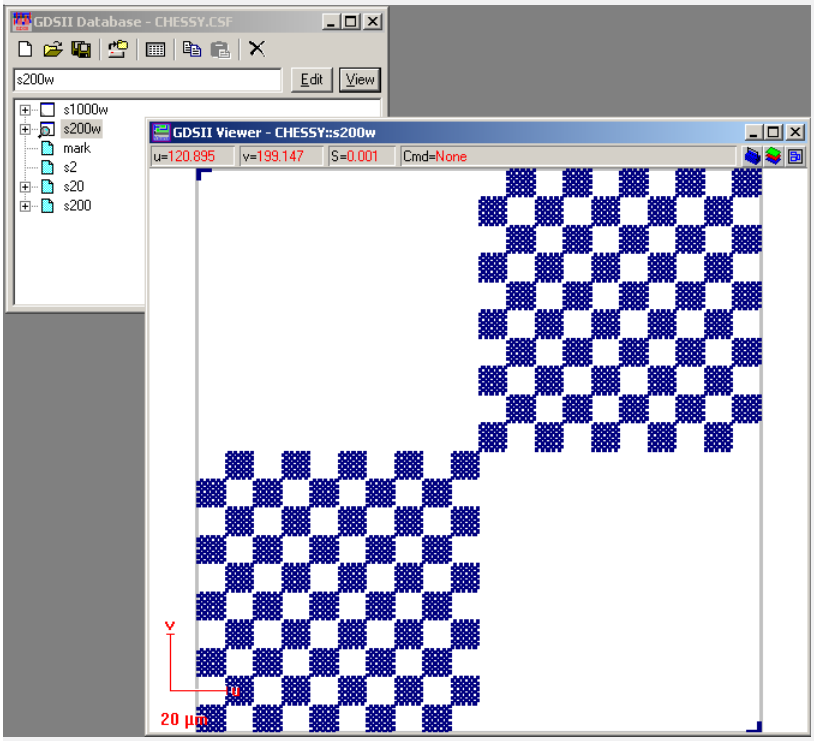
STEP 3 ▶

Now open the pattern S200, this will fill a writing field of 200 µm. Two matrices are shown, each containing a 5x5 S20 pattern. If you select the hierarchy level 1, only the S2 matrices are shown as displayed in the figure below. In order to resolve the single square, you now need to select hierarchy level 2.



STEP 4 ▶

Now open the pattern Mark, it consists of just 2 rectangles forming an L-shape. Within the pattern S200w there are structure references to S200 and two references to Mark. One mark has been rotated by 180 degrees before it was defined as the structure reference. Pattern S200w is shown below. It can only be resolved by a hierarchy level of 3 or higher.



Hint

The same process of hierarchy levels design can be continued from one hierarchy level to the next. For example, the pattern S1000w includes already 125,000 squares. Whereas the total database Chessy.csf, which utilizes a hierarchical design, has a file size of only 1 KB, the same structures without hierarchy levels would require approximately 9 MB.

Mix & Match Exposure

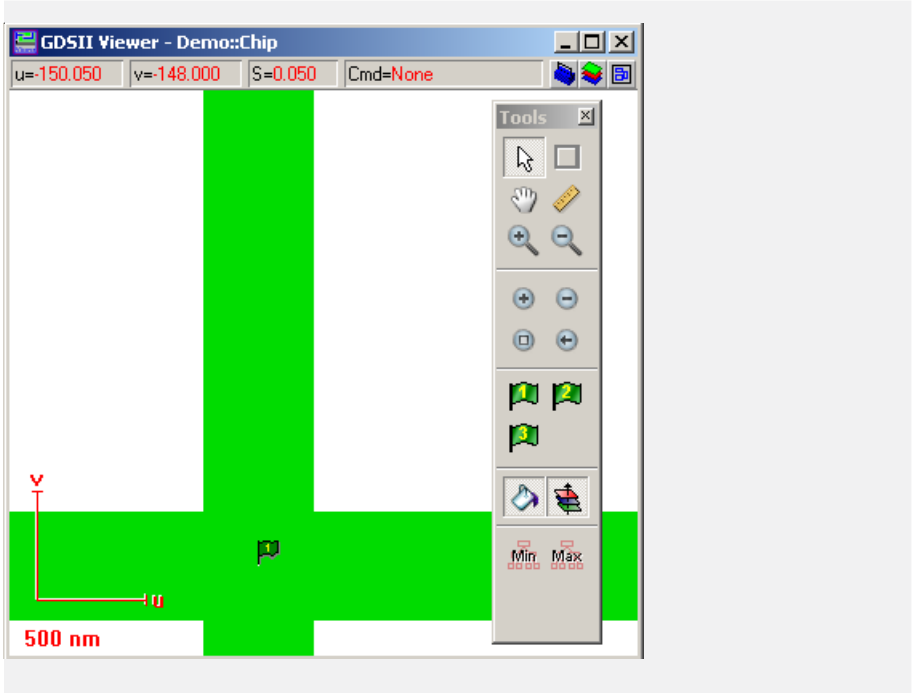
AIM

The aim of this tutorial is to perform a Mix & Match Exposure. In a Mix & Match procedure, a second lithography step is placed into an existing pattern.

Task 1 Defining UV positions of marks

STEP 1 ▶

Open the GDSII Viewer with your corresponding pattern. In our example, open 'Demo.csf' and 'Chip'. Locate mark 1 within your pattern. In our example the mark is located at U=V=-150 µm. Open the toolbar, by pressing the letter T on the keyboard. Drag and drop the green flag 1 onto your mark 1.

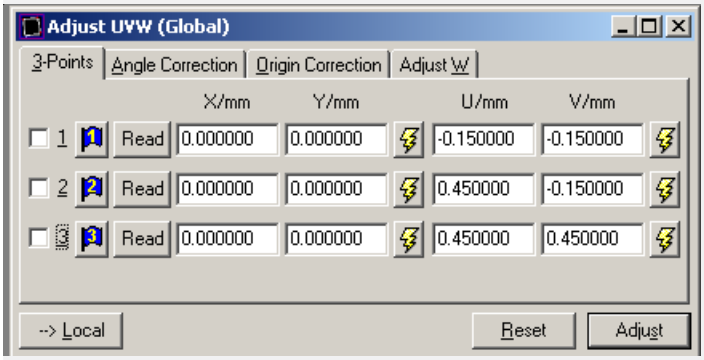


The UV coordinates for mark 1 will now be displayed in the Adjust UVW window.

STEP 2 ▶

Repeat the same procedure for marks 2 and 3. In our example, mark 2 is located at U=450 µm and V= -150 µm and mark 3 is located at U=V=450 µm.

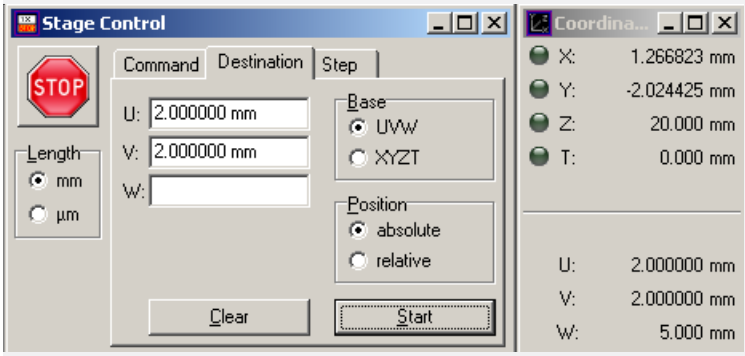
STEP 3 ▶ Uncheck all three positions.



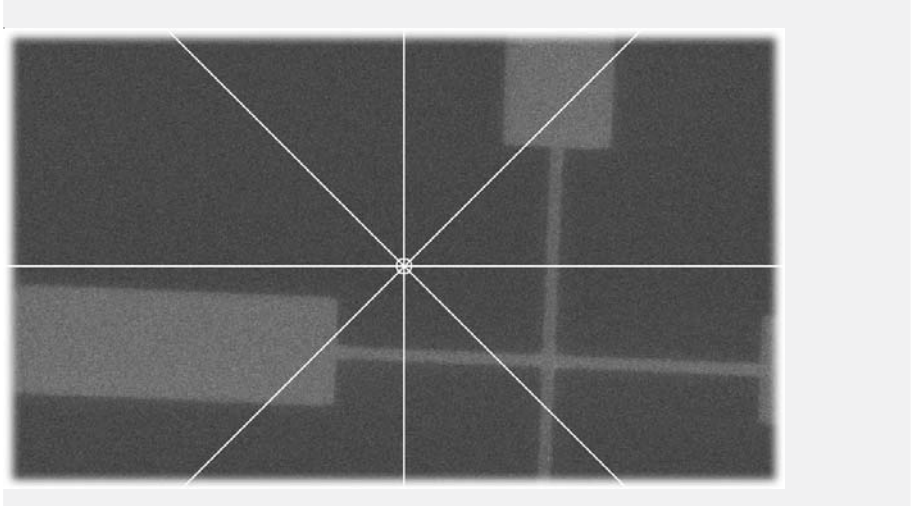
Task 2 Locating the first mark

It is assumed that you have already followed the first few chapters, including the chapter 'Exposure'. After developing the sample, load the sample into your system again and perform the steps described in the chapter, 'Stage Adjustment'

STEP 1 ▶ In order to find the first mark, open the Stage Control window and enter the value 2 for U and V. Click on Start.



STEP 2 ▶ On the column desktop select a magnification of 3000x. Switch on a crosshair and unblank the beam. The first mark should now be visible.

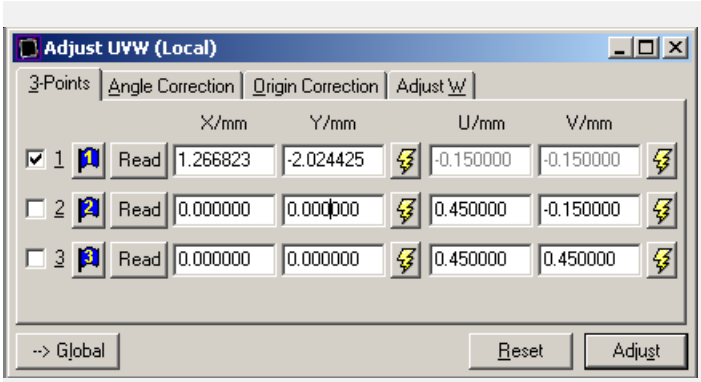


Using the joystick, move the crosshair over the mark and switch off the beam. The next task is to perform a 3-Point Adjustment.

Task 3 3-point adjustment

STEP 1 ▶ Open the Adjust UVW window and select the tab, 3-Points. Switch to the Local coordinator transformation.

STEP 2 ▶ Choose Read to update the current XY coordinates of the first mark position. Activate checkbox 1 of the same position.



In this step, we have performed, in principle, an origin correction, this means that the origin of the coordinates system has been redefined.

STEP 3 ▶ Click on the lightning icon or mark 2 related to the UV coordinates. This will move the stage to the second mark.

STEP 4 ▶ Select a high magnification again, (approximately 3000x) and switch on the beam. Move the second marker so that the crosshair is situated above the mark.

Click on Read of mark 2. The XY coordinates in the Adjust UVW window will be updated.

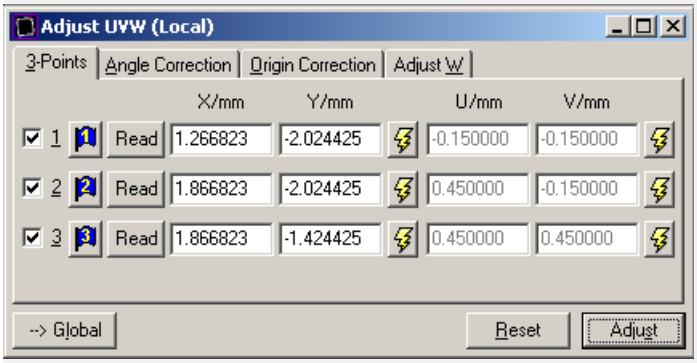
Click the checkbox of mark 2 and click Adjust. Please note that the UV coordinates have been updated after the adjustment has been performed.

STEP 5 ▶ Click on the lightning icon of mark 3 of the UV coordinate system to move the stage to mark 3.

STEP 6 ▶

Make sure that a high magnification, (approximately 3000x) has been selected and switch on the beam. Move the third mark so that the crosshair is situated above the mark. Click on Read of mark 3. The XY coordinates will be updated.

Check mark 3 and click Adjust.



Mix & Match Exposure

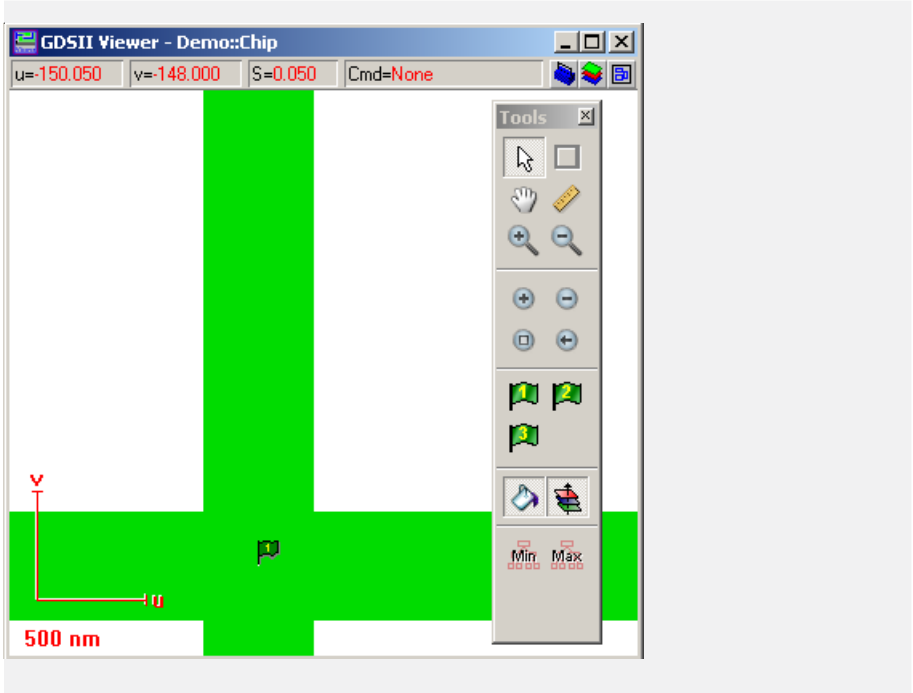
AIM

The aim of this tutorial is to perform a Mix & Match Exposure. In a Mix & Match procedure, a second lithography step is placed into an existing pattern.

Task 1 Defining UV positions of marks

STEP 1 ▶

Open the GDSII Viewer with your corresponding pattern. In our example, open 'Demo.csf' and 'Chip'. Locate mark 1 within your pattern. In our example the mark is located at U=V=-150 µm. Open the toolbar, by pressing the letter T on the keyboard. Drag and drop the green flag 1 onto your mark 1.

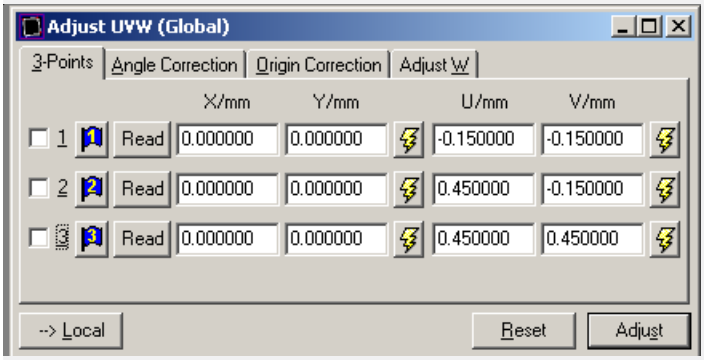


The UV coordinates for mark 1 will now be displayed in the Adjust UVW window.

STEP 2 ▶

Repeat the same procedure for marks 2 and 3. In our example, mark 2 is located at U=450 µm and V= -150 µm and mark 3 is located at U=V=450 µm.

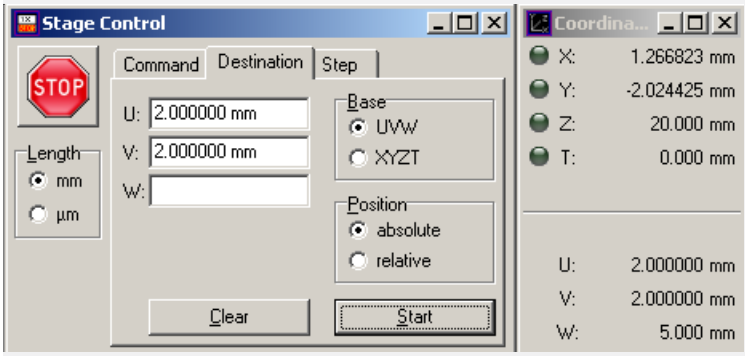
STEP 3 ▶ Uncheck all three positions.



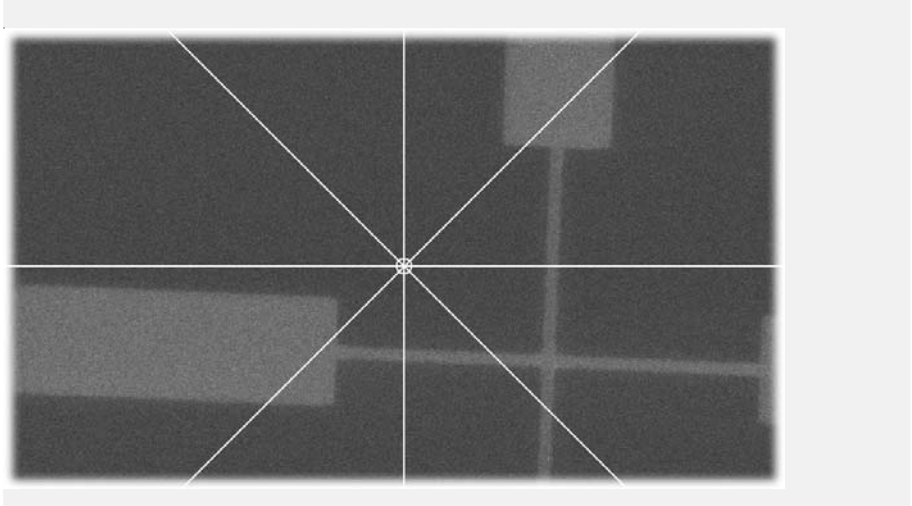
Task 2 Locating the first mark

It is assumed that you have already followed the first few chapters, including the chapter 'Exposure'. After developing the sample, load the sample into your system again and perform the steps described in the chapter, 'Stage Adjustment'

STEP 1 ▶ In order to find the first mark, open the Stage Control window and enter the value 2 for U and V. Click on Start.



STEP 2 ▶ On the column desktop select a magnification of 3000x. Switch on a crosshair and unblank the beam. The first mark should now be visible.

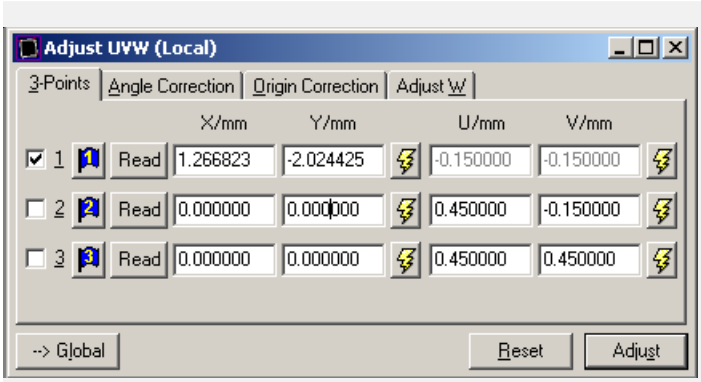


Using the joystick, move the crosshair over the mark and switch off the beam. The next task is to perform a 3-Point Adjustment.

Task 3 3-point adjustment

STEP 1 ▶ Open the Adjust UVW window and select the tab, 3-Points. Switch to the Local coordinator transformation.

STEP 2 ▶ Choose Read to update the current XY coordinates of the first mark position. Activate checkbox 1 of the same position.



In this step, we have performed, in principle, an origin correction, this means that the origin of the coordinates system has been redefined.

STEP 3 ▶ Click on the lightning icon or mark 2 related to the UV coordinates. This will move the stage to the second mark.

STEP 4 ▶ Select a high magnification again, (approximately 3000x) and switch on the beam. Move the second marker so that the crosshair is situated above the mark.

Click on Read of mark 2. The XY coordinates in the Adjust UVW window will be updated.

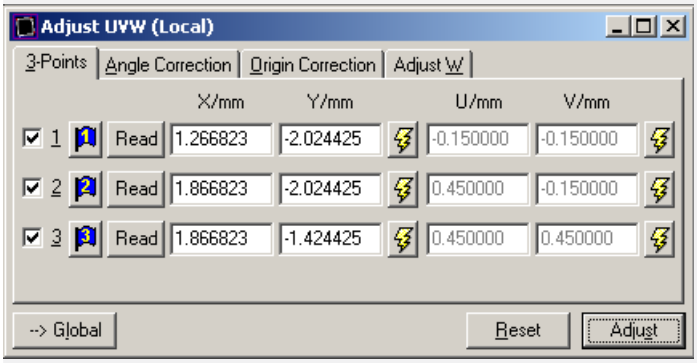
Click the checkbox of mark 2 and click Adjust. Please note that the UV coordinates have been updated after the adjustment has been performed.

STEP 5 ▶ Click on the lightning icon of mark 3 of the UV coordinate system to move the stage to mark 3.

STEP 6 ▶

Make sure that a high magnification, (approximately 3000x) has been selected and switch on the beam. Move the third mark so that the crosshair is situated above the mark. Click on Read of mark 3. The XY coordinates will be updated.

Check mark 3 and click Adjust.

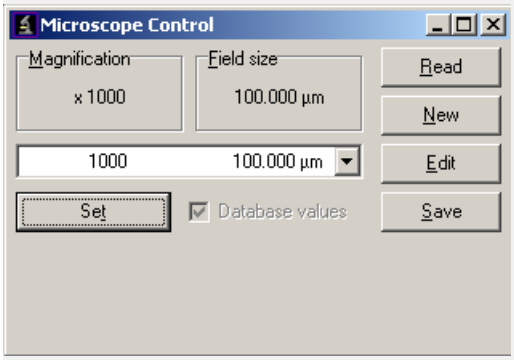


Task 4 Semi-automated write field alignment

In several cases, you may not be required to set-up a fully automated write field alignment. For example, if only a small number of alignments are necessary, or difficult mark detection conditions exist, a semi-automated procedure is more appropriate. This procedure can save time and with the interaction of the operator, more reliable results can be achieved. In the following, we will describe the semi-automated procedure first to familiarize you with the concept. The next task will describe the automated procedure, which is more complex.

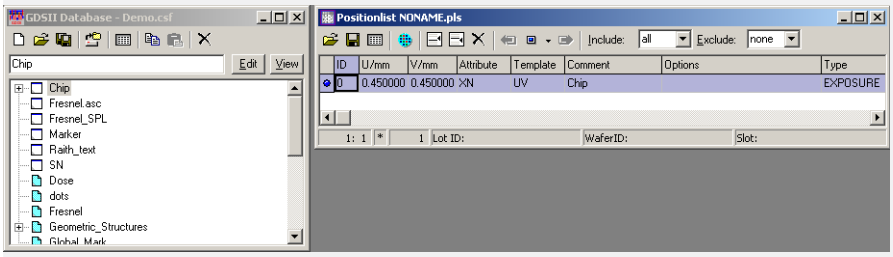
STEP 1 ▶ Move the stage back to the first mark, for example by pressing the corresponding lightning icon.

STEP 2 ▶ Open the Microscope Control window, select 100 µm write field from the list. Check Database values and press Set.



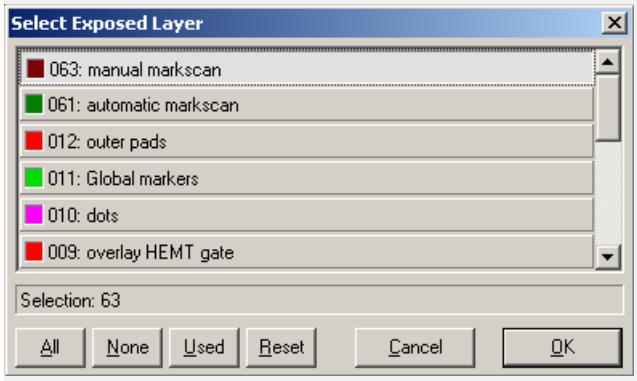
STEP 3 ▶ Open a new positionlist.

STEP 4 ▶ Select the structure ‘Chip’ from the database Demo.csf and drag & drop it into the positionlist.



STEP 5 ▶

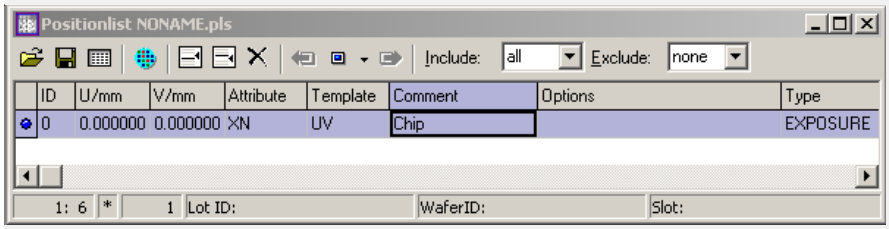
Select the line in the positionlist using the right mouse button. Select Properties. Click on the layer icon and select layer 63.



STEP 6 ▶

Click on the Working area icon and select the Working area 'Write field Calibration' and confirm with OK. Adjust the UV position by pressing the corresponding button.

This command will use the pre-defined working area and the write field size to calculate the correct sample UV position. It is very important to set-up the write field and working area beforehand.

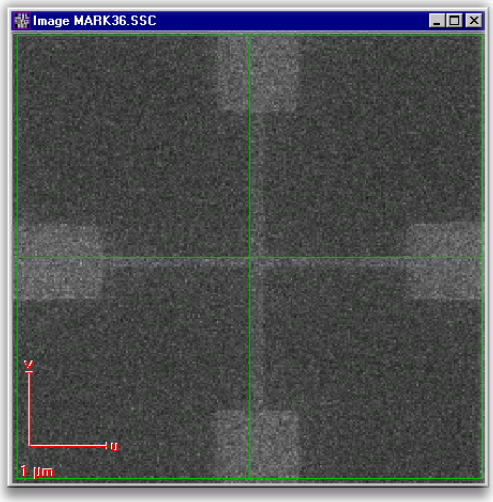


STEP 7 ▶

Activate the positionlist. Select Scan > All from the menu bar. The stage will now drive to the corresponding position and the exposure on Layer 63 will be initiated. The software will generate the positionlist 'Align.pls'. The positionlist will be filled with the corresponding Marks scan. The scanning of the positionlist will start automatically and after the first image, the software will pause to await interaction with the user.

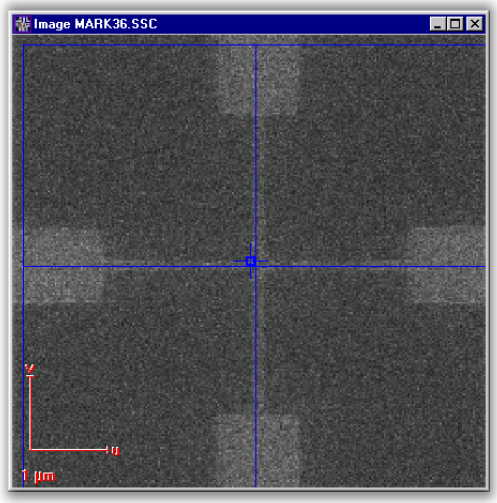
STEP 8 ▶

The green cross displayed in the center of the image defines where the mark is expected. At this stage, the mark will probably not be at the center, but it can now be defined manually. To define the position of the mark, keep the Ctrl key depressed and the left mouse button pressed while moving the mouse cursor to the required position. Once you have reached the new position, release the Ctrl button and a blue cross will be displayed at the selected position.



The green cross shows the position where the mark is expected

The cross can be moved to the exact mark position. Once the location is accepted, a blue cross appears at the mark position and the former center is marked as well.



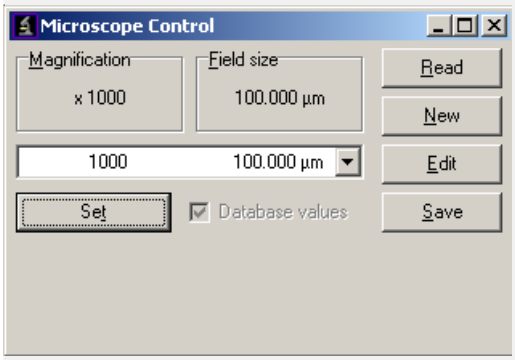
STEP 9 ▶

Click on Continue to proceed with the positionlist and the following mark scans.

Task 5 Automated write field alignment

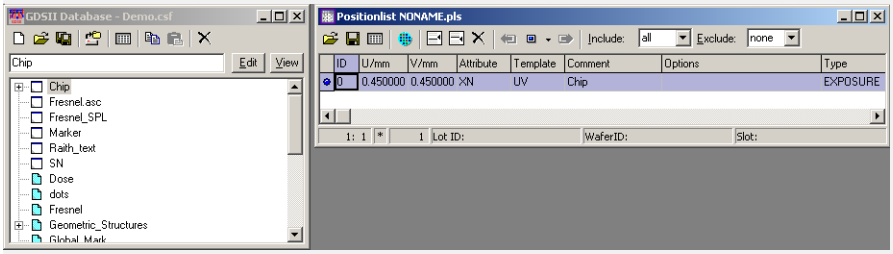
STEP 1 ▶ Move the stage back to the first mark, for example by pressing the corresponding lightning icon.

STEP 2 ▶ Open the Microscope Control window, select 100 µm write field from the list. Check Database values and press Set.



STEP 3 ▶ Open a new positionlist.

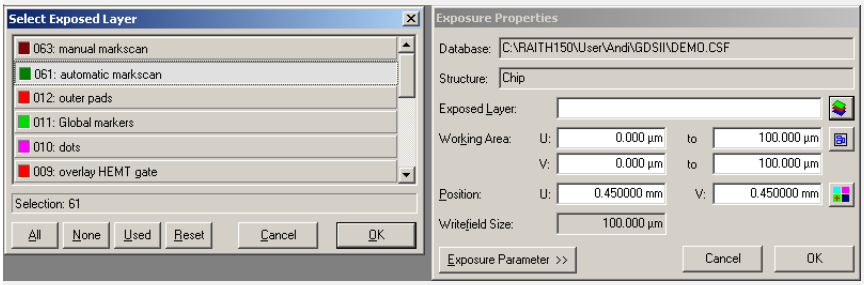
STEP 4 ▶ Select the structure 'Chip' from the database Demo.csf and drag & drop it into the positionlist.



STEP 5 ▶

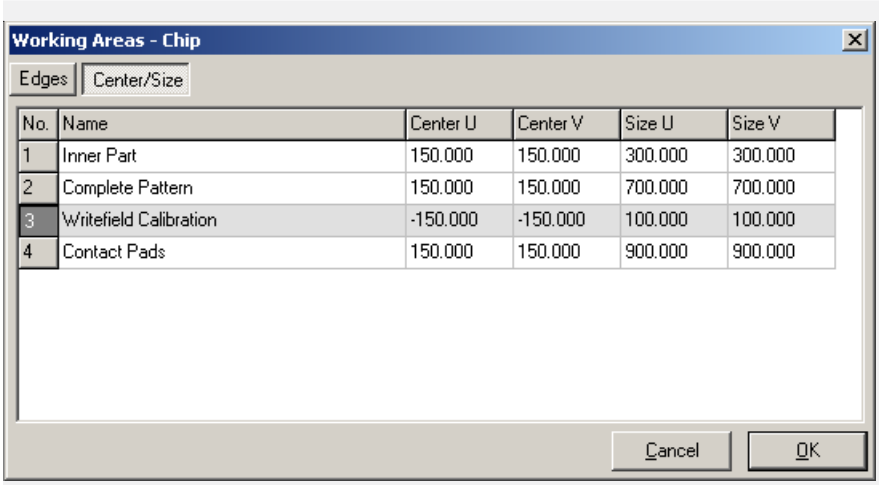
Click once with the right mouse button at the corresponding line and a dialog box will be displayed. Select Properties.

Click then on the Layer icon and select layer 061.



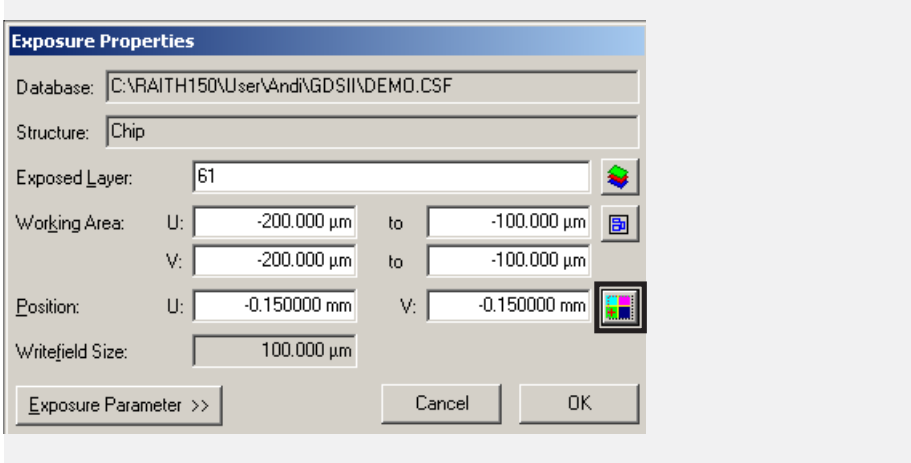
STEP 6 ▶

Click on the Working Area icon.



Select the workfield area ‘Write field Calibration’. Confirm with OK.

STEP 7 ▶ Adjust the UV position by pressing the corresponding button.



This command will use the pre-defined working area and the write field size to calculate the correct sample UV position. It is very important to set-up the write field and working area beforehand.

STEP 8 ▶ Activate the positionlist. Select Scan > All from the Menu bar. The stage will now drive to the corresponding position and the exposure on layer 61 will be initiated.

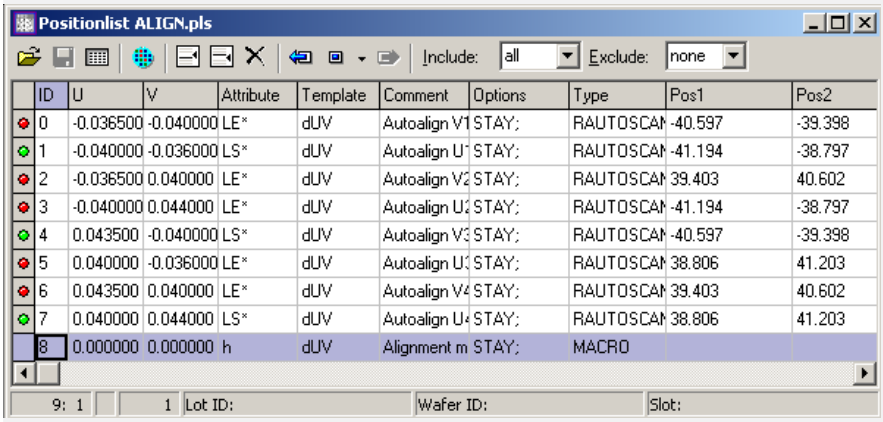
The software will open a new positionlist, called 'Align.pls'. A set of mark detections is stored within this positionlist and executed automatically.

During the execution of the positionlist Align.pls we will be able to observe progress. Several Line scans will be displayed, but it is unlikely that there will be a valid parameter set for mark detection within the line scanning and many errors will be shown. Once the execution of the positionlist is completed, the software will close 'Align.pls' which will close all the Line scans.

STEP 9 ▶

The next step is to find a parameter set such that during the automated align write field procedure, the software will be able to detect all the marks.

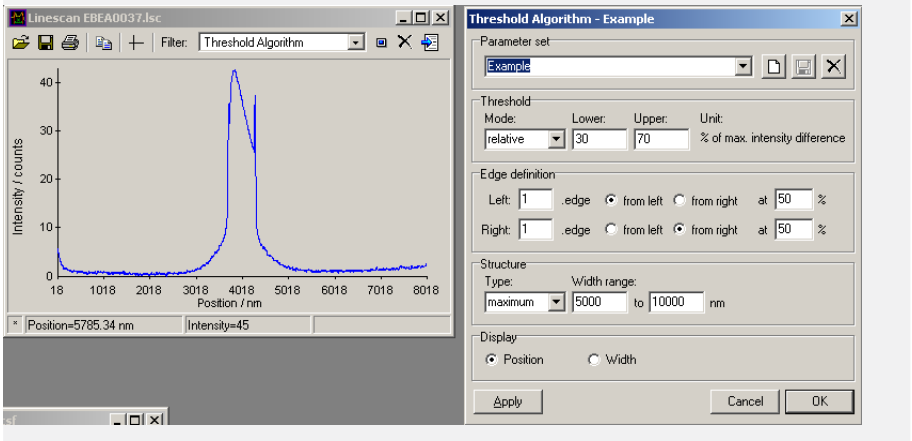
Go to File > Open Positionlist and open the positionlist 'Align.pls', which has been stored in your user directory 'Data'.



As we have not completed the optimization yet, the light is displayed in red, since the Line scan could not be completed successfully. The corresponding Attributes show LE for Line scan error.

STEP 10 ▶

Double click on one of the lines with an error and the corresponding Line scan will be opened. Select the 'Threshold Algorithm' from the dropdown list and choose the Apply button.

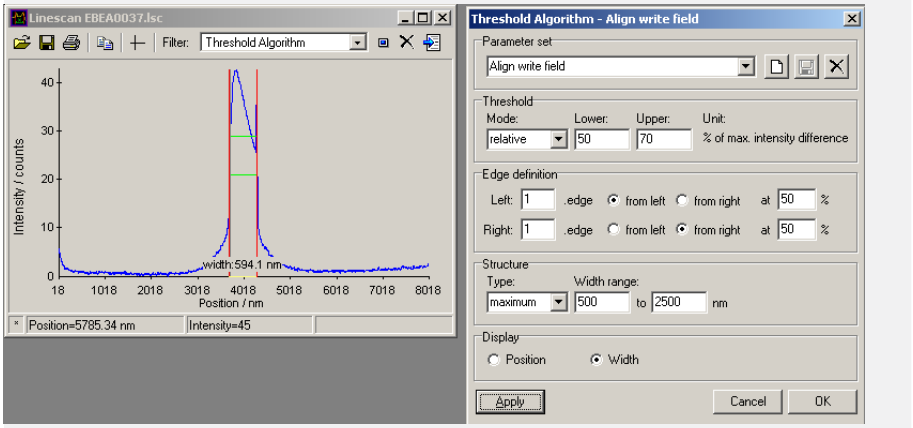


STEP 11 ▶

Select the parameter called 'Align Write field' from the dropdown list. Select Relative. For Lower select 50, for Upper, select 70. For Edge Definition, select 1st edge from left and 1st edge from right. For both edges select 50%. For Structure select type Maximum and a Width range from 500 nm to 2500 nm.

STEP 12 ▶

Press Apply. The software now applies the threshold algorithm with the parameter set chosen to the corresponding Line scan. If you were able to detect a mark, then the corresponding result will be displayed in the line scan by plotting red bars and a particular line width bar.



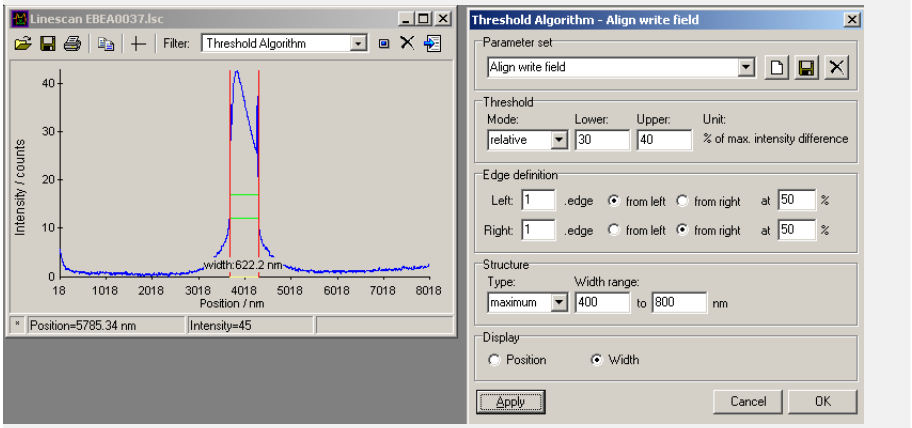
STEP 13 ▶

The next step is to optimize the parameter set. In our example, the width was 594 nm, which means that we restrict the width range.

Go back to the parameter set window and select a structure width range of 400-800. Press Apply again.

STEP 14 ▶

In our example, the threshold of 50% and 70% were not well selected. By reducing both thresholds to 30% to 40%, Improved results were achieved.

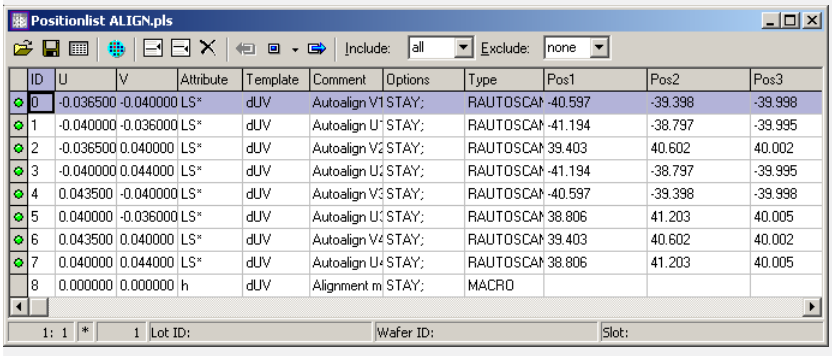


As we have now defined the parameter set. The software will now be able to detect the line successfully in the Threshold Algorithm window. Save the parameters and close the window with OK. In addition, close only the Line scan window but leave the positionlist 'Align.pls' open.

STEP 15 ▶

The next step is the verification of the parameter set. Activate the window Positionlist 'Align.pls'. In the Menu bar, go to Scan > All. The software will start scanning the positionlist again.

It is very likely that the software will now be able to apply the Threshold algorithm to all the Line scans. Therefore, there will no longer be an error message in the positionlist.



After the positionlist 'Align.csf' has been performed successfully, close the window.

Hint

If the positionlist could not be performed successfully, you will need to change the parameters for the Threshold Algorithm. Therefore, start from STEP 10 again.

Task 6 Exposure

STEP 1 ▶ Select the first positionlist, e.g. 'NoName.pls', and press with the right mouse button on the corresponding line. Select Properties again.

STEP 2 ▶ Now we select the layers to be exposed. Choose on the Layer icon. Select Layers 7, 9 and 61 or 63. Confirm with OK.

Click on the Working area icon and select the Working Area, 'Complete the Pattern' and confirm with OK.

Adjust the UV position by pressing the corresponding button.

STEP 3 ▶ Choose Exposure Parameters, which will give you access to the complete set of exposure parameters, which are disabled, prior to selecting calculator.

The next step is to enter the Area Dose, which depends on your resist. For example, if you use PMMA, 950 k molecular weight, thickness 100 nm and beam voltage of 10 keV, the area dose is about 100 $\mu As/cm^2$.

STEP 4 ▶ Enter the step size of 0.016 µm. Press the Calculator button next to the Dwell time. This will recalculate the corresponding Area dwell time according to the formula shown at the bottom. Of course the beam current has to be read beforehand. Confirm with OK.

STEP 5 ▶ The last task is now to execute the positionlist. Depending if you have chosen automated procedure (Layer 61) or semi-automated procedure (63), user interactions might be required. After completion, the sample can be developed and inspected.