NanoSuite
Software Operation Manual

Release 6.0

Product:	Software Operation Manual
Document number:	1-01-1.0
Date generated:	Thursday, July 01, 2010

Statement

All information in this document have been carefully prepared and are believed to be accurate and complete. If doubts exist about any detail or additional information is necessary, please contact 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.
GDSII	is the registered trademark of Calma GE.

Addresses

Raith GmbH	Tel.: +49 (0)231 / 95004 - 0
Konrad-Adenauer-Allee 8	Fax.: +49 (0)231 / 95004 - 460
44263 Dortmund	WWW: www.raith.com
Germany	Email: support@raith.de

Raith USA, Inc.	Tel.: +1 631 738 9500
2805 Veteran’s Highway	Fax.: +1 631 738 2055
Suite 23	WWW: www.raith.com
Ronkonkoma, NY 11779	Email: support@raithusa.com

Raith Asia Ltd.	Tel.: +852 2887 6828
Two Chinachem Exchange Square	Fax.: +852 6627 0368
No. 338 King's Road	WWW: www.raith.com
Floor 11, Unit 03	Email: support@raithasia.com
North Point, Hong Kong	

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 a patterning task, you will need to carry out chapters 1-5 first before performing the patterning in chapter 7. It is important to study the chapters in the given order.

1 Getting Started

2 E-beam Optimization

3 Stage Adjustment

4 Writefield Alignment

4.1 Standard Procedure

4.2 Using FBMS and Beam Tracking

5 General Pattern Design

6 Advanced Pattern Design

6.1 Standard

6.2 Using FBMS

7 Patterning

7.1 Standard

7.2 Patterning for FBMS Elements

8 Mix & Match Patterning

9 Automation

10 Patterning on Wafer

1 Getting Started

AIM

The aim of this chapter is to familiarize yourself with the basic functions of the RAITH Turnkey. The first task is to switch the system on, load the sample and to obtain an 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
Task 2 Preparing a suitable sample
Task 3 Loading and unloading samples
Task 4 Obtaining an image
Task 5 Finding your sample

Task 1 Start the system

HINT

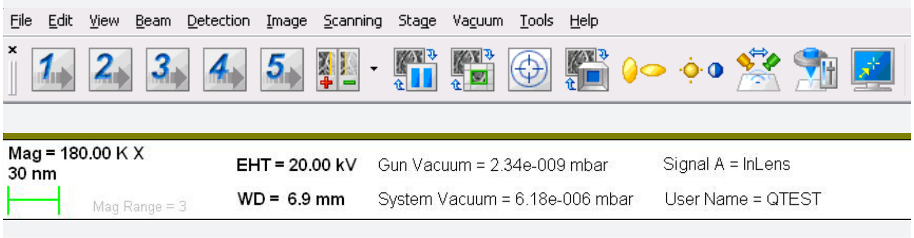
If the system has been left in another status, i.e. switched off completely, please contact a specialist for advice. For the operation of the RAITH Turnkey system, both the column and lithography software have to be installed and in addition the RemCon32 at the column PC must be running in order to provide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

STEP 1 ▶

Start the column software and log in as user **training** and password **training**.

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

Figure 1- 1 Operation icons of the column desktop.

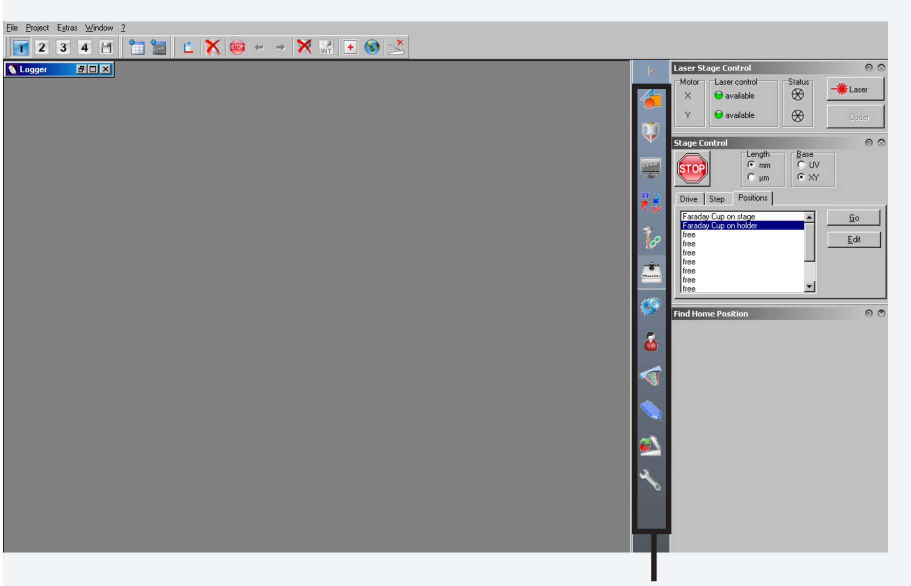


Mag = 180.00 K X
30 nm
Mag Range = 3
EHT = 20.00 kV
WD = 6.9 mm
Gun Vacuum = 2.34e-009 mbar
System Vacuum = 6.18e-006 mbar
Signal A = InLens
User Name = QTEST

STEP 2 ▶

Start the RAITH lithography software and log in as user **training** and password **training**.

Figure 1-2 Opening window of RAITH Lithography software.

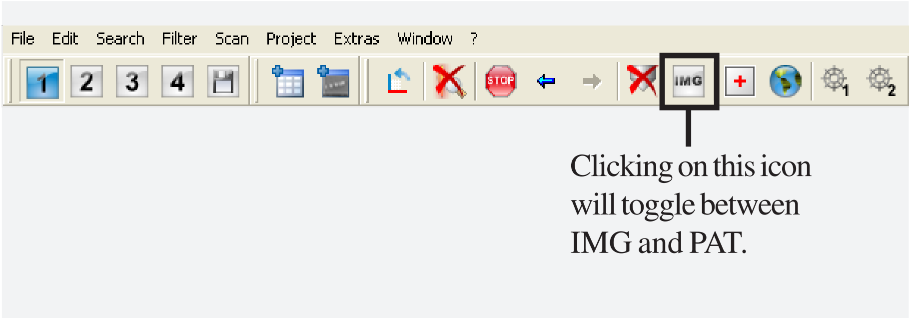


The control bar enables different windows to be displayed.

STEP 3 ▶

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

Figure 1-3 Selecting the IMG icon.

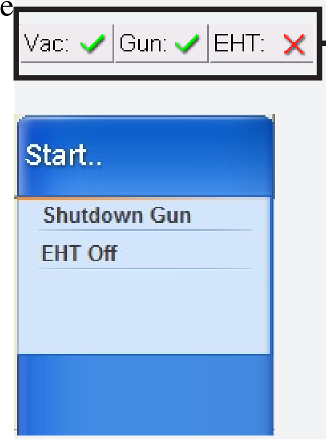


Clicking on this icon will toggle between IMG and PAT.

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 check mark) and that the acceleration voltage EHT is switched off (red cross).

Figure 1-4 Checking the EHT status.



To change the status of Vac, Gun or EHT, click on the red cross or the green check mark.

A dialog will be shown, in which the shutdown procedure for the Gun can be initiated, or the EHT can be switched off or switched on.

HINT

The toggle between **Coarse** and **Fine** control is a most useful feature. Coarse and Fine control is always related to the currently selected parameters, 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 electron optics optimization which coincides automatically with the optimized beam conditions for patterning.

For this chapter we recommend a small sample, for example a 1 cm x 1cm square, with positive 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 inside the vacuum chamber. Click on the **Chamber Scope/ Detector Control** icon in the column desktop.

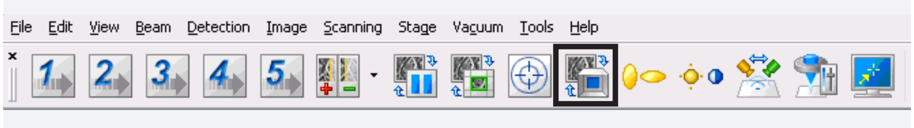


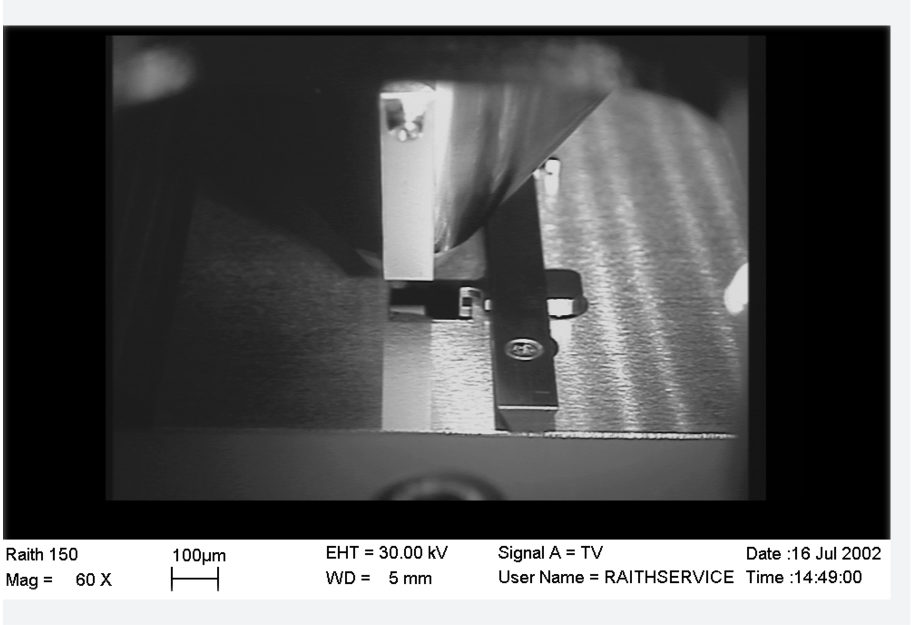
Figure 1-5 Selecting the Chamber Scope/ Detector Control icon.

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 need 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, into the loadlock.



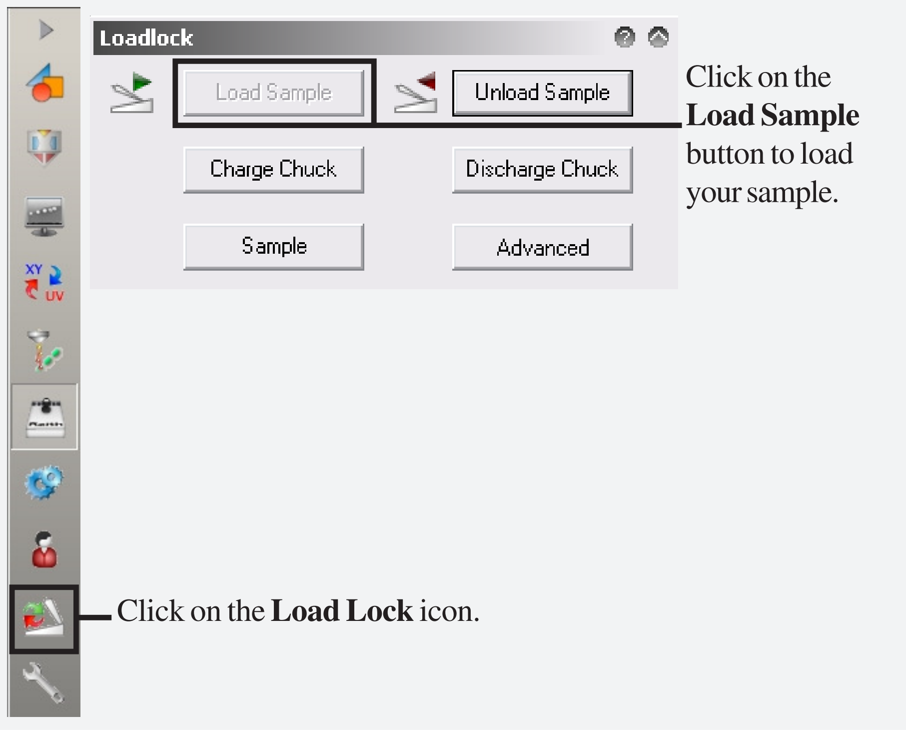
Raith 150
Mag = 60 X
100µm
EHT = 30.00 kV
WD = 5 mm
Signal A = TV
User Name = RAITHSERVICE
Date :16 Jul 2002
Time :14:49:00

Figure 1-6 CCD Camera view.

STEP 2 ▶

Click on the **Load Lock** icon in the control bar and then on the **Load Sample** button. This button is marked gray if a sample is already loaded.

Figure 1-7 Load Lock window for sample loading.



Click on the **Load Sample** button to load your sample.

Click on the **Load Lock** icon.

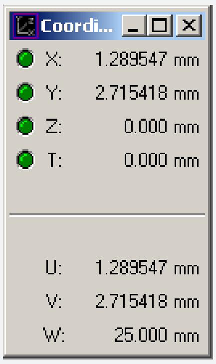
STEP 3 ▶

After the loading procedure is completed, the voltage is switched off. Switch the voltage on again via the **Column Control**.

Once the acceleration voltage is switched on, the EHT button should show a green check mark.

Check the **Home Position**. Using the lithography desktop, go to the Coordinates window and check if XYZ are displayed as zero.

Figure 1- 8 Coordinates Window showing XYZ and UVW coordinates.



● X: 1.289547 mm
● Y: 2.715418 mm
● Z: 0.000 mm
● T: 0.000 mm

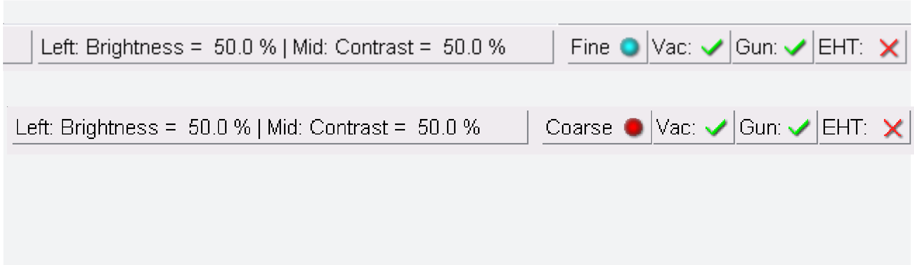
U: 1.289547 mm
V: 2.715418 mm
W: 25.000 mm

Task 4 Obtaining an 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 gray picture. To obtain an image you need to adjust the column parameters as explained in the following steps.

Figure 1- 9 Coarse/ Fine control.



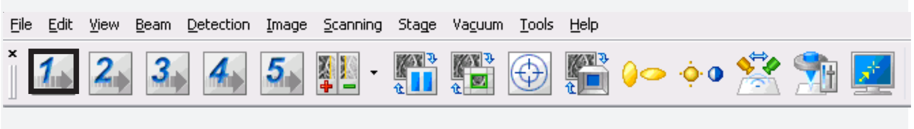
Left: Brightness = 50.0 % | Mid: Contrast = 50.0 %

Left: Brightness = 50.0 % | Mid: Contrast = 50.0 %

STEP 1 ▶

Select **Scan Speed 1** 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.

Figure 1-10 Selecting the Scan Speed.



STEP 2 ▶

If the **EHT** 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 to the left of the INT icon in the lithography desktop and check if the beam blanker changes the signal during the scan. Leave the beam on.

Figure 1-11 Checking the Beam Blanker status.

STEP 4 ▶

In addition switch to imaging mode IMG.

Figure 1-12 Imaging mode (IMG)

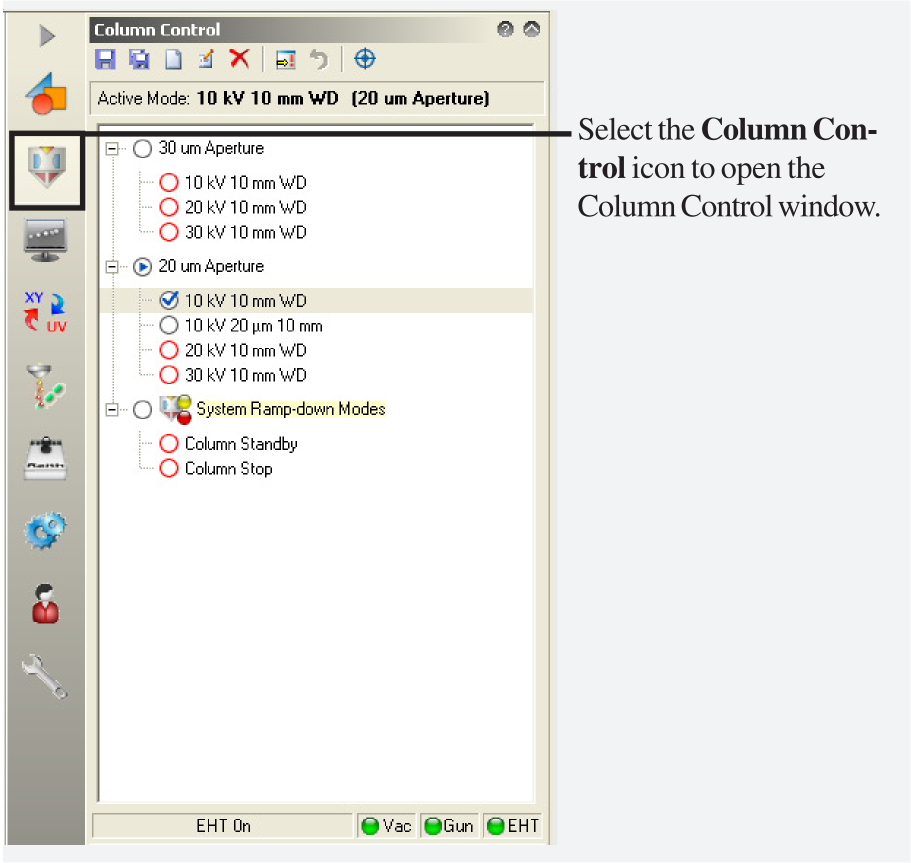
Ensure that the icon is set to IMG.

STEP 5 ▶

The next step is to check the acceleration voltage.

Click the **Column Control** icon in the **control** bar to open the Column Control window. Select a pre-written parameter set for **Aperture**, **EHT** and **Working Distance.**

Figure 1-13 Setting the Acceleration value



Active Mode: **10 kV 10 mm WD (20 um Aperture)**

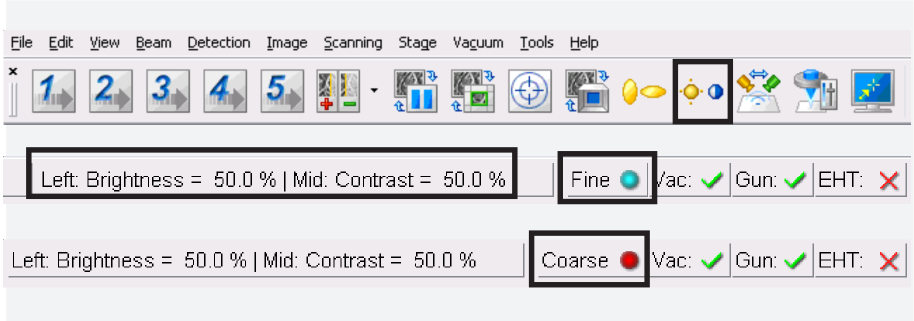
- ○ 30 um Aperture
  - ○ 10 kV 10 mm WD
  - ○ 20 kV 10 mm WD
  - ○ 30 kV 10 mm WD
- ● 20 um Aperture
  - ☑ 10 kV 10 mm WD
  - ○ 10 kV 20 µm 10 mm
  - ○ 20 kV 10 mm WD
  - ○ 30 kV 10 mm WD
- ○ System Ramp-down Modes
  - ○ Column Standby
  - ○ Column Stop

Select the **Column Control** icon to open the Column Control window.

STEP 6 ▶

The next step is to adjust brightness and contrast. Click the icon for **Brightness** and **Contrast**. The left and middle mouse buttons 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.

Figure 1-14 Setting the **Brightness and Contrast** values.



Left: Brightness = 50.0 % | Mid: Contrast = 50.0 %

Fine ●

Vac: ✓ Gun: ✓ EHT: ✗

Left: Brightness = 50.0 % | Mid: Contrast = 50.0 %

Coarse ●

Vac: ✓ Gun: ✓ EHT: ✗

The left mouse button **Left** is assigned to brightness control and the middle mouse button **Mid** is assigned to contrast control.

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

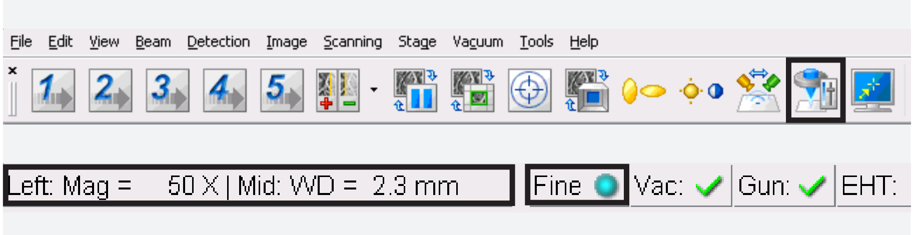
HINT

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 7 ▶

Now that the Brightness and Contrast have been optimized, we can start to focus onto a surface using a selected magnification of 50x . Click on the **Magnification** icon using the left mouse button and assign the left and middle mouse buttons 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.

Figure 1-15 Setting the **Magnification.**



Left: Mag = 50 X | Mid: WD = 2.3 mm

Fine ●

Vac: ✓

Gun: ✓

EHT:

The left mouse button **Left** is now assigned to **Magnification** control and the middle mouse button **Mid** is assigned to **Working Distance**. The mouse movement can be toggled between **Fine** and **Coarse** control.

HINT

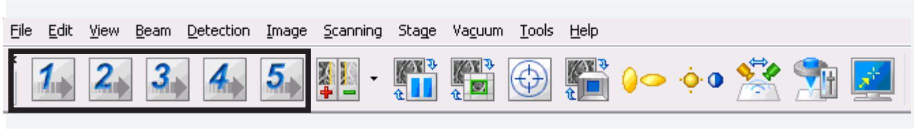
Please note that focus is related to working distance.

STEP 8 ▶

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 reduces the scan speed. The pre-defined Scan Speed 1 is the fastest scan speed, whereas the Scan Speed 5 is the slowest scan speed. The user can select the individual scan speed via the **Raith EOControl > Scanning** tab assigned to the **Scan Speed** icons.

Select a slower scan speed in order to reduce the noise by clicking the left numbered icons.

Figure 1-16 Setting the Scan Speed.



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.

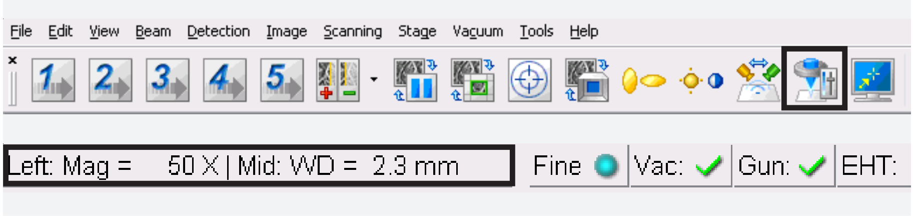
Figure 1-17 Joystick control.

Move close to your sample but do not move over it, otherwise you would expose the sample.

STEP 2 ▶

We can now start to focus onto the sample holder using our selected magnification of 50x. Click on the **Magnification** icon using the left mouse button to assign the left and middle mouse buttons 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.

Figure 1- 18 Selecting the Focus icon to adjust the focus of the sample.



Left: Mag = 50 X | Mid: WD = 2.3 mm

Fine ●

Vac: ✓

Gun: ✓

EHT:

HINT

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

HINT

If you are operating the RAITH150-TWO system, when the stage is moved to the right, the electron optics image will move to the right. The same is valid for the CCD camera, e.g. the stage is moved to the right, the CCD camera view is moved to right.

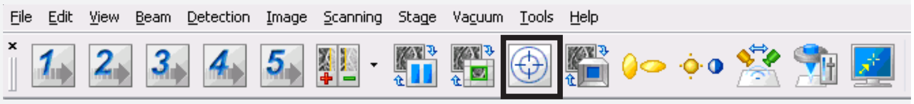
If you are operating the e_LINE system, when the stage is moved e.g. to the right, the image will move to the right, but the CCD camera view is 180° rotated, so it will show an apparent move to the left.

STEP 3 ▶

Now that you have optimized the focus, you need to locate the sample at low magnification. Click on the **Chamber Scope/Detector Control** icon to switch back to the electron optics image. Move the lower left corner of your sample into the center of the field of view.

HINT

You can turn on the crosshairs, indicating the center of your screen, by clicking on the icon with the centered cross.



STEP 4 ▶

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 reduces the scan speed. Reduce the scan speed in order to reduce the noise by clicking the left numbered icons. 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).

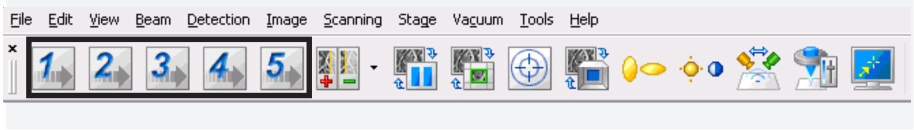


Figure 1-19 Changing the **Scan Speed.**

2 E-beam Optimization

AIM

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

Task 1 Focusing on the sample
Task 2 Aperture alignment
Task 3 Astigmatism correction
Task 4 Further E-beam optimization
Task 5 Creating a spot
Task 6 Checking the leveling limits

Task 1 Focusing on the sample

STEP 1 ▶ It is assumed that you have loaded a 1 cm x 1 cm 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.

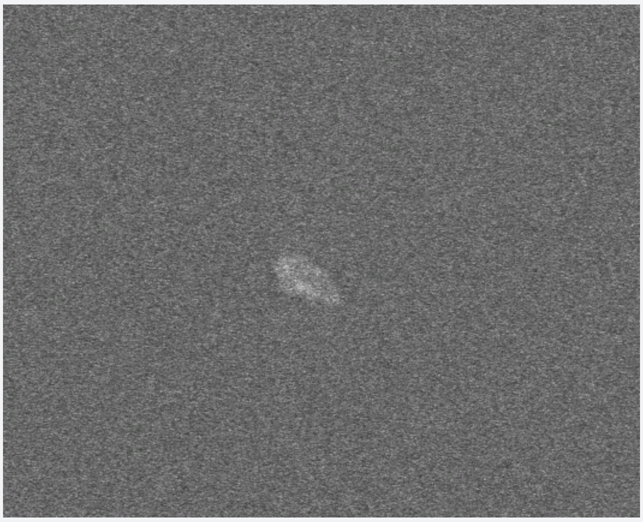


Figure 2-1 Focusing on a **Particle** on the sample.

STEP 3 ▶ Zoom onto the particle until you seem to lose the focus. Remember that **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 **focus** is assigned to the middle mouse button.

STEP 5 ▶ Zoom in further and readjust the focus.

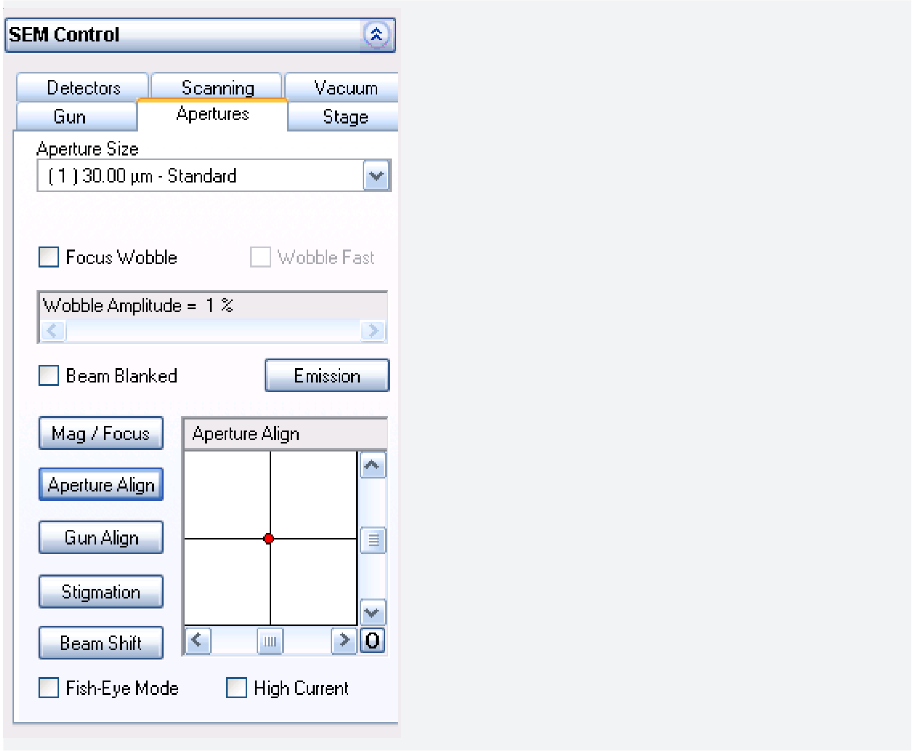
STEP 6 ▶ Repeat the zoom and refocus procedure until no further improvement in focus can be achieved.

Task 2 Aperture alignment

STEP 1 ▶

Open the Raith EO control panel **Tools > Go to Control Panel** (Ctrl-G) and select the **Apertures** tab.

Figure 2-2 Opening the **Aperture** alignment.



Aperture Size: (1) 30.00 µm - Standard

☐ Focus Wobble ☐ Wobble Fast

Wobble Amplitude = 1 %

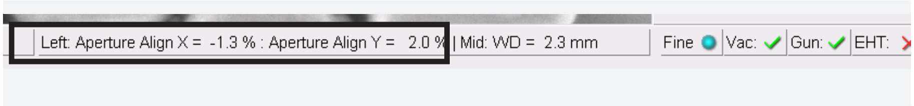
☐ Beam Blanked

☐ Fish-Eye Mode ☐ High Current

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.

Figure 2-3 Viewing the **Status** bar.



Left: Aperture Align X = -1.3 % : Aperture Align Y = 2.0 % | Mid: WD = 2.3 mm | Fine ● | Vac: ✓ | Gun: ✓ | EHT: ✗

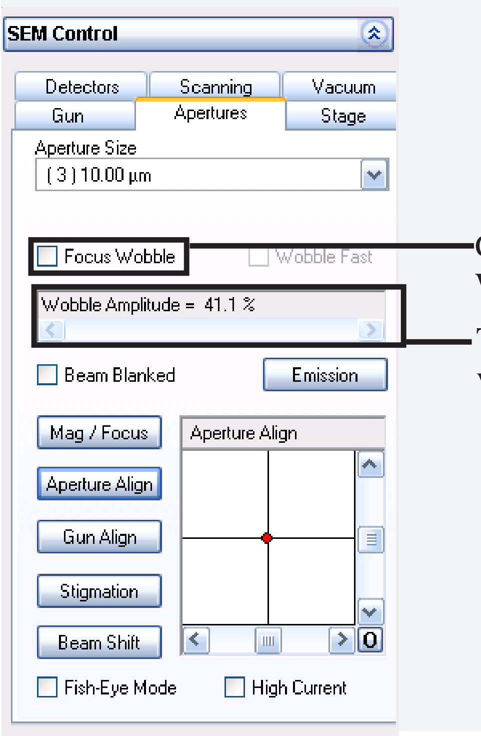
STEP 3 ▶

Go to the **Raith EO Control** window and select the **Aperture** tab. Check the checkbox for the **Focus Wobble**. This will initiate the focus wobble. Its intensity can be varied by the **Wobble Amplitude** slider bar.

STEP 4 ▶

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. Alternatively, 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 scroll bars.

Figure 2-4 Performing the **Aperture Align** procedure.



Aperture Size: (3) 10.00 µm

Wobble Amplitude = 41.1 %

Check the checkbox for Focus Wobble.

The Wobble Amplitude can be varied using the slide bar.

HINT

The key to 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 5 ▶

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

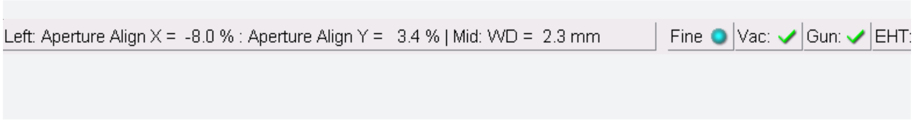
HINT

If the particle is becoming too large at high magnification, move to a smaller particle and continue the optimization. In order to change the magnification, click on the button **Mag/Focus**. Do not forget to switch off the **Focus Wobble** 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.

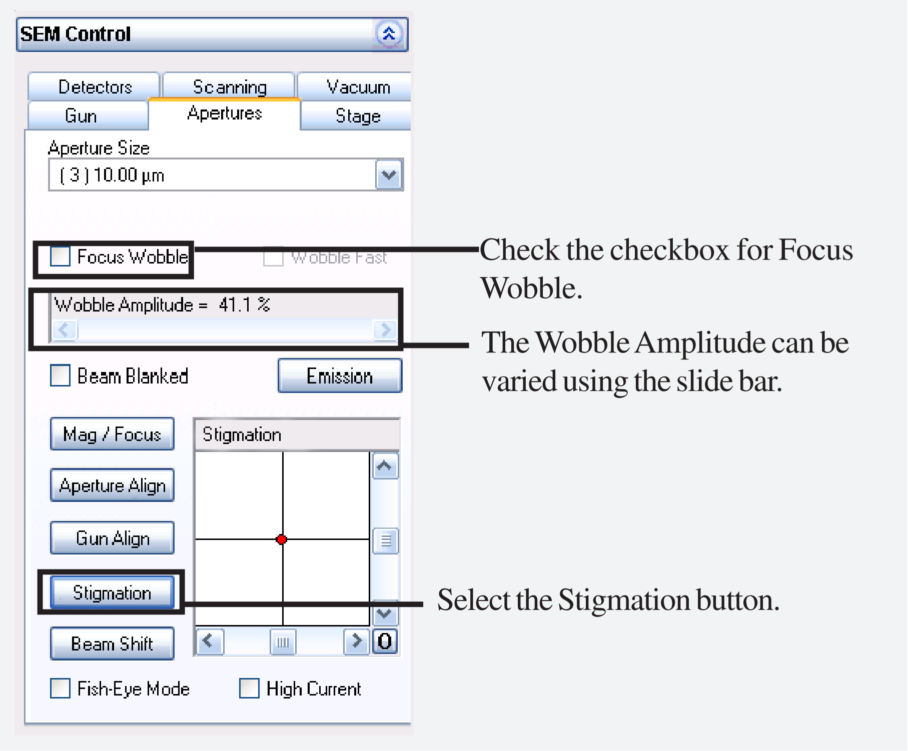


Left: Aperture Align X = -8.0 % : Aperture Align Y = 3.4 % | Mid: WD = 2.3 mm | Fine ● | Vac: ✓ | Gun: ✓ | EHT:

Figure 2-5 Assigning **Stigmation** to the mouse buttons.

STEP 2 ▶

Switch on **Focus Wobble** in the **Stigmation** tab of the Raith EO Control by clicking on the corresponding field and selecting 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.



Aperture Size: (3) 10.00 µm

☐ Focus Wobble

☐ Wobble Fast

Wobble Amplitude = 41.1 %

☐ Beam Blanked

☐ Fish-Eye Mode

☐ High Current

Check the checkbox for Focus Wobble.

The Wobble Amplitude can be varied using the slide bar.

Select the Stigmation button.

Figure 2-6 Performing the **Focus Wobble** procedure.

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 spot as described in the next task.

HINT

Please note that during aperture alignment we concentrate on the image movement whilst 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/Focus**. Magnification is now assigned to the left mouse button.

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 (uncheck **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 spot

It is recommended to burn a spot 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 area of interest to ensure that a free area of the sample is visible in order to burn the spot.

STEP 2 ▶ Click on the **Spot** icon on the column desktop using the left mouse button to burn a spot 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 spot, click on the middle mouse button which will start the spot mode. Wait for 1 minute while the spot is burned into the sample and click the middle mouse button again to end the spot mode.

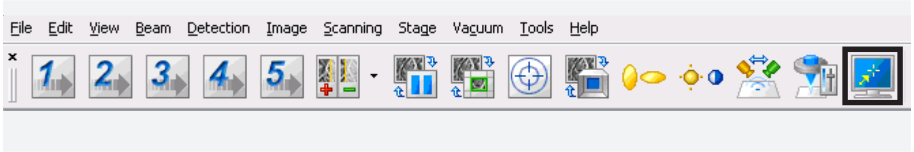


Figure 2-7 Selecting the Spot icon.

STEP 3 ▶ Focus now on the spot, move the stage and burn another spot. The new spot should be smaller since the focus has been improved.

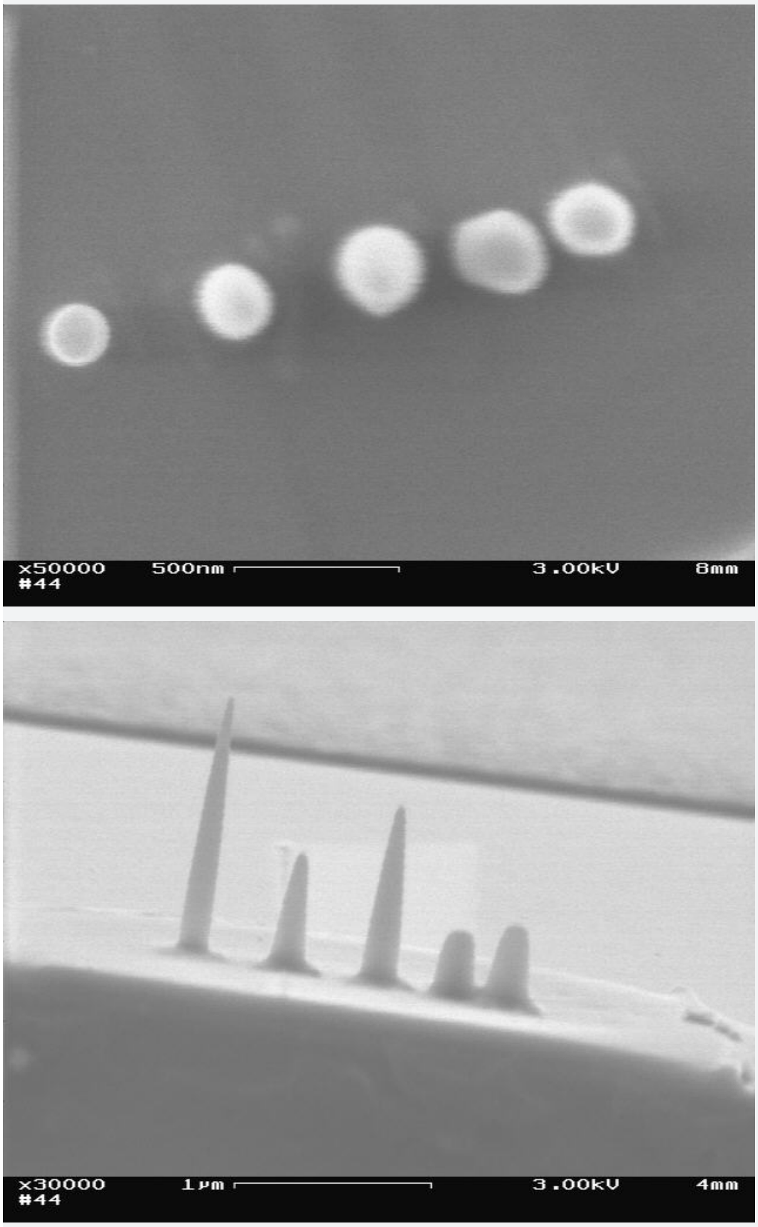
HINT

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

HINT

An example of a series of spots is shown in the images below to illustrate top and side views.

Figure 2-8 Creating a **spot.**



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 steps, but this task is not necessarily required prior to a patterning task.

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 spots.

STEP 2 ▶

Burn another spot and view the result. This spot is now 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.

3 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 1,2 and 3 how to set up a coordinate system on an sample. In task 4 we will explain how to navigate on this.

Task 1 Angle correction
Task 2 Origin correction
Task 3 Adjust W
Task 4 Digital addressing

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 ▶

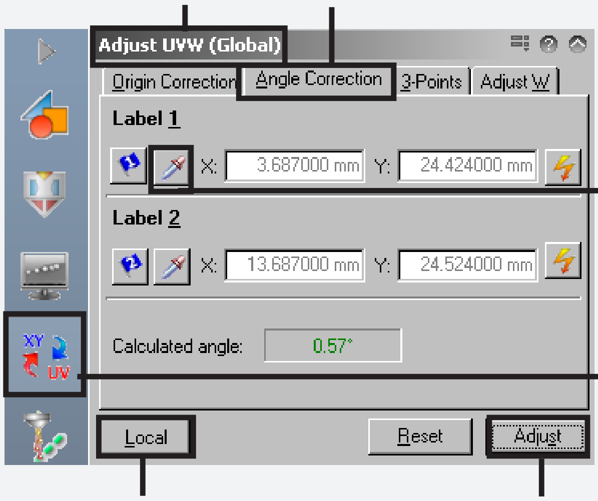
To carry out the angle correction, image the sample at medium magnification, approximately 100x. Ensure that the crosshairs are switched on by selecting the **Crosshairs** button in the column desktop. Identify the lower edge of the sample and follow this edge to the lower left corner. The crosshairs are now situated above the lower left corner.

STEP 2 ▶

On the lithography desktop, open the window **Adjust UVW** by clicking on the corresponding **Adjustment** icon in the control bar. 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 the **Pipette** icon (Read XY position) next to the Flag 1 in Adjust UVW to read in the coordinates. The coordinates will be displayed in the window.



X: 3.687000 mm Y: 24.424000 mm

X: 13.687000 mm Y: 24.524000 mm

Calculated angle: 0.57°

Figure 3-1 Adjust UVW (Global) window to read in the coordinates.

Adjust UVW is now in the Global mode.

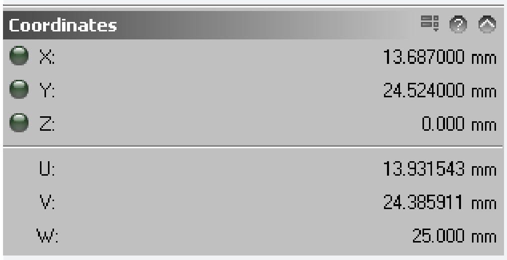
Select the Angle Correction tab.

Click on the Pipette icon to read in the coordinates.

Click on the Adjustments icon to open the Adjust UVW coordinates window.

Clicking this button will toggle between Local/ Global.

Click on Adjust to activate the Angle Correction.


Coordinates

X:	13.687000 mm
Y:	24.524000 mm
Z:	0.000 mm
U:	13.931543 mm
V:	24.385911 mm
W:	25.000 mm

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 the **Pipette** icon next to Flag 2 to read in the second position. The second set of coordinates will now be displayed in the window.

STEP 5 ▶

Click on **Adjust** to activate the angle correction.

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

Task 2 Origin correction

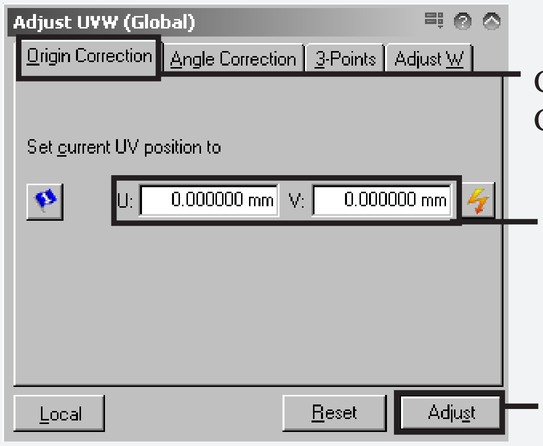
The sample can be placed at any location on the sample holder. To compensate for the different origins of XY and UV, the origin correction can be applied.

STEP 1 ▶ Ensure that the beam is blanked.

STEP 2 ▶ Within the **Angle Correction** tab, click on the **Flash** icon 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 system. It is now possible to move the stage to any point on the sample using UV coordinates.

Figure 3-2 Performing the **Origin Correction.**



Set current UV position to

U: 0.000000 mm V: 0.000000 mm

Click on the Origin Correction tab.

Enter 0 for U and V.

Click on Adjust to confirm.

HINT

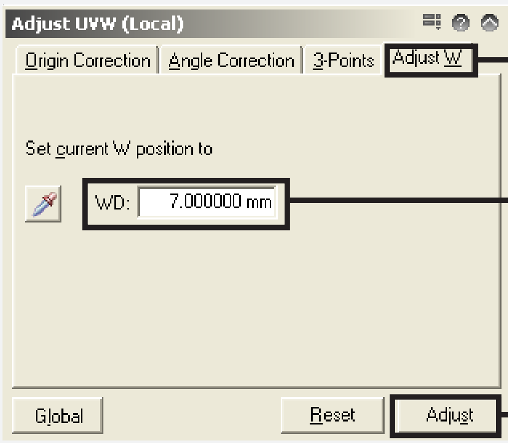
The adjustment via angle and origin is mostly used for an empty sample.

Task 3 Adjust W

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

STEP 2 ▶ Click on the **Adjust W** tab in the **Adjust UVW** (Global) window. Click the **Pipette** icon to read in the working distance. Then click on **Adjust** to confirm.

Figure 3-3 Adjust W coordinate.



WD: 7.000000 mm

Select the tab Adjust W.

This value displays the working distance.

Click on Adjust to confirm.

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 patterning sequence.

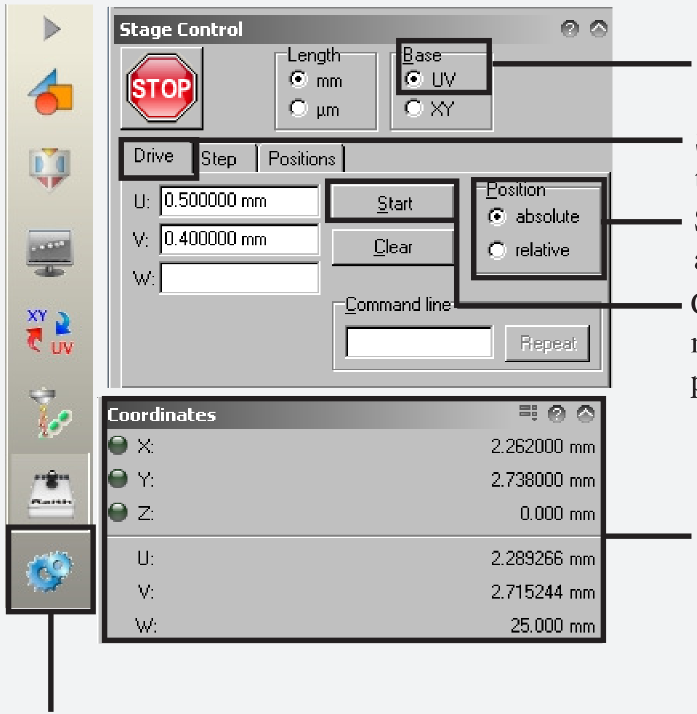
In tasks 1-3 we have established a coordinate system in UVW, which we can now use to address certain points on the sample. This will be explained in this task.

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

STEP 1 ▶

Click on the **Stage Control** icon in the control bar. Click on the **Drive** tab. Click on **Base UV** 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.

Figure 3-4 Stage Control window to address the stage position.



Length: ● mm ○ µm

Base: ● UV ○ XY

U: 0.500000 mm
V: 0.400000 mm
W:

Position: ● absolute ○ relative

Coordinates

X:	2.262000 mm
Y:	2.738000 mm
Z:	0.000 mm
U:	2.289266 mm
V:	2.715244 mm
W:	25.000 mm

Click on Base UV.

Select the Drive tab.

Select Position absolute.

Click on Start to move to the new position.

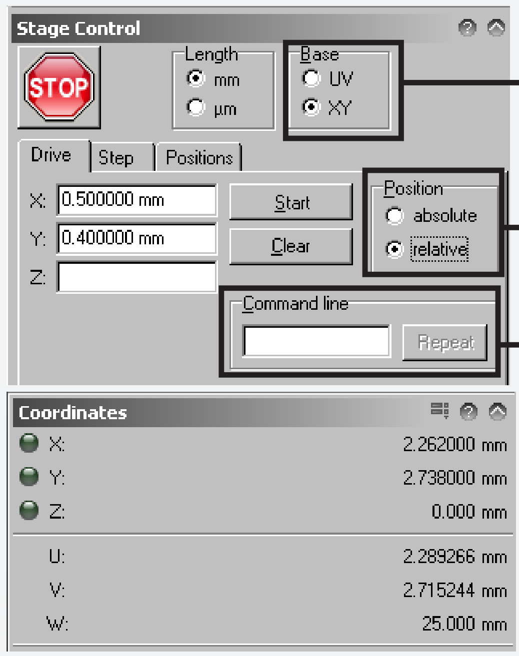
The new coordinates will be displayed.

Click on the Stage Control icon to open the Stage Control window.

STEP 2 ▶

Change the **Base** to XY, address a point in XY coordinates and monitor the coordinates window. The calculated corresponding UV coordinates are displayed continuously, while the entered XY coordinate is being addressed.

Figure 3-5 Changing the **Coordinates** in XY.



Stage Control

Length: ● mm ○ µm

Base: ○ UV ● XY

X: 0.500000 mm

Y: 0.400000 mm

Z:

Position: ○ absolute ● relative

Coordinates

X:	2.262000 mm
Y:	2.738000 mm
Z:	0.000 mm
U:	2.289266 mm
V:	2.715244 mm
W:	25.000 mm

Change Base to XY.

Select Position relative.

You can address one or two axes. Type in small letters for absolute, and capital letter for relative positioning.

STEP 3 ▶

Move the stage relative to the existing position by selecting relative. Select the Base of your choice, either UV or XY.

STEP 4 ▶

In the **Command** line it is possible to address just one axis absolutely or relatively by entering the required position or distance followed by the letter of the axis.

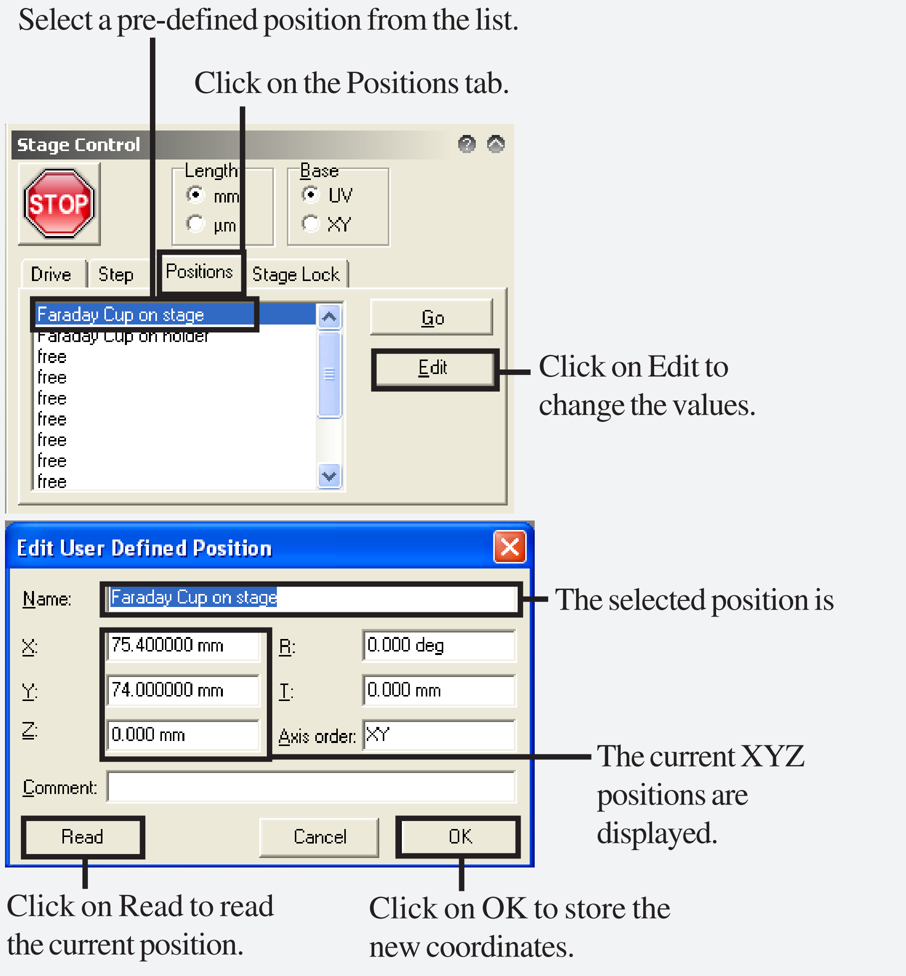
Type in small letters (x, y, u or v) for absolute positioning and capital letters (X, Y, U or V) for relative positioning. If relative addressing is selected, the movement command can be repeated in order to move stepwise in equal distances along the sample.

STEP 5 ▶

It is also possible to go to a stored position, via the **Stage Control > Positions** tab. 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 desired position.

If the stage is already at the specified position, click on **Edit**. A new dialog box, **Edit User Defined Position**, will open. Click on **Read** to read in the coordinates, and click on **OK** to store the new coordinates.

Figure 3-6 Moving to a stored position in the **Edit User Defined Position** dialog box.



Select a pre-defined position from the list.

Click on the Positions tab.

Stage Control

Length: ● mm ○ µm

Base: ● UV ○ XY

Faraday Cup on stage
Faraday Cup on holder
free
free
free
free
free
free
free

Click on Edit to change the values.

Edit User Defined Position

Name: Faraday Cup on stage
X: 75.400000 mm
Y: 74.000000 mm
Z: 0.000 mm
R: 0.000 deg
T: 0.000 mm
Axis order: XY
Comment:

The selected position is

The current XYZ positions are displayed.

Click on Read to read the current position.

Click on OK to store the new coordinates.

HINT

For 3-Points adjustment, please refer to Chapter 8 (Mix and Match Patterning), Task 3.

4 Writefield 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 Writefield alignment has to be performed. The procedure described in this chapter via Writefield alignment is required for stitching and for any patterning on a bare sample. The alignment of the field size to the previously written marks for multi-layer lithography will be explained in a later chapter.

Writefield alignment is a very important task, as it aligns the Writefield 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 Writefield is exactly parallel to UV and this can be achieved with Align Write procedures.

4.1 Writefield Alignment (Standard) Procedure

Chapter 4.1 explains the standard procedure of Writefield alignment.

Task 1 Locating a mark or particle
Task 2 Defining the alignment procedure
Task 3 Executing the alignment procedure manually
Task 4 Setting up the automated alignment
Task 5 Checking the precision of the alignment

4.2 Writefield Alignment using FBMS and Beam Tracking (Options)

Chapter 4.2 is only applicable to users who have the option for FBMS and Beam Tracking installed on their Turnkey System.

Task 1 Continue with located particle
Task 2 Defining the alignment procedure
Task 3 Executing the alignment procedure
Task 4 Setting up the automated alignment
Task 5 Checking the precision of the alignment

4.1 Writefield Alignment Procedure

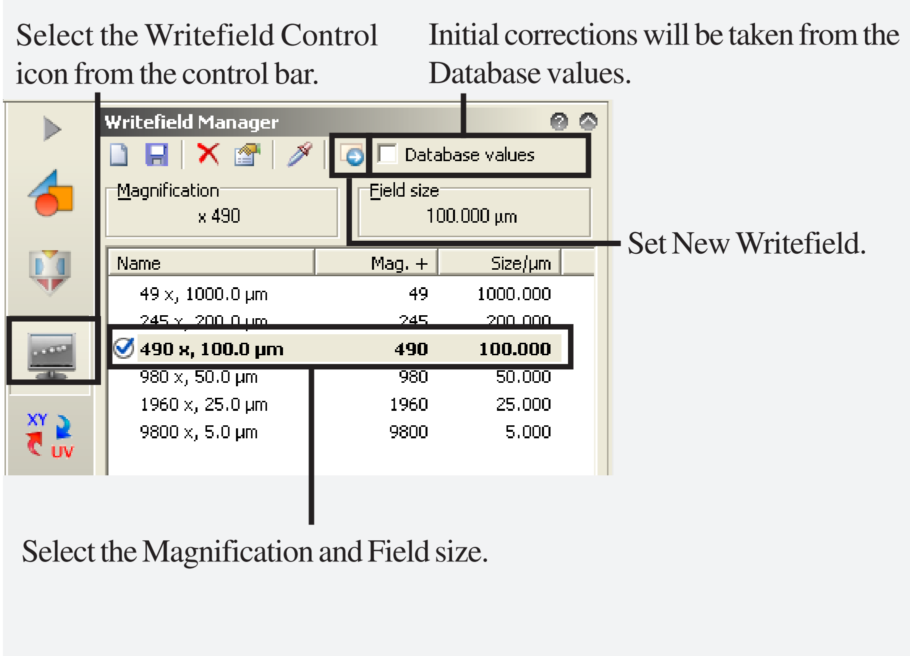
Task 1 Locating a mark or particle

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

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

STEP 3 ▶ Choose the **Writefield Manager** icon from the control bar. A list of pre-written **Magnification** and **Field size** parameters will be displayed. Select the Writefield size, in this case 100 µm. Click on the **Set New Writefield** icon to activate that line and to set the corresponding magnification. As a default, initial correction values will be taken from a database, the checkbox **Database values** is checked by default.

Figure 4-1 Open the **Writefield Manager** window.



Select the Writefield Control icon from the control bar.

Initial corrections will be taken from the Database values.

Set New Writefield.

Select the Magnification and Field size.

Writefield Manager

☐ Database values

Magnification: x 490

Field size: 100.000 µm

Name	Mag. +	Size/µm
49 x, 1000.0 µm	49	1000.000
245 x, 200.0 µm	245	200.000
☑ **490 x, 100.0 µm**	**490**	**100.000**
980 x, 50.0 µm	980	50.000
1960 x, 25.0 µm	1960	25.000
9800 x, 5.0 µm	9800	5.000

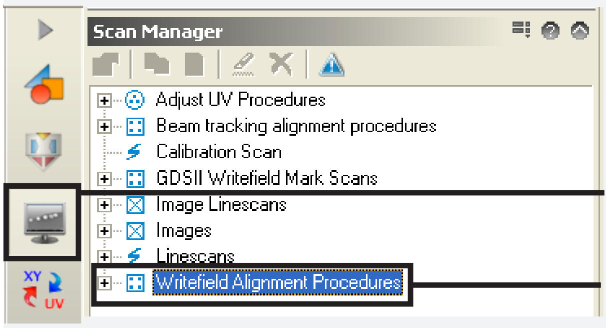
Task 2 Defining the alignment procedure

The Writefield needs to be calibrated and rotated. This procedure is called **Writefield Alignment**. 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 Writefield. Within the scan manager, all the parameters of such a procedure are stored and can be recalled for later use.

STEP 1 ▶

The **Scan Manager** window opens automatically when the **Writefiled Control** icon is selected from the control bar.

Figure 4-2 Go to Scan Manager window.



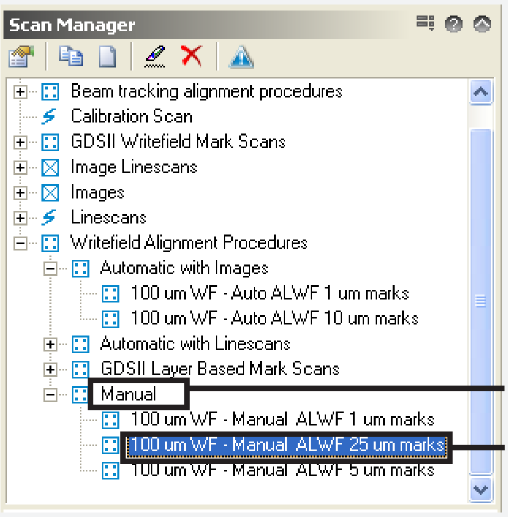
When the Writefield Control icon is selected, the Scan Manager window will open as well.

Go to the Scan Manager window and select Writefield Alignment Procedures.

STEP 2 ▶

Double click on **Writefield Alignment Procedures** and select **Manual** from the sub-procedure menu.

Figure 4-3 Select Manual procedure.



Select Manual procedure.

Select the pre-defined alignment procedure.

STEP 3 ▶

If a suitable sub-procedure is already available, task 2 is complete and you can continue with task 3.

STEP 4 ▶

If no suitable sub-procedure is available, or you would like to edit it, double click on the pre-defined alignment procedure. A new dialog box, **Scan properties**, will be displayed. You can edit the parameter values and save it by clicking **OK**. Select **New**, which will create a new sub-procedure for **Manual**. The Scan properties window for the chosen procedure will automatically open.

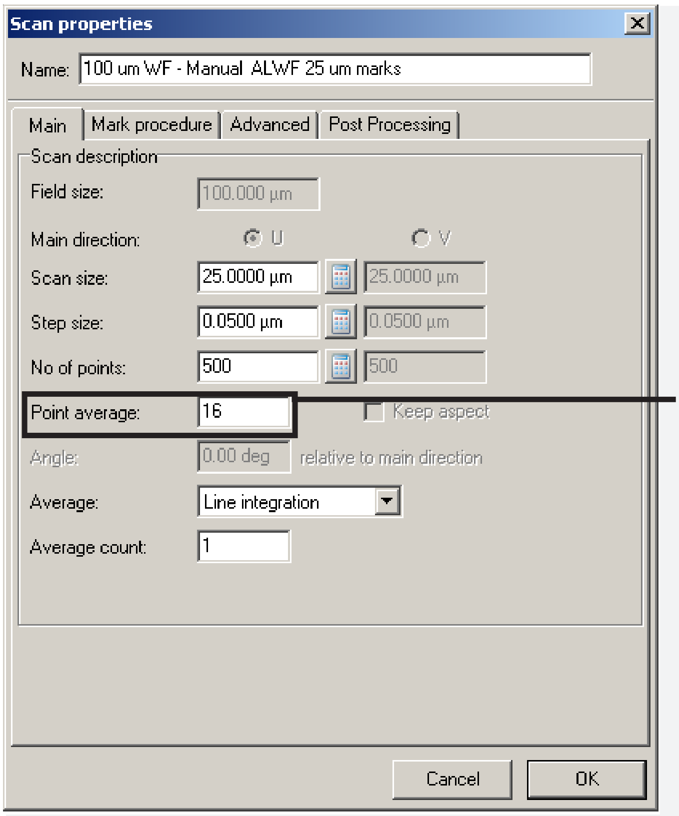


Figure 4-4 Enter the values in the **Scan properties** window.

Scan properties

Name: 100 um WF - Manual ALWF 25 um marks

Field size: 100.000 µm

Main direction: ● U ○ V

Scan size: 25.0000 µm 25.0000 µm

Step size: 0.0500 µm 0.0500 µm

No of points: 500 500

Point average: 16 ☐ Keep aspect

Angle: 0.00 deg relative to main direction

Average: Line integration

Average count: 1

Enter the number 16 for Point average.

HINT

It is recommended to choose **16** for **Point average** in order to slow down the beam to avoid any dynamic effects.

HINT

Finding the suitable scan size is dependent on several factors such as:

- For a newly defined Writefield, it is recommended to start with large scan sizes. We have selected a 100 µm Writefield. The scan size should be of the order of 25 µm.
- For a Writefield 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. 10 µm.

HINT

It is recommended to rename the manual Writefield procedure to include the Writefield size in the title. Right mouse click opens a context menu. 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.

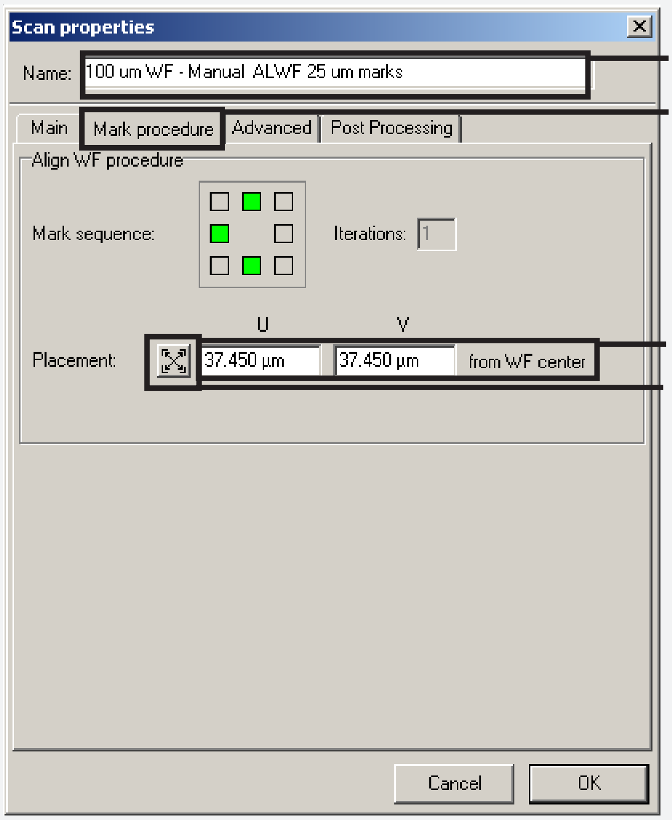
HINT

When changing the values, you must be careful about the correlation between the parameters.
Step size x No of points = scan size
If this correlation cannot be fulfilled, the entered value is non-valid.

STEP 5 ▶

Choose the **Mark procedure** tab. Check the Mark sequence as shown in the example. For **Placement** parameters, enter 37.450 µm in U and V.

Figure 4-5 Mark Procedure parameters.



Name: 100 um WF - Manual ALWF 25 um marks

Mark sequence:
☐ 🟩 ☐
🟩   ☐
☐ 🟩 ☐

Iterations: 1

Placement: U 37.450 µm V 37.450 µm from WF center

Select the sub-procedure.

Select the tab Mark procedure.

Enter the values for the Placement in U and V.

Auto-placement button.

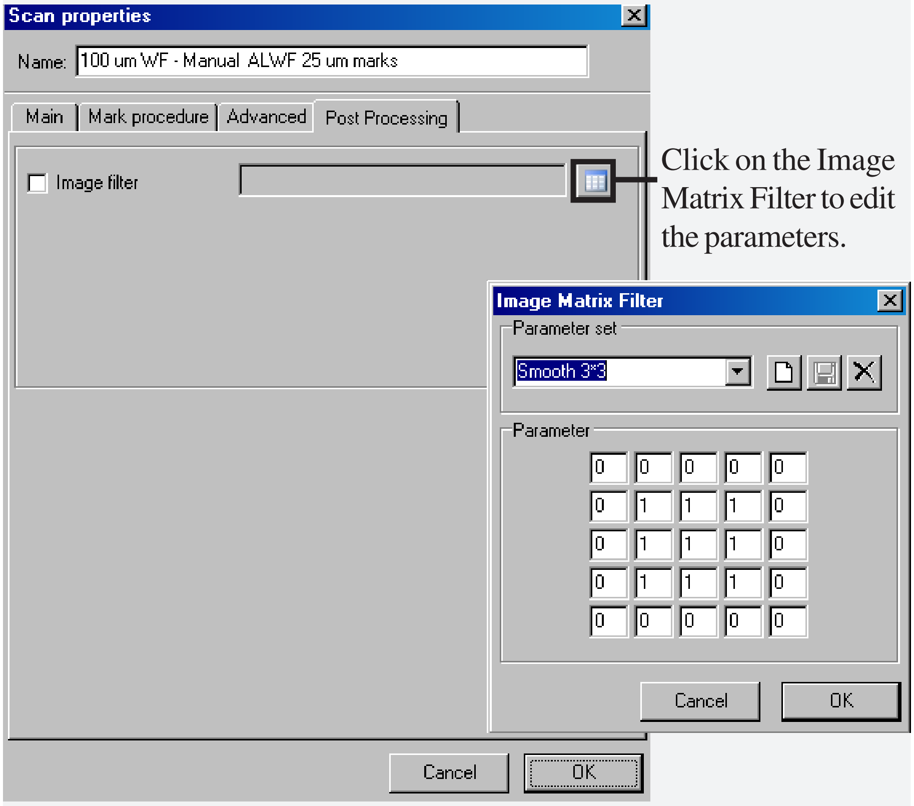
HINT

You can enable the **Auto placement** 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 Writefield.

STEP 6 ▶

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 Software Reference manual). Confirm with **OK**.

Figure 4-6 Post Processing tab.



Scan properties

Name: 100 um WF - Manual ALWF 25 um marks

☐ Image filter

Image Matrix Filter

Parameter set: Smooth 3*3

Parameter

0	0	0	0	0
0	1	1	1	0
0	1	1	1	0
0	1	1	1	0
0	0	0	0	0

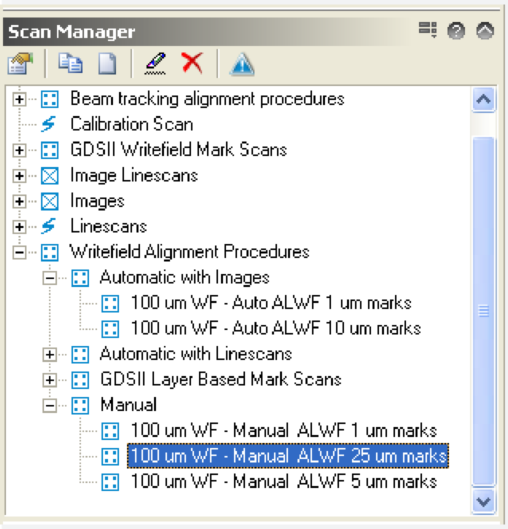
Click on the Image Matrix Filter to edit the parameters.

Task 3 Executing the alignment procedure manually

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 ▶

Highlight the procedure in the **Scan Manager** window, then press **F9**. A **positionlist** will be opened and executed automatically.



Scan Manager

- Beam tracking alignment procedures
- Calibration Scan
- GDSII Writefield Mark Scans
- Image Linescans
- Images
- Linescans
- Writefield Alignment Procedures
  - Automatic with Images
    - 100 um WF - Auto ALWF 1 um marks
    - 100 um WF - Auto ALWF 10 um marks
  - Automatic with Linescans
  - GDSII Layer Based Mark Scans
  - Manual
    - 100 um WF - Manual ALWF 1 um marks
    - 100 um WF - Manual ALWF 25 um marks
    - 100 um WF - Manual ALWF 5 um marks

Figure 4-7 Highlight the procedure in **Scan Manager** window.

STEP 2 ▶

Firstly, the stage will move 37.45 µm in U and V towards the first corner and an image will be scanned at the reference point. The image will cover a 25 µm x 25 µm square. The **Mark** window will be automatically opened, in which the particle should be visible.
The **green cross** shows the position at which the mark is expected.

HINT

If no mark shows up, confirm the **Continue** prompt. Repeat the task, now choosing a larger scan 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 the mouse button and a blue cross will be displayed at the selected position.

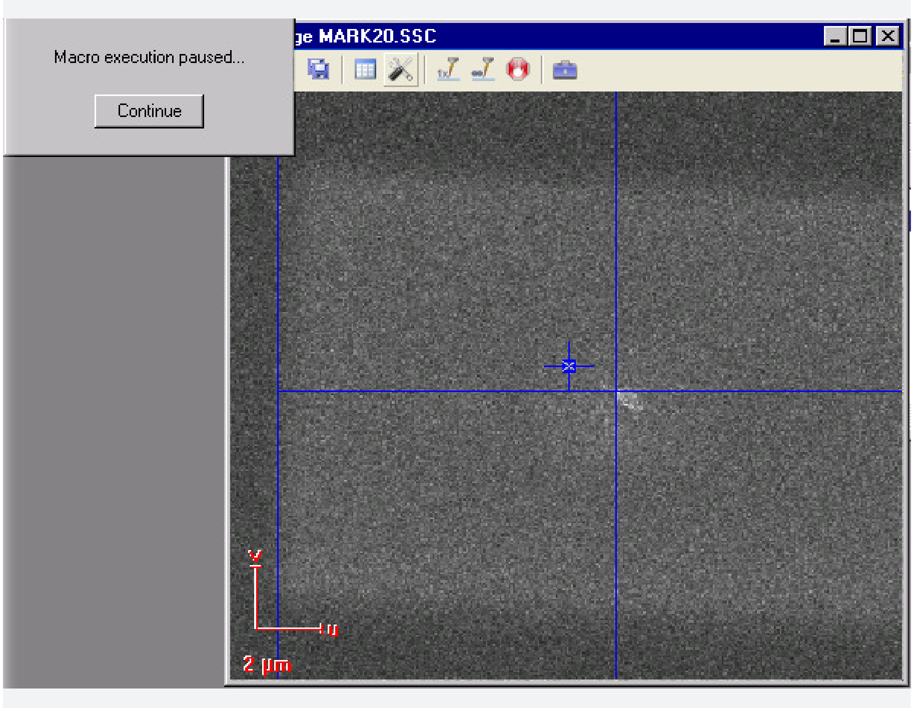


Figure 4-8 Executing the **Positionlist** procedure.

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.

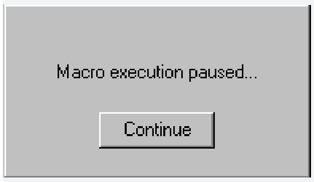
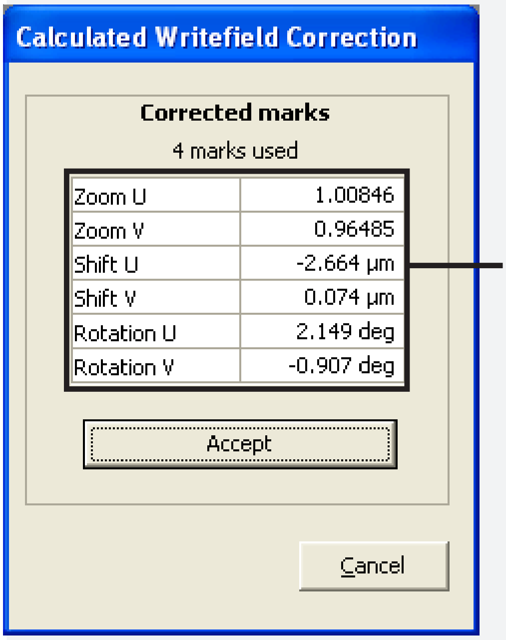


Figure 4-9 **Macro execution** paused while stage moves to new position.

STEP 5 ▶

At the end of the procedure a dialog window opens and the Writefield correction must be confirmed. Note the values of the **Writefield Alignment** for **Zoom**, **Shift** and **Rotation** in UV and confirm if the values are acceptable.

Figure 4-10 Confirming the values for the Writefield correction.



Calculated Writefield Correction

Corrected marks

4 marks used

Zoom U	1.00846
Zoom V	0.96485
Shift U	-2.664 µm
Shift V	0.074 µm
Rotation U	2.149 deg
Rotation V	-0.907 deg

Values for Zoom, Shift and Rotation are now displayed.

HINT

The left column of numbers shows the alignment parameters before alignment. Here the scaling factors are around 0.96. Due to the alignment procedure, new alignment parameters have been calculated as shown in the right 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 for **Zoom** will be multiplied, whereas the new values for **Shift** and **Rotation** are added to the values displayed in the gray field.

STEP 6 ▶

Go back to the **Scan Manager** 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 Writefield, 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 Writefield Alignment window should be very small or close to 1 for the zoom.

STEP 7 ▶

Activate the **Writefield Manager** window and click on the **Save** icon. 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.

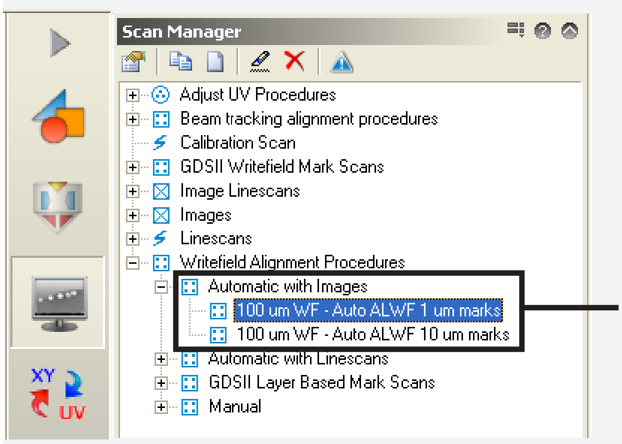
Task 4 Setting up the automated alignment procedure

Once we have performed an alignment procedure manually, with decreasing scan sizes, the final task is now to perform an automated alignment procedure.

STEP 1 ▶

In the **Scan Manager** window double click on **AlignWriteField Procedures** and then double click on **Automatic with Images**.

Figure 4-11 Selecting an **Automated Procedure**.



Select the Automatic with Images procedure.

Double click on the procedure **100µm WF-Auto ALWF 1µm marks** to open the **Scan Properties** window.

STEP 2 ▶ Select the **Main** tab and enter the value **16** for the **Point average**.

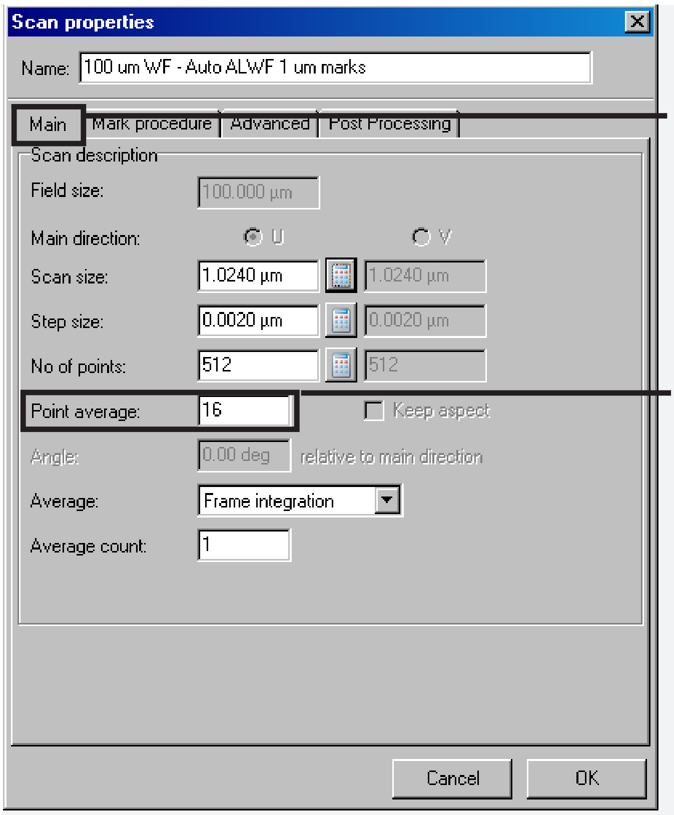


Figure 4-12 The Main tab in the Scan properties window.

Scan properties

Name: 100 um WF - Auto ALWF 1 um marks

Scan description

Field size: 100.000 µm

Main direction: ● U ○ V

Scan size: 1.0240 µm 1.0240 µm

Step size: 0.0020 µm 0.0020 µm

No of points: 512 512

Point average: 16 ☐ Keep aspect

Angle: 0.00 deg relative to main direction

Average: Frame integration

Average count: 1

Select the Main tab.

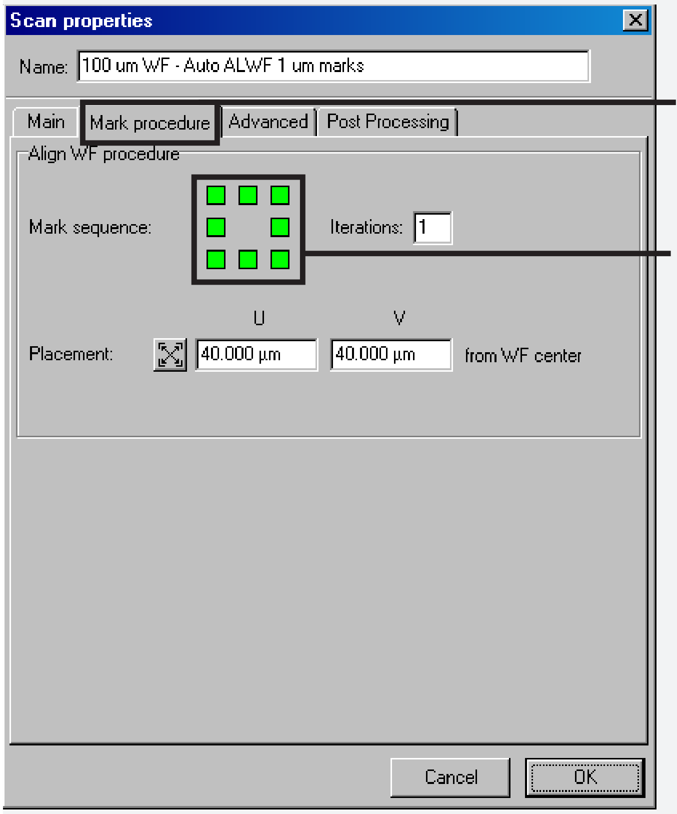
Enter the number 16 for Point average.

HINT

It is recommended that the **No of points** is either 256, 512, 1024 etc, otherwise the Writefield Alignment procedure might be slow.

STEP 3 ▶ Select the **Mark procedure** tab. The **Marked sequence** is displayed. Select all eight marks.

Figure 4-13 The **Mark procedure** tab in the Scan properties window.



Scan properties

Name: 100 um WF - Auto ALWF 1 um marks

Align WF procedure

Mark sequence: 🟩🟩🟩 / 🟩 🟩 / 🟩🟩🟩

Iterations: 1

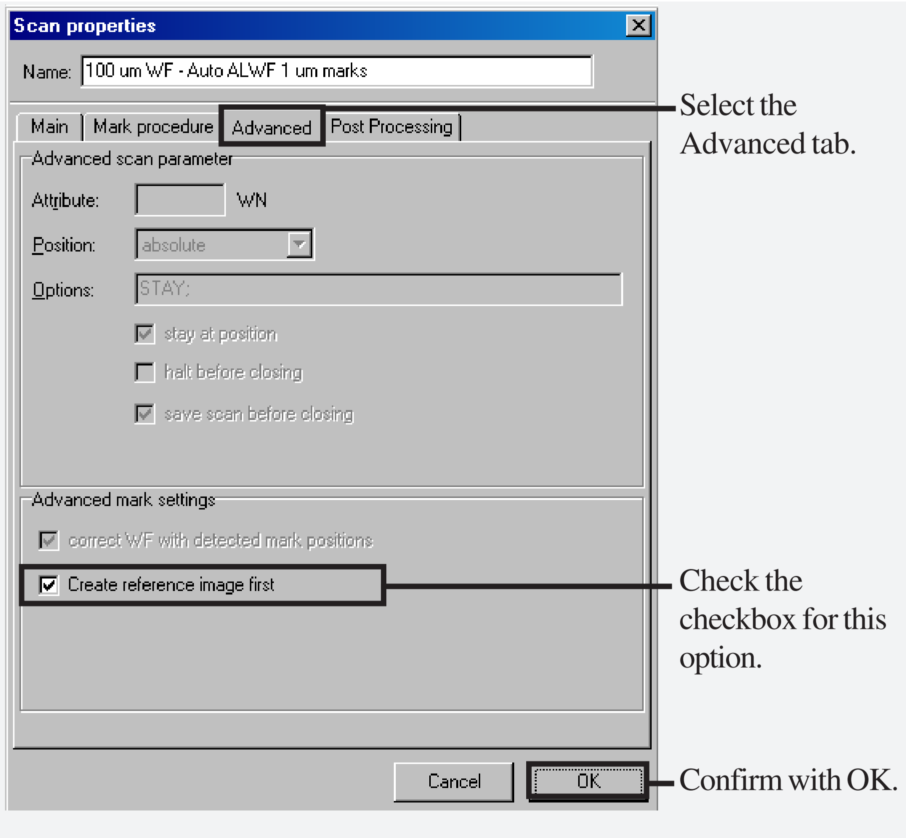
Placement: U 40.000 µm V 40.000 µm from WF center

Select the Mark procedure tab.

Select all eight marks.

HINT It is important that the placement does not exceed 80% of the overall Writefield size. For example, the placement for a 100 µm Writefield should be a maximum of 40 µm. The distance of 40 µm is measured in both directions from the center, yielding a total of 80 µm, which is equal to 80% of the 100 µm Writefield.

STEP 4 ▶ Select the **Advanced** tab and make sure that the **Create reference image first** option is checked.



Scan properties

Name: 100 um WF - Auto ALWF 1 um marks

Advanced scan parameter

Attribute: WN

Position: absolute

Options: STAY;

☑ stay at position

☐ halt before closing

☑ save scan before closing

Advanced mark settings

☑ correct WF with detected mark positions

☑ Create reference image first

Select the Advanced tab.

Check the checkbox for this option.

Confirm with OK.

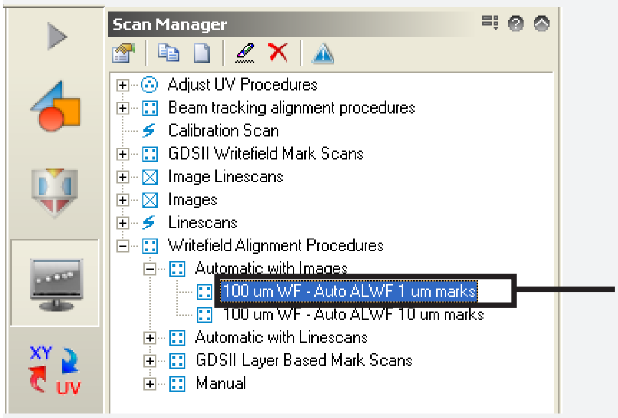
Figure 4-14 Selecting an **Automated Procedure**.

Finally, confirm with **OK**.

STEP 5 ▶

To execute an automated Writefield alignment, click in the **Scan Manager** window on the required procedure name to highlight it. Press **F9** on the keyboard to execute the process.

Figure 4-15 Executing the **Automated Writefield** alignment.



Highlight the required procedure, then press F9 to execute the procedure.

HINT

After the procedure is highlighted in the Scan Manager window, F9 on the keyboard will automatically open the positionlist and execute the alignment procedure.

Task 5 Checking the precision of the alignment procedure

HINT

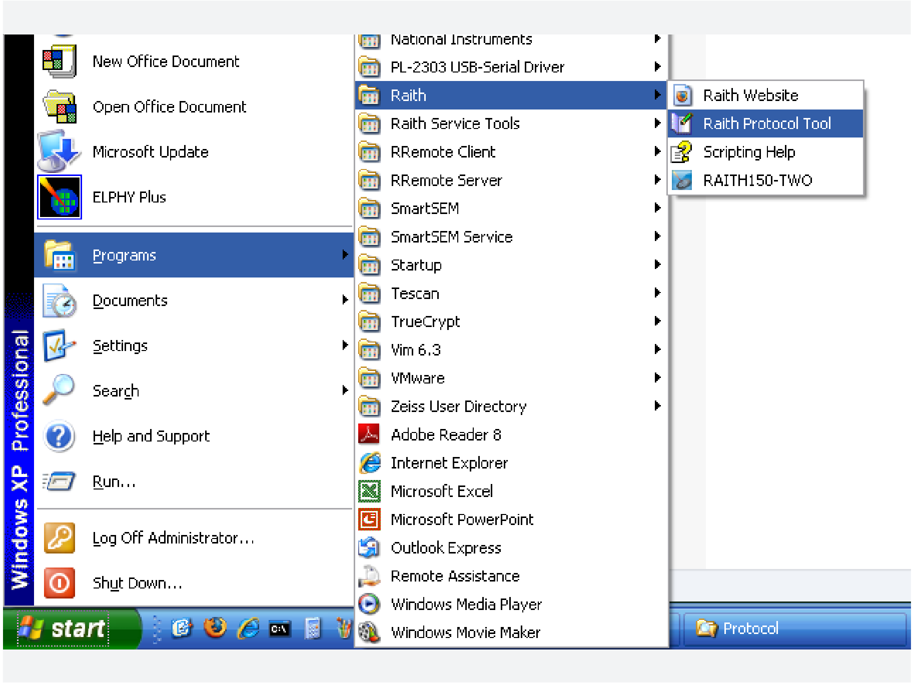
After you have completed the automated alignment procedure, it is highly recommended to open the RAITH protocol and check the variance within the last few alignment procedures.

STEP 1 ▶

Opening the RAITH protocol to view the variance.

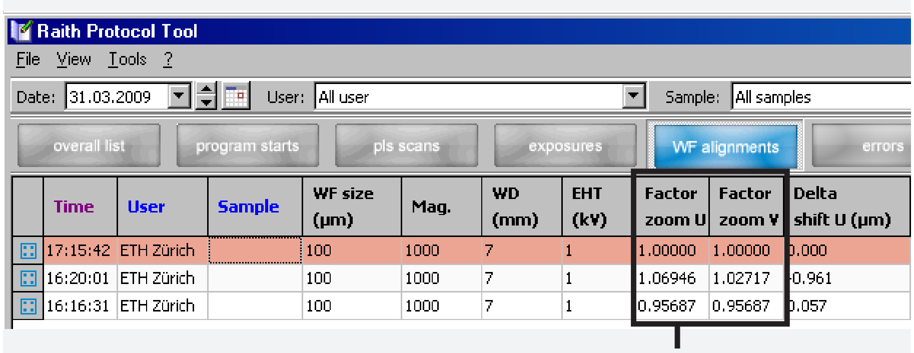
To open the **Protocol**, go to **Windows start > Programs > Raith > Raith Protocol Tool**

Figure 4-16 Opening the RAITH Protocol.



For example, if you are using a 100 µm Writefield, you can check the alignment precision in the fields **zoom U** and **zoom V**.

Figure 4-17 Checking the values in the **Raith Protocol Tool**.



Date: 31.03.2009 User: All user Sample: All samples

	Time	User	Sample	WF size (µm)	Mag.	WD (mm)	EHT (kV)	Factor zoom U	Factor zoom V	Delta shift U (µm)
⊞	17:15:42	ETH Zürich		100	1000	7	1	1.00000	1.00000	.000
⊞	16:20:01	ETH Zürich		100	1000	7	1	1.06946	1.02717	0.961
⊞	16:16:31	ETH Zürich		100	1000	7	1	0.95687	0.95687	.057

Check the values for zoom U and zoom V.

In our example, excellent precision has been achieved. Zoom U shows five decimal places. The variance should be within the last decimal place. For example, if Zoom U shows a value of 1.00001, then the variance would only be 10 nm, which is an excellent value in a 100 µm field.

HINT

For some applications, you may not need such high precision in the Writefield Alignment procedure.

4.2 Writefield Alignment using FBMS and Beam Tracking

FBMS and Beam Tracking are options. If they are not installed, please continue with the next chapter.

Task 1 Continue with located particles

Continue with the located particles, as described in 4.1, Standard Writefield Alignment.

HINT

It is important not to change the column settings.

Task 2 Defining the alignment procedure

Once the alignment procedures have been completed, the **calibrated beam** can now be used to define the stage movements and so calibrate beam tracking.

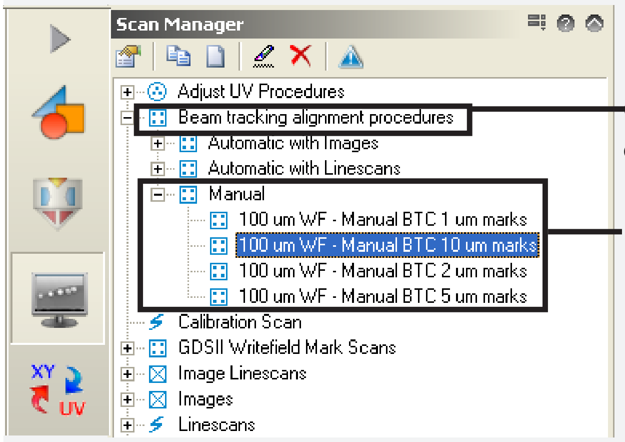
For the beam tracking calibration, the previous Writefield calibration is required. This procedure is comparable to the procedure for the Writefield alignment, using a particle for calibration of scaling, shift and rotation. In the **Scan Manager**, the parameters for this procedure can be stored and recalled for later use.

In this procedure, the calibrated beam remains fixed and the stage will be moved instead. This calibration procedure avoids stitching of large scale patterns.

STEP 1 ▶ Go to the **Scan Manager** as described in Chapter 4.1, The Scan Manager window opens automatically when the **Writefield Control** icon is selected in the control bar.

STEP 2 ▶ Select the required pre-defined **Alignment Procedure**.

Figure 4-18 Select the **Manual BTC** procedure.

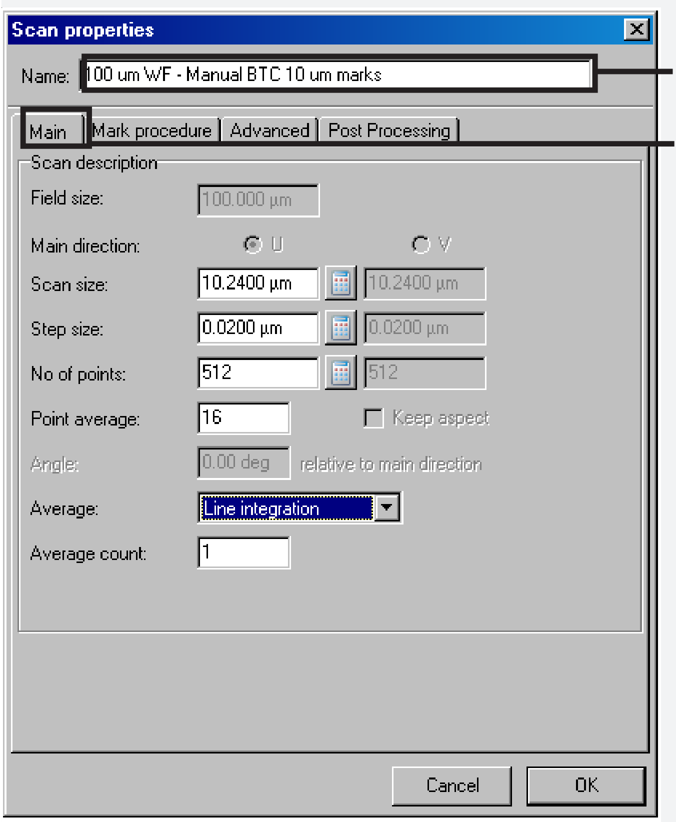


Open Beam tracking calibration procedures.

Select the pre-defined alignment procedure.

HINT

Beam tracking speeds up the writing procedure. For example, if you wish to stitch several fields, with the stage moving to a new position each time, there is normally a waiting time (delay) for final precise positioning of the stage.



Name: 100 um WF - Manual BTC 10 um marks

Scan description

Field size: 100.000 µm

Main direction: ● U ○ V

Scan size: 10.2400 µm 10.2400 µm

Step size: 0.0200 µm 0.0200 µm

No of points: 512 512

Point average: 16 ☐ Keep aspect

Angle: 0.00 deg relative to main direction

Average: Line integration

Average count: 1

Figure 4-19 The **Manual BTC** procedure file is displayed in the **Scan properties** window.

A beam tracking procedure has been selected.

Select the Main tab.

If you select beam tracking, the software automatically calculates the movement of the stage and compensates the movement with a counter-movement of the beam. The delay is thus not required for the new area and writing can start immediately. The whole procedure is therefore much quicker.

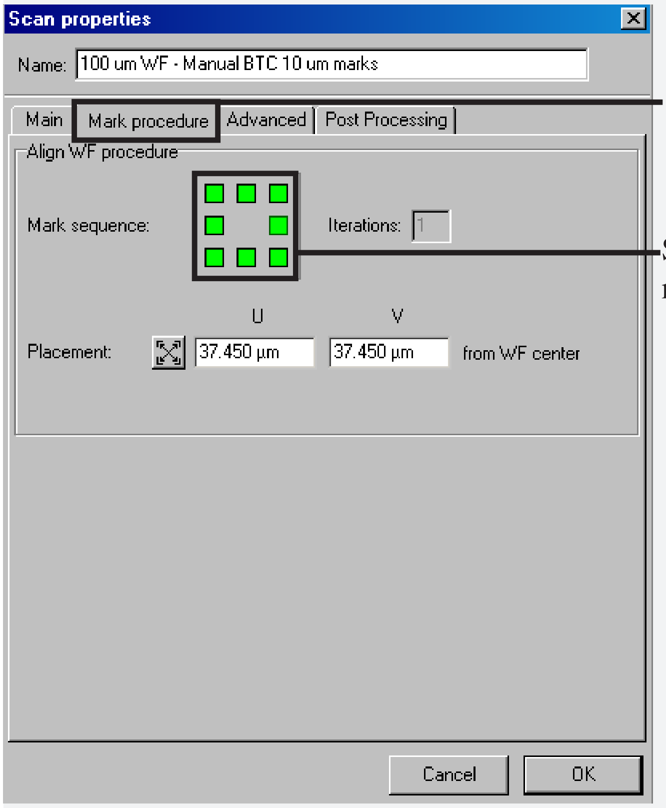
HINT

If you choose to use FBMS, the feature will allow you to write large fields without stitching. It is still of advantage to use beam tracking, as this will eliminate the waiting (delay) time for the stage after movements, since the software will compensate for final precise positioning of the stage after the stage has been driven to its new coordinates.

STEP 3 ▶

Choose the **Mark procedure** tab. Check the **Mark sequence** as shown in the example. For the **Placement** parameter, enter 37.450 µm in U and V.

Figure 4-20 The Mark procedure tab in Scan properties window.



Name: 100 um WF - Manual BTC 10 um marks

Align WF procedure

Mark sequence: 🟩🟩🟩 🟩 🟩 🟩🟩🟩

Iterations: 1

Placement: U 37.450 µm V 37.450 µm from WF center

Select the Mark procedure tab.

Select all eight marks.

STEP 4 ▶

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 Software Reference manual). Confirm with **OK**.

Task 3 Executing the alignment procedure

Follow the description of Task 3 in chapter 4.1.

Task 4 Setting up the automated alignment

Follow the description of Task 4 in chapter 4.1.

Task 5 Checking the precision of the alignment

Follow the description of Task 5 in chapter 4.1.

5 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^{TM}, 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
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

Task 1 Creating a design

STEP 1 ▶

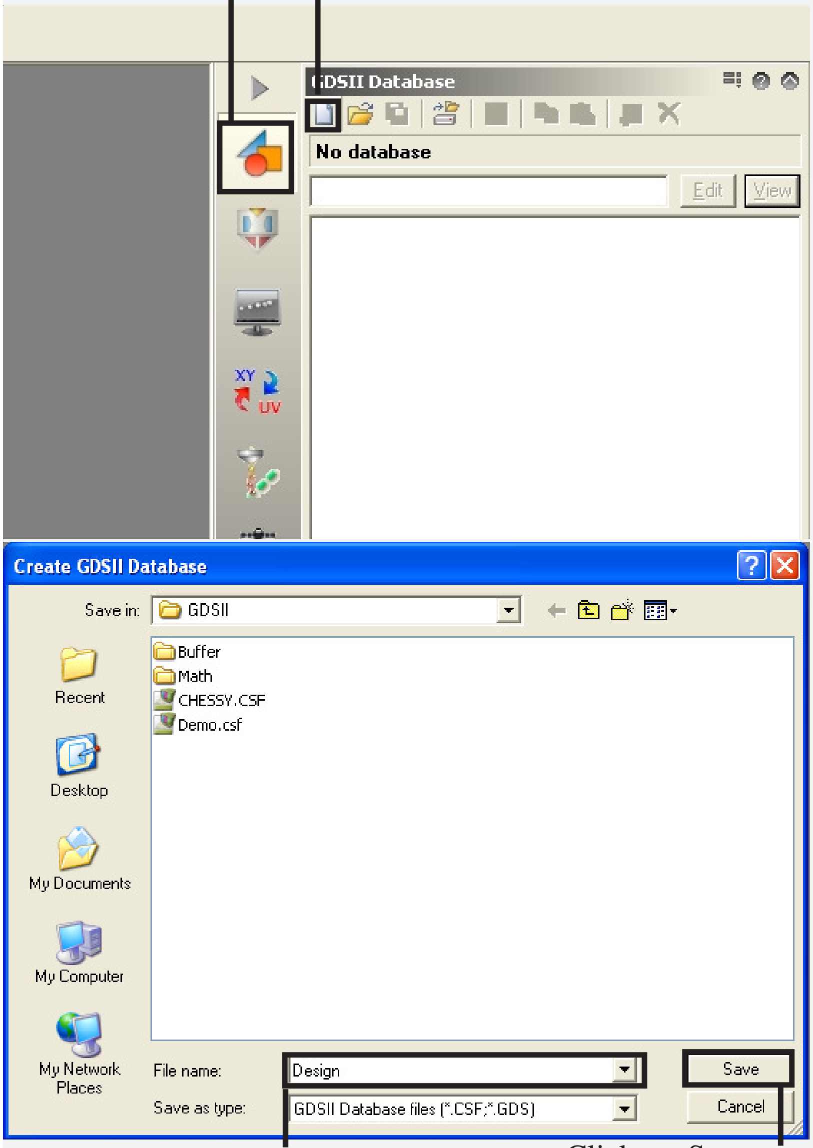
To open the **GDSII Database** window, click on the corresponding **Design** icon in the control bar. To create a new design, click on the corresponding **New** icon.

A new window, **Create GDSII Database**, will be displayed. Enter the file name **Design** and click the **Save** button. After saving, you will get an empty GSDII Database with the name **Design.csf.**

Figure 5-1 Opening **GDSII Database** to create a new design.

Click on the Design icon.

Click on this icon to create a new design.



Save in: GDSII

Buffer
Math
CHESSY.CSF
Demo.csf

File name: Design

Save as type: GDSII Database files (*.CSF;*.GDS)

Enter the File name.

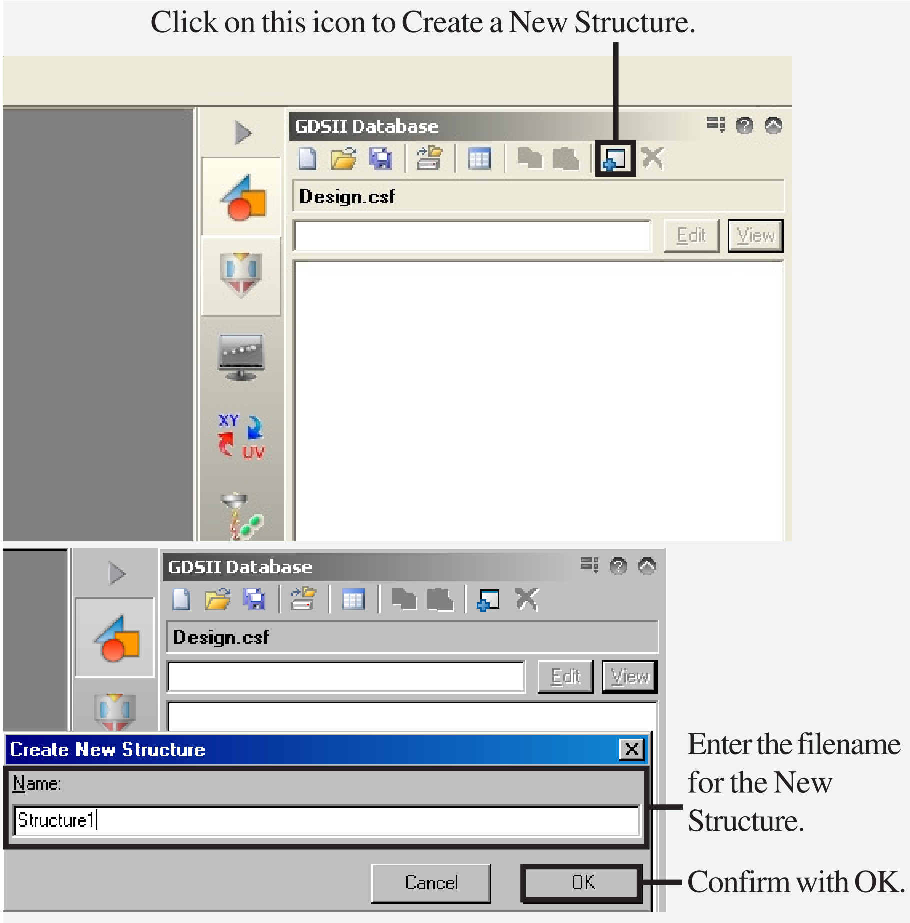
Click on Save to save the new GDSII Databases file.

STEP 2 ▶

The Design.csf file is now displayed in the GDSII Database window.

To create a new structure, click on the corresponding **Create a New Structure** icon.

Figure 5-2 Creating a New Structure.



Click on this icon to Create a New Structure.

GDSII Database

Design.csf

Create New Structure

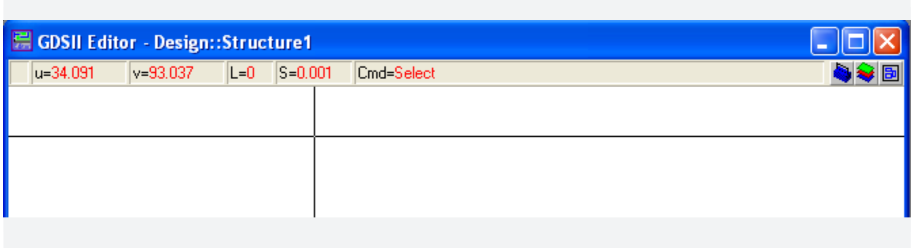
Name: Structure1

Enter the filename for the New Structure.

Confirm with OK.

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.

Figure 5-3 Opening the GDSII Editor.



GDSII Editor - Design::Structure1

u=34.091 v=93.037 L=0 S=0.001 Cmd=Select

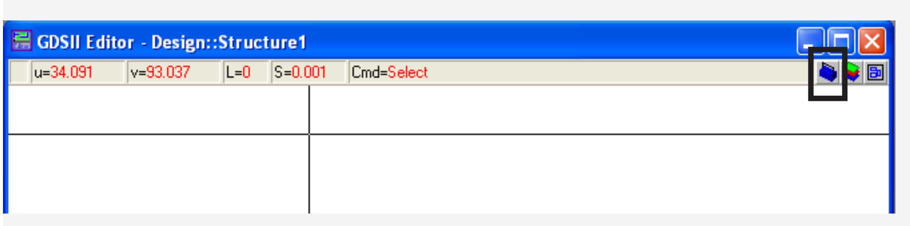
At the top of the Editor a number of pieces of information are given in the status bar:

*: 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 * keys. At the moment the step size is 1 nm, which means that the cursor can only be located at positions with integer nanometres, 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).



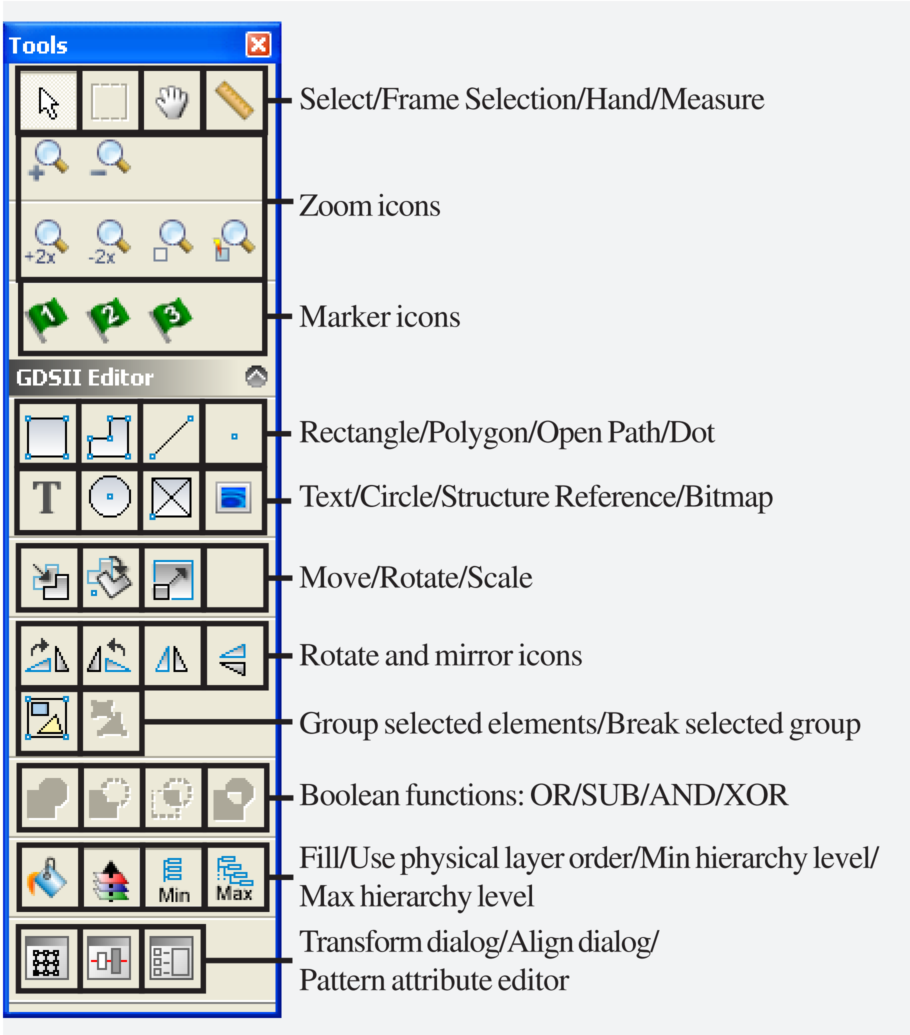
GDSII Editor - Design::Structure1

u=34.091 v=93.037 L=0 S=0.001 Cmd=Select

Figure 5-4 Open the **GDSII Toolbox**.

STEP 2 ▶

The icons of the tool box give easy access to the main design functions.



Select/Frame Selection/Hand/Measure

Zoom icons

Marker icons

Rectangle/Polygon/Open Path/Dot

Text/Circle/Structure Reference/Bitmap

Move/Rotate/Scale

Rotate and mirror icons

Group selected elements/Break selected group

Boolean functions: OR/SUB/AND/XOR

Fill/Use physical layer order/Min hierarchy level/
Max hierarchy level

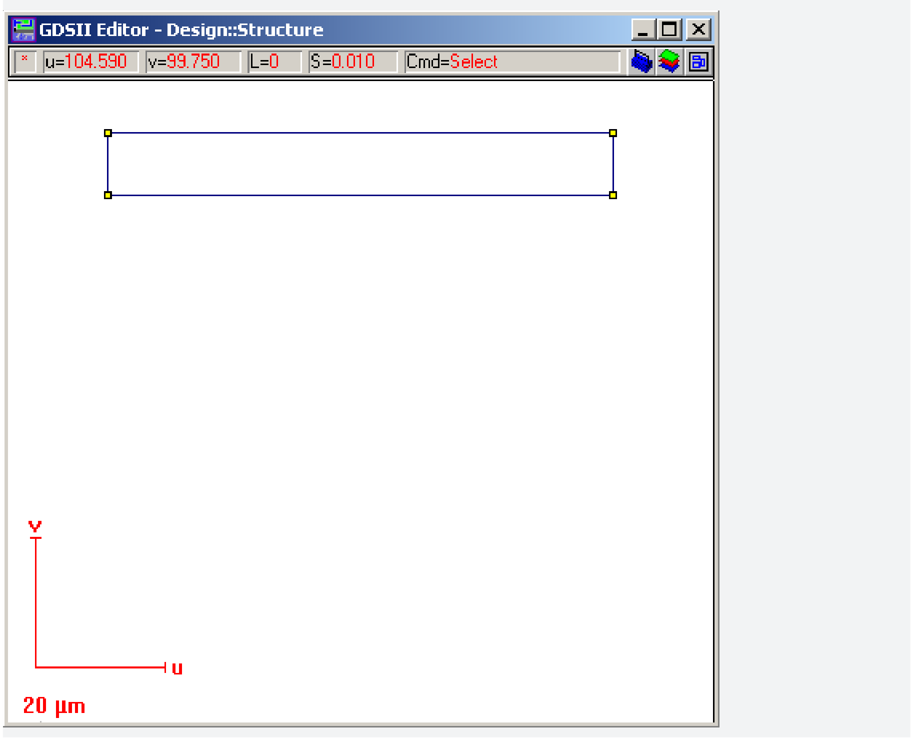
Transform dialog/Align dialog/
Pattern attribute editor

Figure 5-5 Toolbox icons functions.

STEP 3 ▶

Select the **Rectangle** icon. The first mouse click defines one corner of the rectangle and the second mouse click defines the opposite corner. Once the rectangles are completed, choose the red cross icon or press Esc key to cancel the active command.

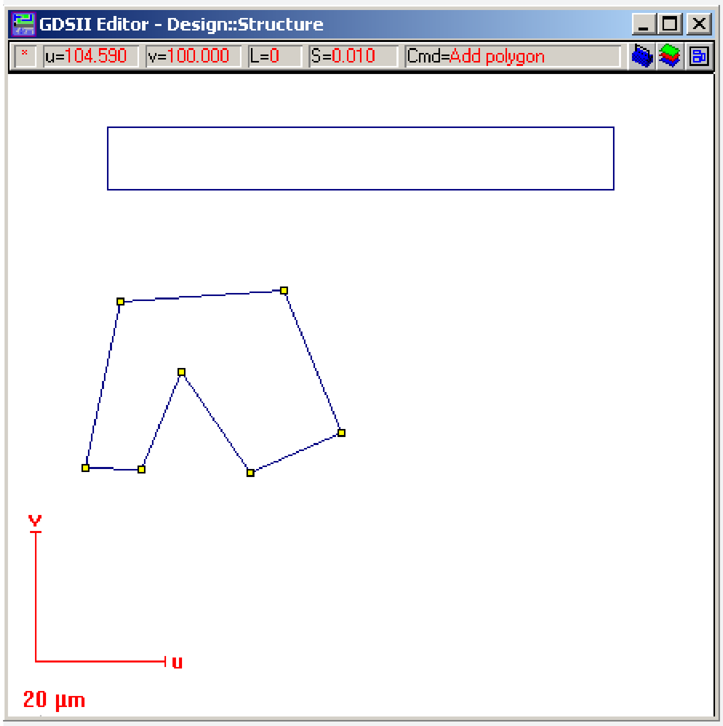
Figure 5-6 Designing a **Rectangle**.



u=104.590 v=99.750 L=0 S=0.010 Cmd=Select

STEP 4 ▶

Draw polygons by activating the corresponding **Polygon** 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. Once the polygons are completed, choose the red cross icon or press the **Esc** key to cancel the active command.



u=104.590 v=100.000 L=0 S=0.010 Cmd=Add polygon

Figure 5-7 Designing a **Polygon**.

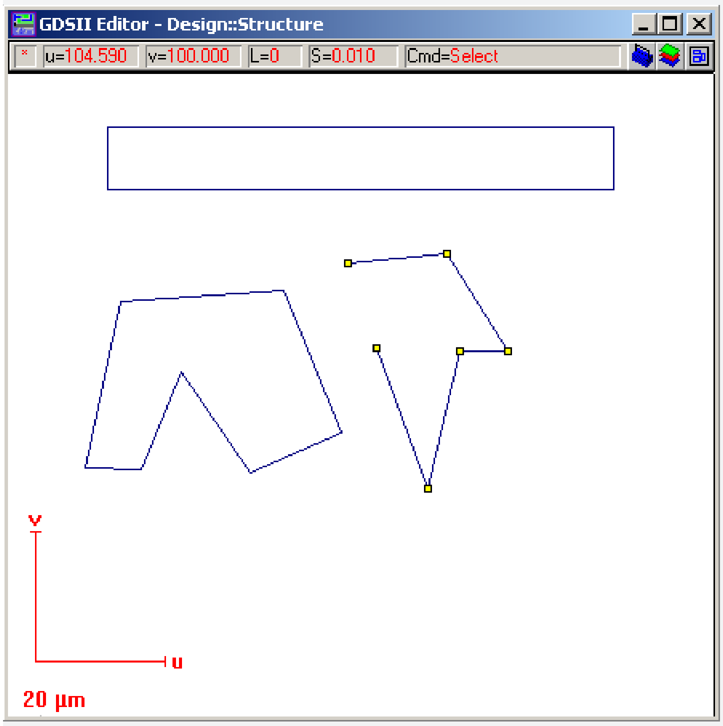
HINT

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.

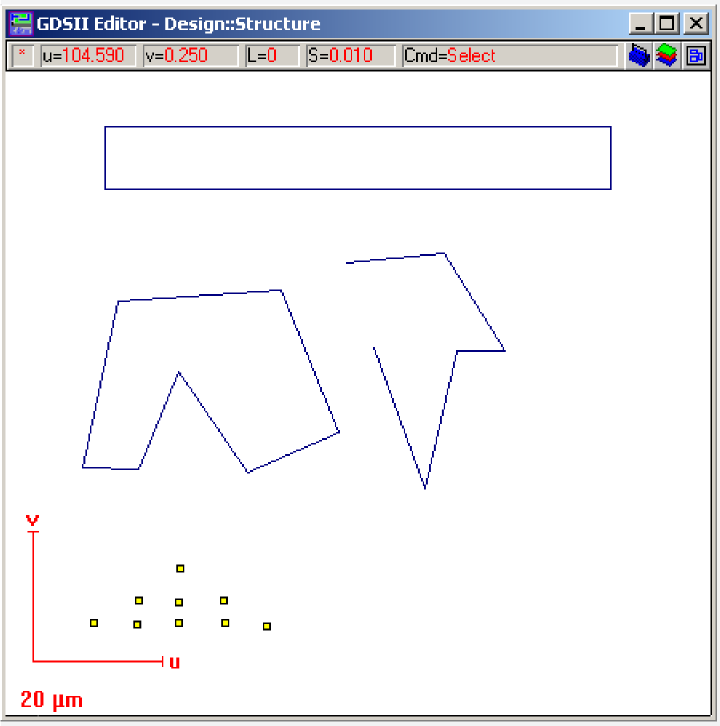
Figure 5-8 Designing an **Open Path**.



STEP 6 ▶

Place dots after clicking the corresponding **Dots** icon, one with each mouse click.

Figure 5-9 Placing **Dots** into the structure.



* u=104.590 v=0.250 L=0 S=0.010 Cmd=Select

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.

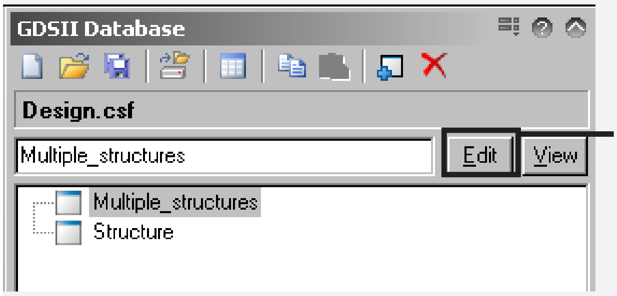
Task 3 Modifying structures

STEP 1 ▶

The next step is to create a new structure in the same database called **Multiple_structures**. Save and close this window.

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

Figure 5-10 Modifying structures in **GDSII Database**.



Click on Edit to modify the pattern.

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 squares.

STEP 3 ▶

Choose **Edit > Copy**. Now open the new design field in the Editor **Multiple_structures**. Choose **Edit > Paste**.

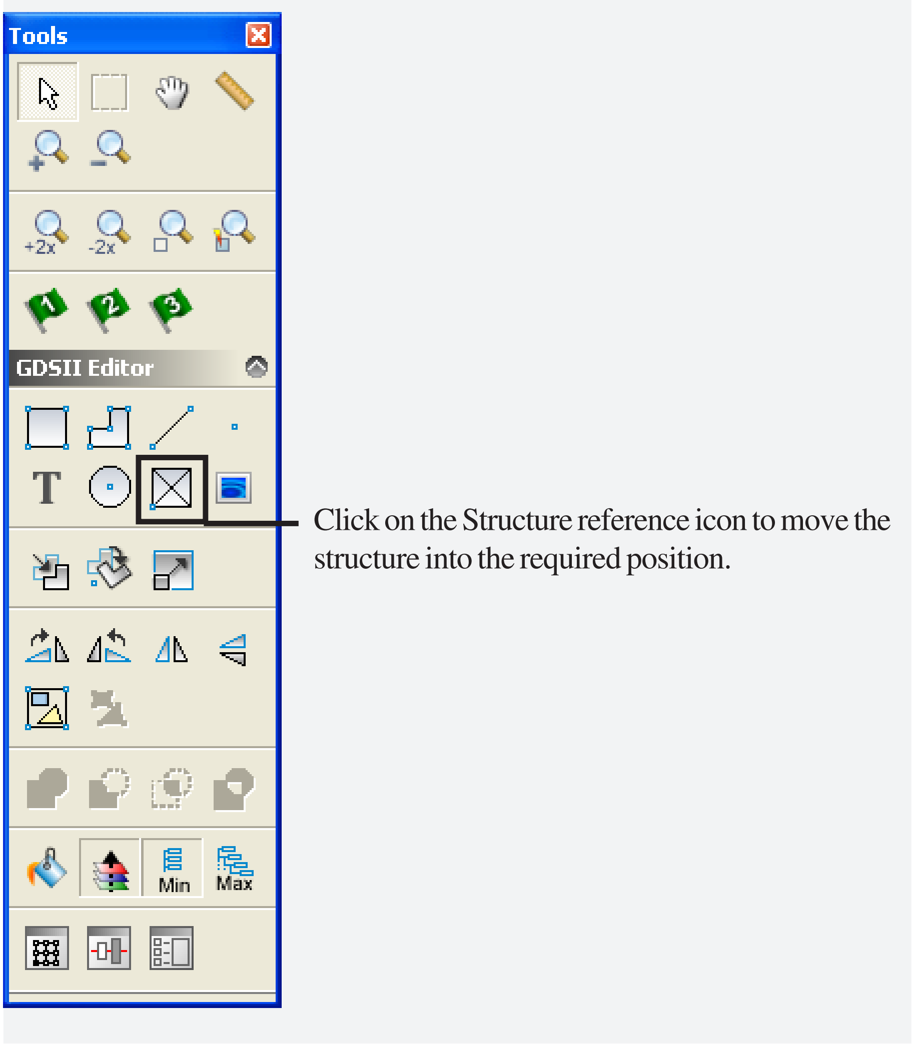
HINT

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.

STEP 4 ▶

Use the corresponding tool button for **Structure reference** to move this structure into the center of the lower left quadrant.

Figure 5-11 Using the Structure reference icon.

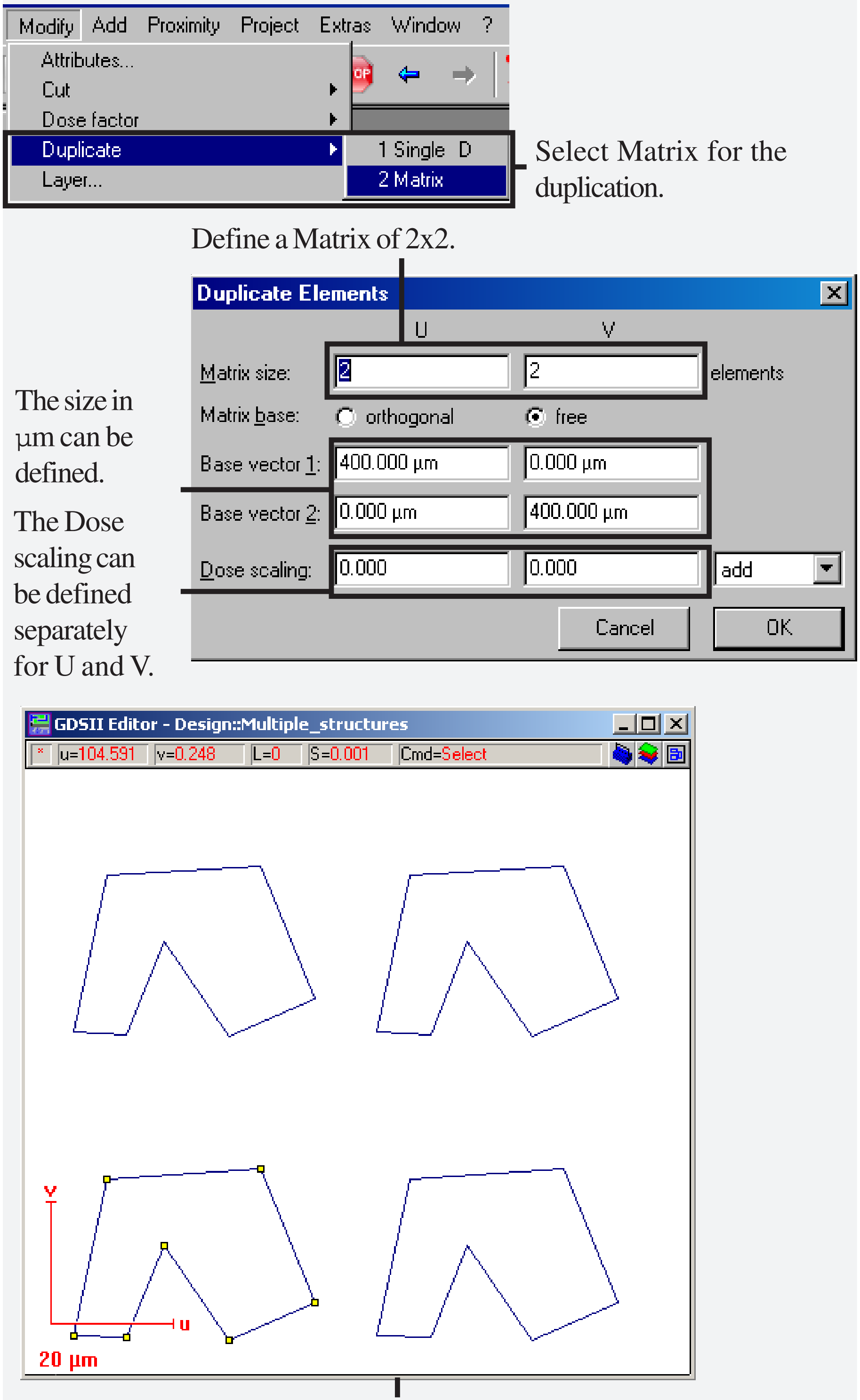


Click on the Structure reference icon to move the structure into the required position.

STEP 5 ▶

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

Figure 5-12 Duplicated structures.



Select Matrix for the duplication.

Define a Matrix of 2x2.

Matrix size: 2 | 2 elements

Matrix base: ○ orthogonal ● free

Base vector 1: 400.000 µm | 0.000 µm

Base vector 2: 0.000 µm | 400.000 µm

Dose scaling: 0.000 | 0.000 | add

u=104.591 v=0.248 L=0 S=0.001 Cmd=Select

The size in µm can be defined.

The Dose scaling can be defined separately for U and V.

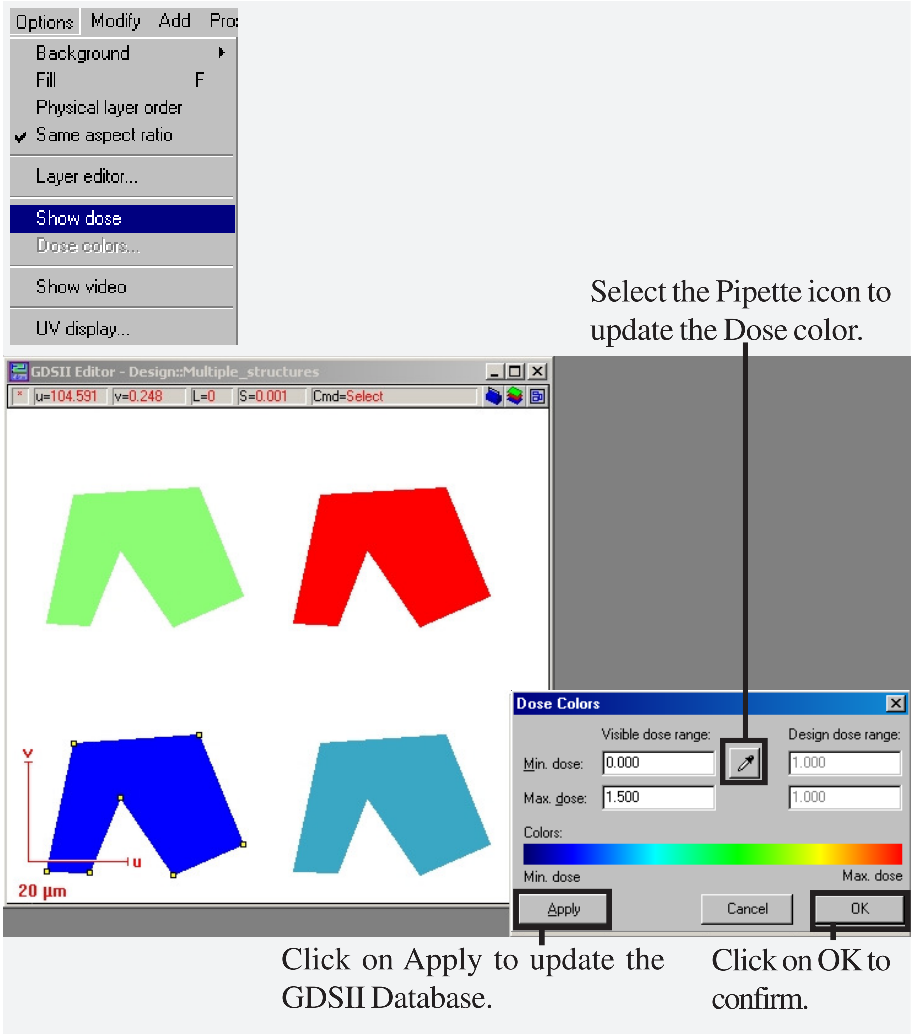
The structure has been duplicated as previously defined.

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.

Figure 5-13 Selecting different **Dose Values**.



u=104.591 v=0.248 L=0 S=0.001 Cmd=Select

Visible dose range:
Min. dose: 0.000
Max. dose: 1.500

Design dose range:
1.000
1.000

Select the Pipette icon to update the Dose color.

Click on Apply to update the GDSII Database.

Click on OK to confirm.

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.

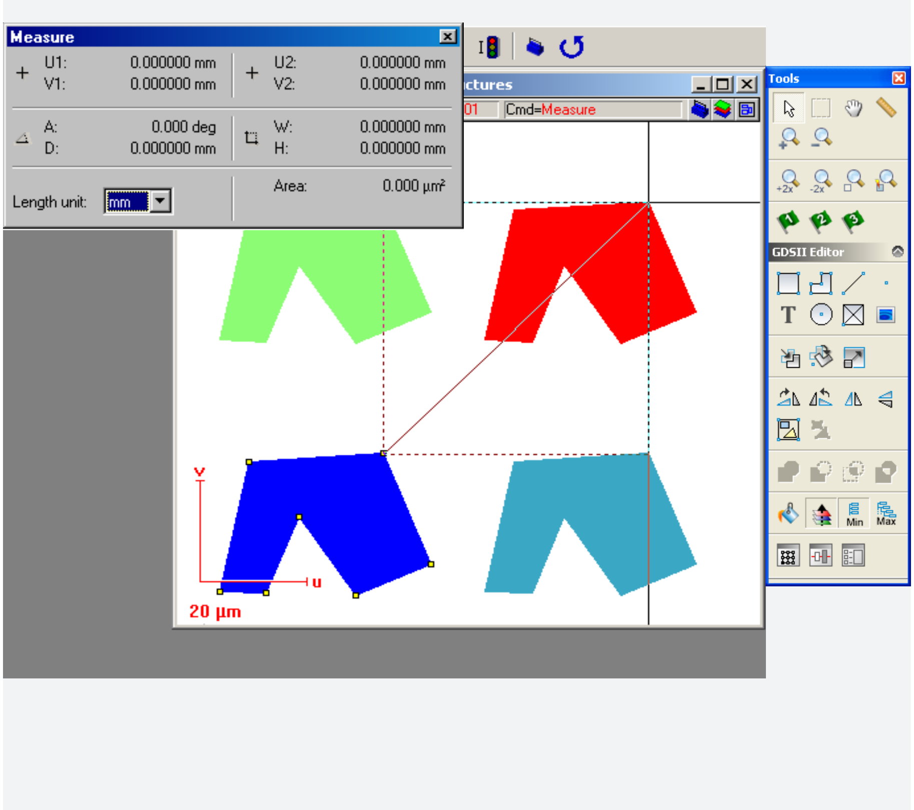
Task 4 Measuring a distance

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.

Figure 5-14

Measuring a distance



U1: 0.000000 mm
V1: 0.000000 mm
U2: 0.000000 mm
V2: 0.000000 mm
A: 0.000 deg
D: 0.000000 mm
W: 0.000000 mm
H: 0.000000 mm
Area: 0.000 µm²
Length unit: mm

Cmd=Measure

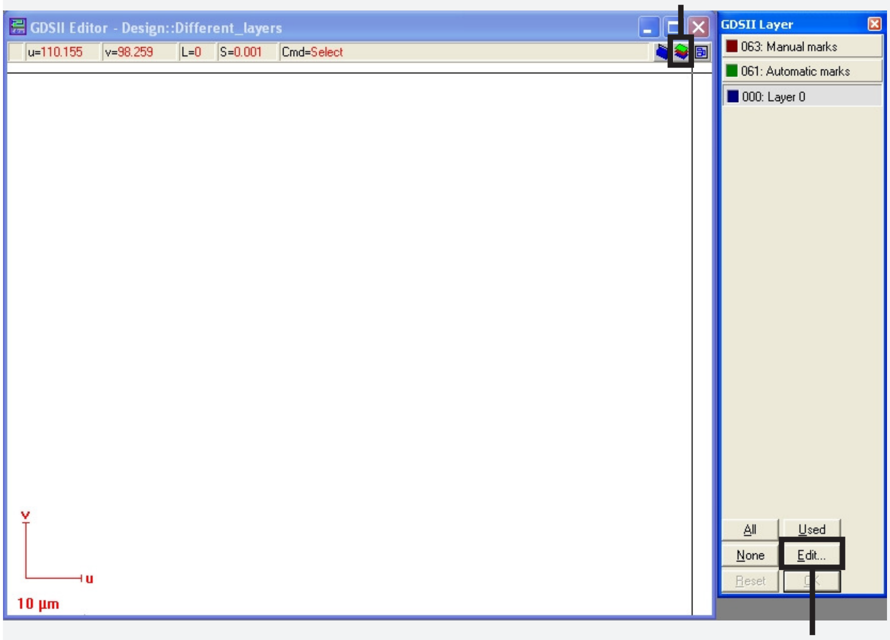
Task 5 Placing of elements in different layers

STEP 1 ▶

Create a new structure, we have named it here **Different_layers**. Click the **Layer** icon next to the toolbox. A dialog window will open, showing the existing layers. Click **Edit** and a new dialog window will open.

Figure 5-15 Creating **Different layers**.

Click on the Layer icon to create different layers.



Click on Edit to open the Layer Properties

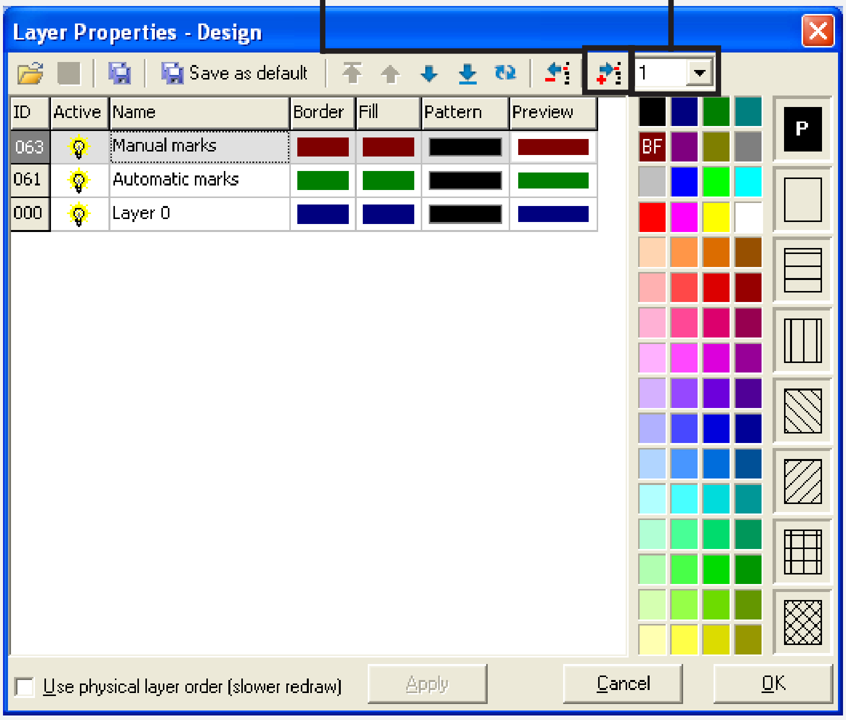
STEP 2 ▶

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.

Figure 5-16 Add a Layer to the structure.

Click on the Add a Layer icon.

Choose the layer number from the dropdown list.



ID	Active	Name	Border	Fill	Pattern	Preview
063	💡	Manual marks	🟥	🟥	⬛	🟥
061	💡	Automatic marks	🟩	🟩	⬛	🟩
000	💡	Layer 0	🟦	🟦	⬛	🟦

☐ Use physical layer order (slower redraw)

The Layer Properties window will now show the new layer.

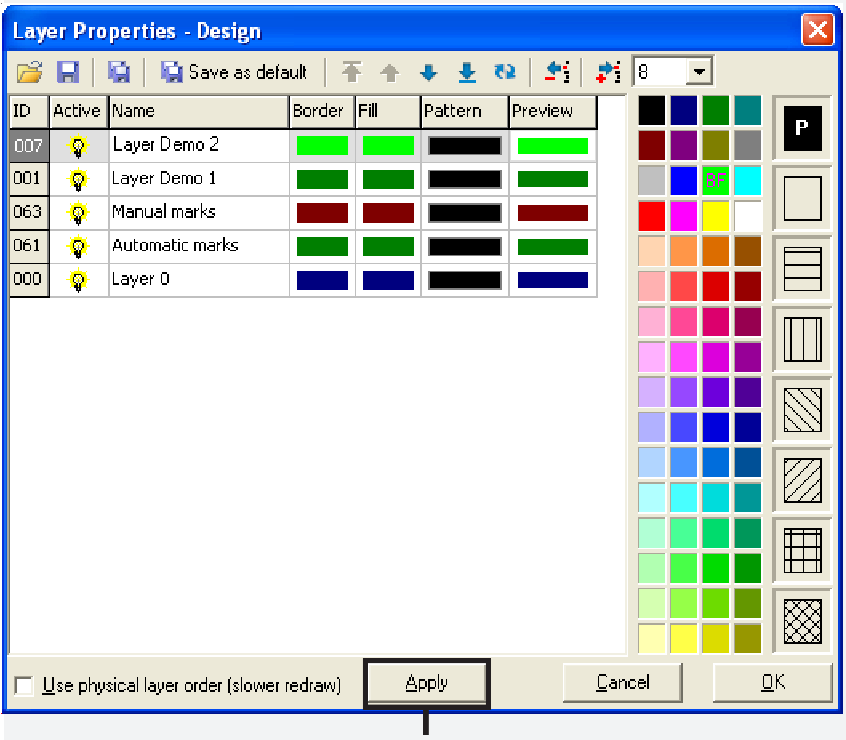
STEP 3 ▶

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.

Figure 5-17 Add a **New Layer**.



ID	Active	Name	Border	Fill	Pattern	Preview
007	💡	Layer Demo 2	🟩	🟩	⬛	🟩
001	💡	Layer Demo 1	🟩	🟩	⬛	🟩
063	💡	Manual marks	🟥	🟥	⬛	🟥
061	💡	Automatic marks	🟩	🟩	⬛	🟩
000	💡	Layer 0	🟦	🟦	⬛	🟦

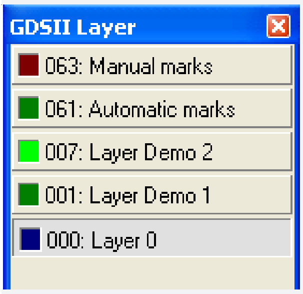
☐ Use physical layer order (slower redraw)

Click on the Apply button.

STEP 5 ▶

Make these layers visible in the **GDSII layer** window by clicking on the **Apply** button.

Figure 5-18 Selecting the **GDSII Layer** in normal window.



🟥 063: Manual marks
🟩 061: Automatic marks
🟩 007: Layer Demo 2
🟩 001: Layer Demo 1
🟦 000: Layer 0

The layers are now visible.

STEP 6 ▶

Choose **OK** and confirm with **Yes** to save the changes.

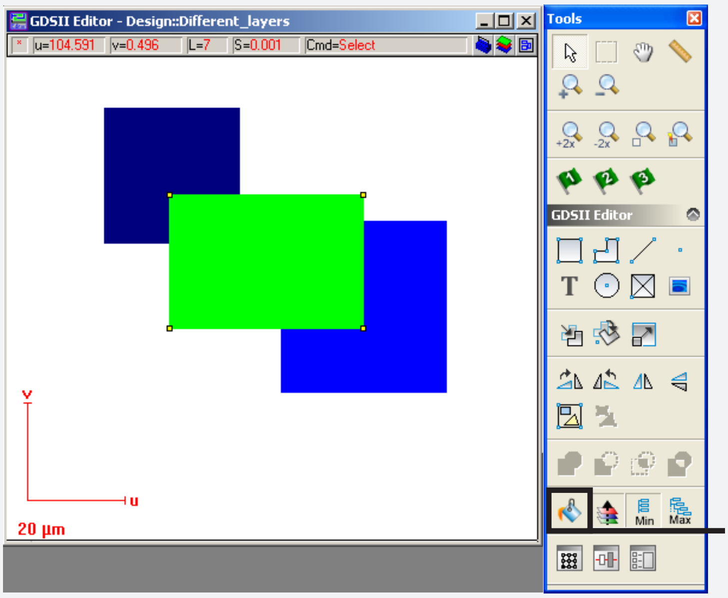
STEP 7 ▶ The active layer is displayed in the top of the **GDSII Editor** window, in our example layer 0. Place a rectangle in this layer.

STEP 8 ▶ Choose **Add > Preset > Layer > Show All** from the dialog window. Choose Layer 1 and confirm with OK. Layer 1 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.

Figure 5-19 Creating an Active layer.



GDSII Editor - Design::Different_layers

u=104.591 v=0.496 L=7 S=0.001 Cmd=Select

Click on the Filling icon.

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 > Duplicate**, 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**.

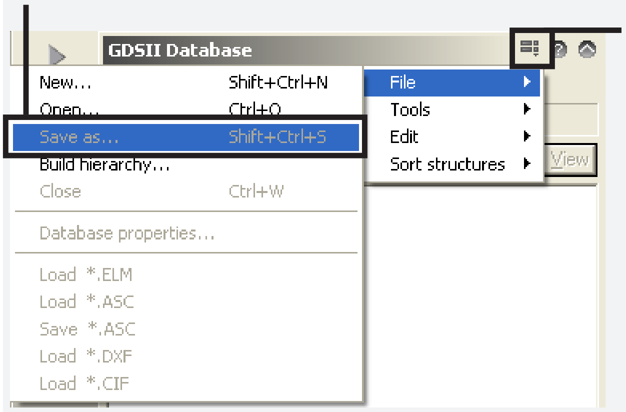


Figure 5-20 GDSII Database.

Select **Save as** .

Click on this button to open the File menu.

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**, select the **New** icon to create a new design enter the filename **ResTest** and click on **Save**.

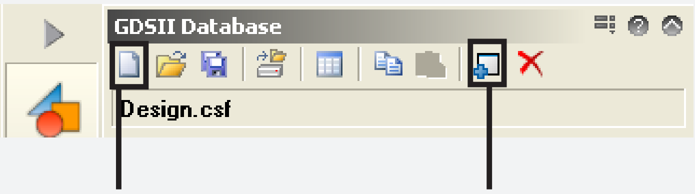


Figure 5-21 Creating a new structure in **GDSII Database.**

Click on the New icon to create a new design.

Click on this icon to Create a New Structure.

STEP 2 ▶

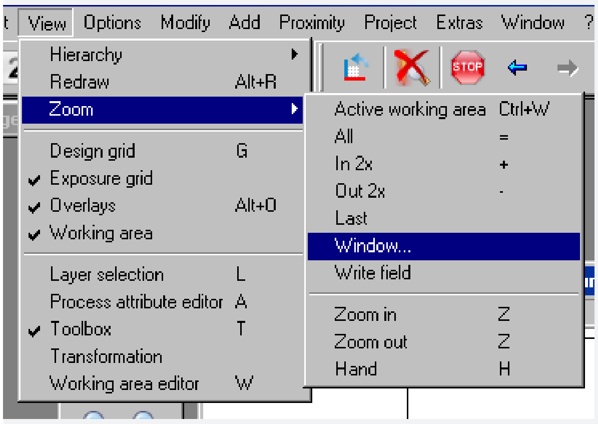
To create a new structure, click on the corresponding **Create a New Structure** icon. Enter filename **RES** and click on **OK**.
The GDSII Editor window opens automatically.

STEP 3 ▶

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

STEP 4 ▶ Choose **View > Zoom > Active working area** from the menu bar.

Figure 5-22 Zoom function to View Window.

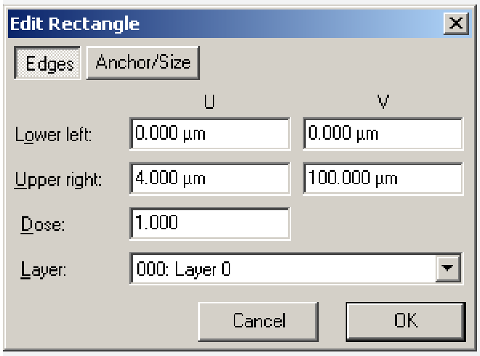


STEP 5 ▶ Select the **Rectangle** icon in the toolbox and draw one rectangle.

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

STEP 7 ▶ Double click inside the rectangle 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.

Figure 5-23 Edit Rectangle.



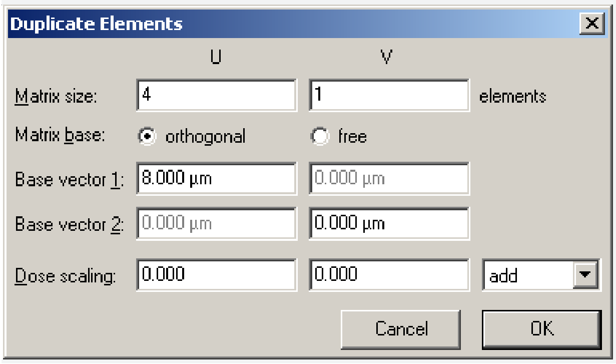
	U	V
Lower left:	0.000 µm	0.000 µm
Upper right:	4.000 µm	100.000 µm
Dose:	1.000	
Layer:	000: Layer 0	

Double click inside the rectangle to edit the parameters. The Edit Rectangle dialog box will open.

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. Finally, click on **OK**.

Figure 5-24 Duplicate Elements.



Duplicate Elements

	U	V	
Matrix size:	4	1	elements
Matrix base:	● orthogonal	○ free	
Base vector 1:	8.000 µm	0.000 µm	
Base vector 2:	0.000 µm	0.000 µm	
Dose scaling:	0.000	0.000	add

Enter the parameters for the Matrix.

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:

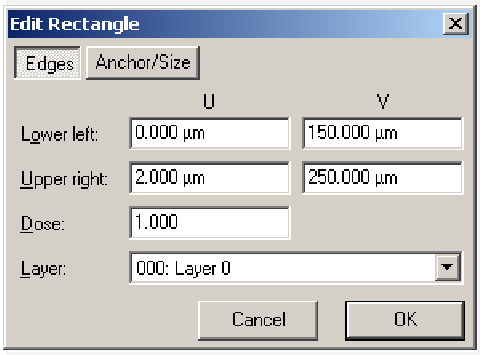
U 0 and 2 µm

V 150 and 250 µm

Layer 0 and Dose 1.

Click on OK.

Figure 5-25 Edit Rectangle.



Edit Rectangle

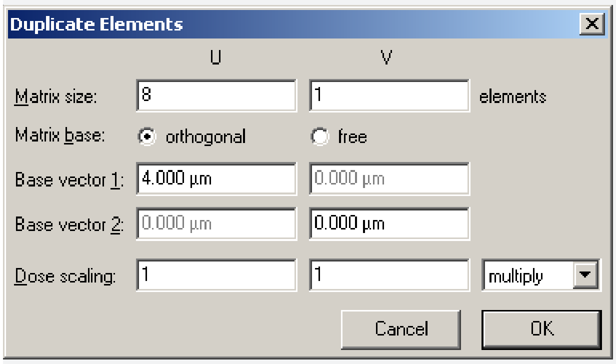
	U	V
Lower left:	0.000 µm	150.000 µm
Upper right:	2.000 µm	250.000 µm
Dose:	1.000	
Layer:	000: Layer 0	

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. Click on **OK**.

The width of the lines as well as the distance between them is now only half compared to the previous grid.

Figure 5-26 Duplicate Elements parameters.



Duplicate Elements

	U	V	
Matrix size:	8	1	elements
Matrix base:	● orthogonal	○ free	
Base vector 1:	4.000 µm	0.000 µm	
Base vector 2:	0.000 µm	0.000 µm	
Dose scaling:	1	1	multiply

Enter the new parameters for the Matrix.

STEP 12 ▶

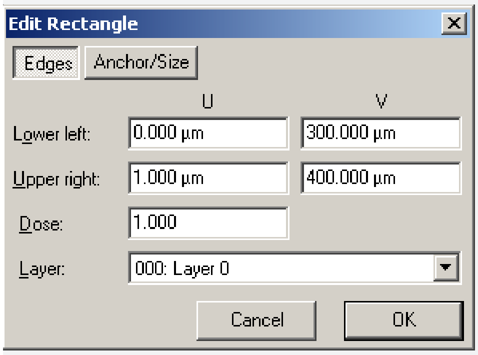
Select the **Rectangle** icon in the GDSII toolbox and draw another rectangle. Click on the Red cross icon (or Esc key) to cancel the repeat command. Double click inside the rectangle and enter the following coordinates:

U 0 and 1 µm

V 300 and 400 µm

Layer 0 and Dose 1.

Figure 5-27 Edit Rectangle parameters.



Edit Rectangle

	U	V
Lower left:	0.000 µm	300.000 µm
Upper right:	1.000 µm	400.000 µm
Dose:	1.000	
Layer:	000: Layer 0	

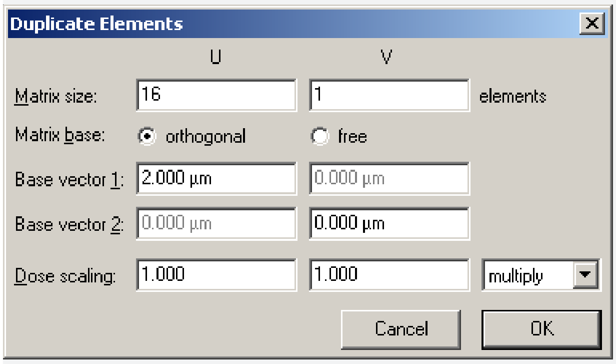
Enter the parameters for the rectangle.

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. Click on **OK**.

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.

Figure 5-28 Altering the **Matrix size** of the elements for duplication.



Duplicate Elements

	U	V	
Matrix size:	16	1	elements
Matrix base:	● orthogonal	○ free	
Base vector 1:	2.000 µm	0.000 µm	
Base vector 2:	0.000 µm	0.000 µm	
Dose scaling:	1.000	1.000	multiply

Change the Matrix size to the new parameters.

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.

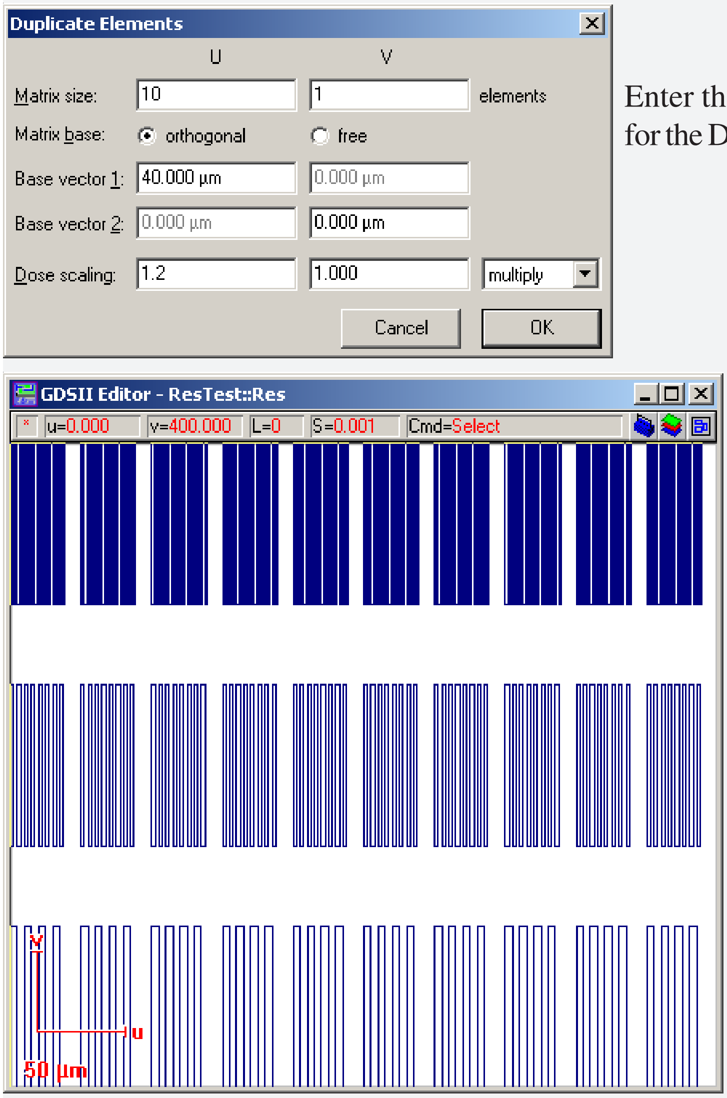


Figure 5-29 Altering the Dose scaling value.

Duplicate Elements

	U	V	
Matrix size:	10	1	elements
Matrix base:	● orthogonal	○ free	
Base vector 1:	40.000 µm	0.000 µm	
Base vector 2:	0.000 µm	0.000 µm	
Dose scaling:	1.2	1.000	multiply

GDSII Editor - ResTest::Res

u=0.000 v=400.000 L=0 S=0.001 Cmd=Select

Enter the new value for the Dose scaling.

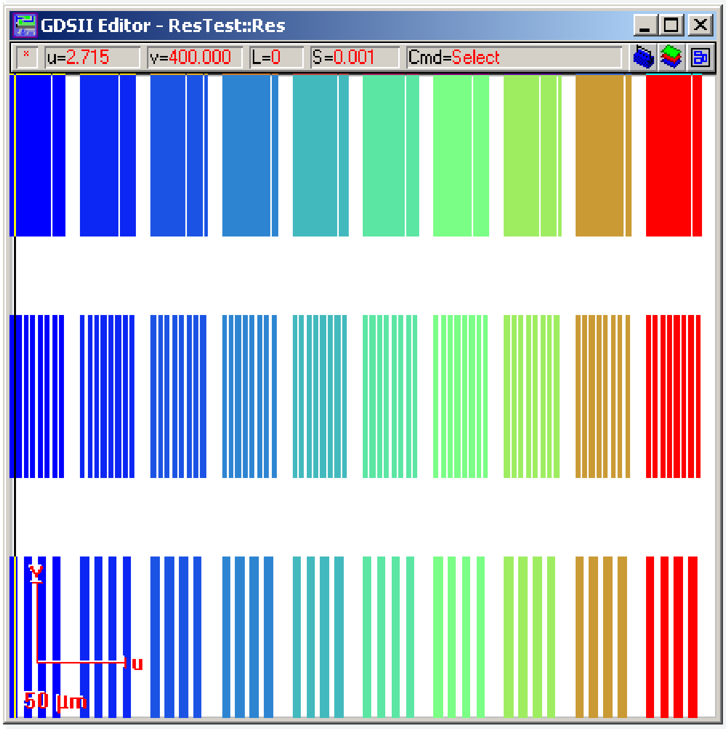
The line structure has now been duplicated, filling the complete working area. The different spacing from row to row can easily be observed.

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**.

Figure 5-30 Saving the structure.



GDSII Editor - ResTest::Res

u=2.715 v=400.000 L=0 S=0.001 Cmd=Select

The new Dose scaling is now displayed.

6 Advanced Pattern Design

AIM

In the previous chapter, we learned how to multiply structures within a matrix so that 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.

6.1 Advanced Pattern Design (Standard)

Chapter 6.1 explains how to design an advanced pattern.

Task 1 Design using hierarchy
Task 2 Studying chessy.csf

6.2 Advanced Pattern Design using FBMS (Option)

Chapter 6.2 is only applicable to users who have the FBMS option installed on their Turnkey System.

Task 1 Designing FBMS elements

6.1 Advanced Pattern Design

Task 1 Design using hierarchy

STEP 1 ▶

Create a new **GDSII Database file** and name it **Hierarchy**. Then create a **Structure** and name it **Cell1**. The **GDSII Editor** will now open automatically, so that you can place several elements within the field of approximately 100 µm. **Save** the structure and close the Editor.

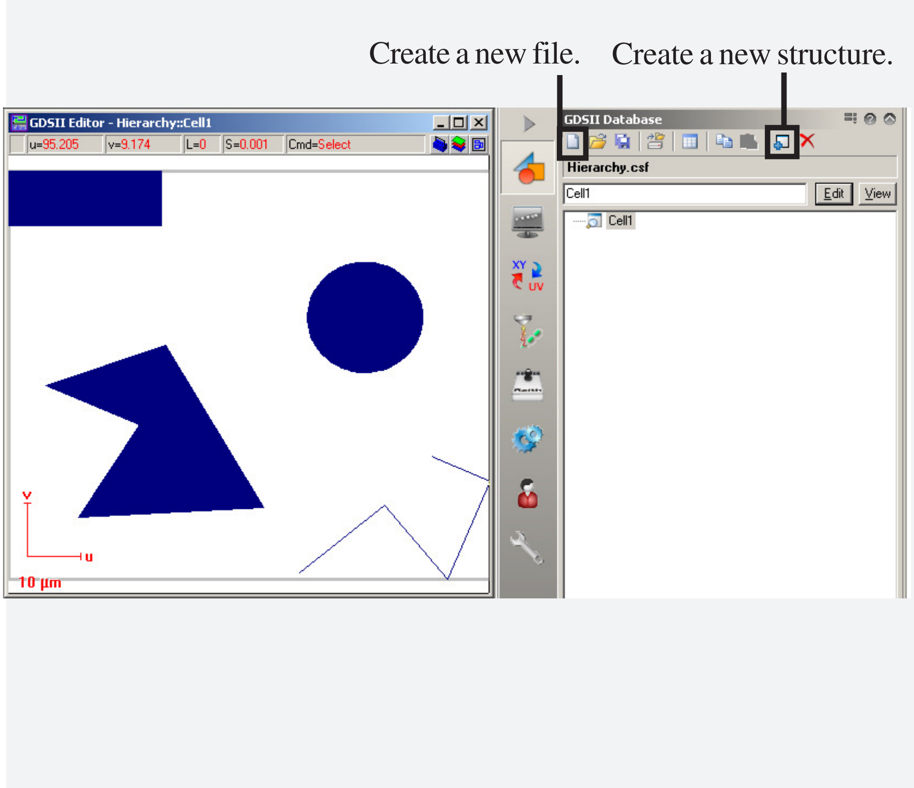
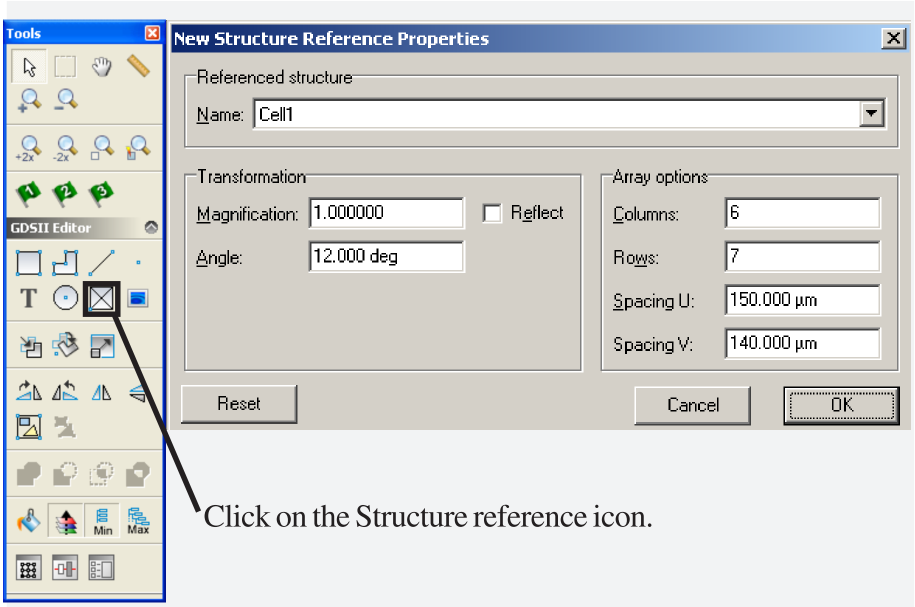


Figure 6-1 Creating a **Hierarchy** structure.

STEP 2 ▶

We will now create a hierarchical structure. Create the structure **Matrix1**. In the **Editor Toolbox** click on the **Structure Reference** icon. The following dialog window will open.

Figure 6-2 The New Structure Reference Properties window.



New Structure Reference Properties

Referenced structure

Name: Cell1

Transformation

Magnification: 1.000000 ☐ Reflect

Angle: 12.000 deg

Array options

Columns: 6

Rows: 7

Spacing U: 150.000 µm

Spacing V: 140.000 µm

Reset Cancel OK

Click on the Structure reference icon.

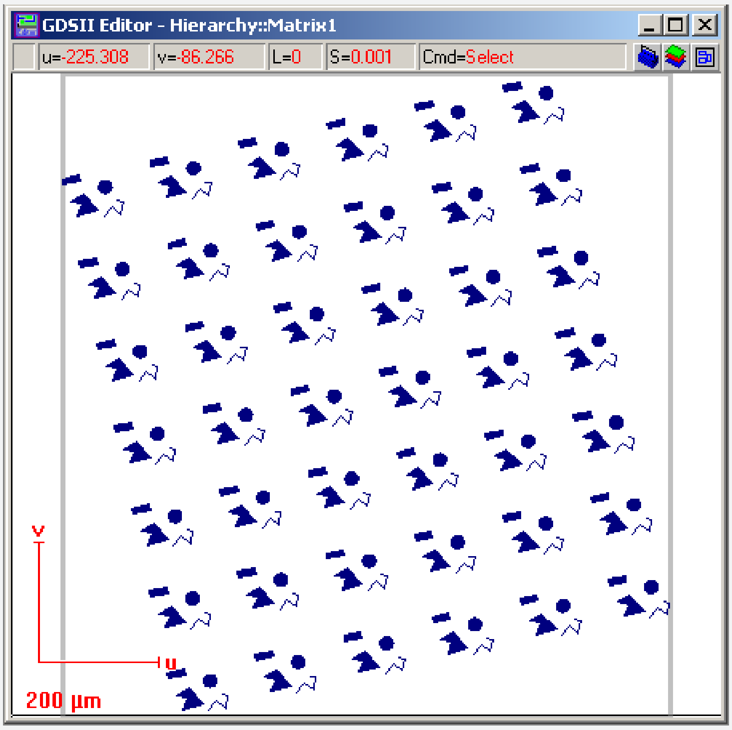
STEP 3 ▶

Click on the downward arrow next to the **Name** field and select a structure from the dropdown list. 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 by mouse click. Press **Escape** to place the structure only once.

STEP 4 ▶

After finishing placement of structure reference, 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.

Figure 6-3 Re-open the **Hierarchy** structure.



GDSII Editor - Hierarchy::Matrix1

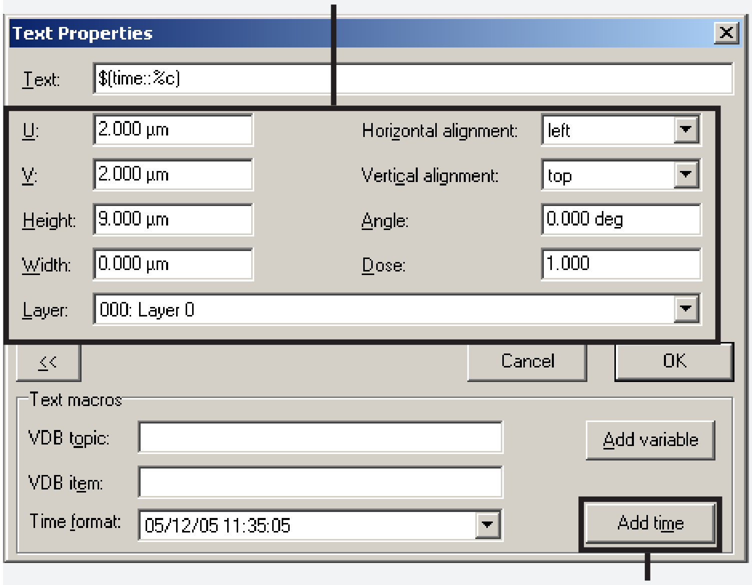
u=-225.308 v=-86.266 L=0 S=0.001 Cmd=Select

STEP 5 ▶

Create a new structure within the GDSII database with the name **Time.** In the toolbox, click on the **Text** icon, and click on that location within the Editor, where you want to place the text. You can now insert any text. By clicking **>>**, additional parameters are available. You can call current variables such as the time or any other variables from the VDB file used for the current patterning. Select **Time format** and press the **Add time** button, which will display a special command string in the text field.

Figure 6-4 Text Properties window.

The parameters for the Text properties can be entered.



Text: $(time::%c)

U: 2.000 µm

V: 2.000 µm

Height: 9.000 µm

Width: 0.000 µm

Horizontal alignment: left

Vertical alignment: top

Angle: 0.000 deg

Dose: 1.000

Layer: 000: Layer 0

VDB topic:

VDB item:

Time format: 05/12/05 11:35:05

Click on Add time to display the command string.

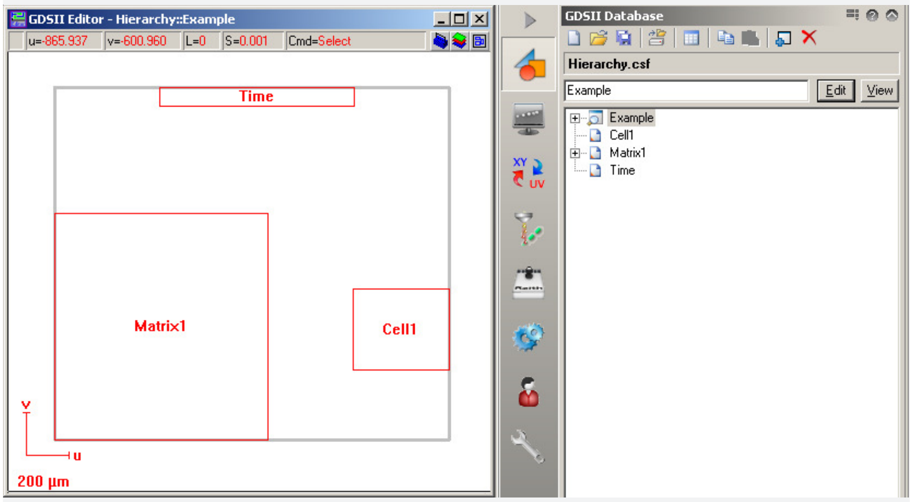
STEP 6 ▶

There is a wider variety of command strings available for other formats or variables, which are described in more detail in the Software Reference 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 new structure we will insert the structures designed earlier.

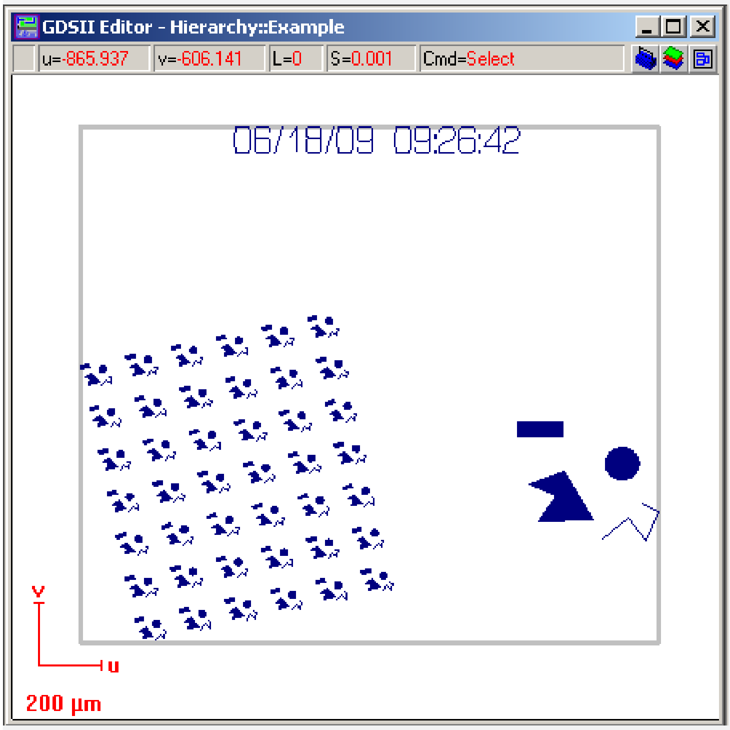
Figure 6-5 Inserting pre-designed structure into the **GDSII Editor**.



STEP 8 ▶

Click on the **Structure Reference** icon 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. Select hierarchy level 2 or higher to resolve the pattern containing the elements of structure cell 1.

Figure 6-6 Selecting the **Hierarchy** level.



Task 2 Studying chessy.csf

STEP 1 ▶

Open the file Chessy.csf.

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

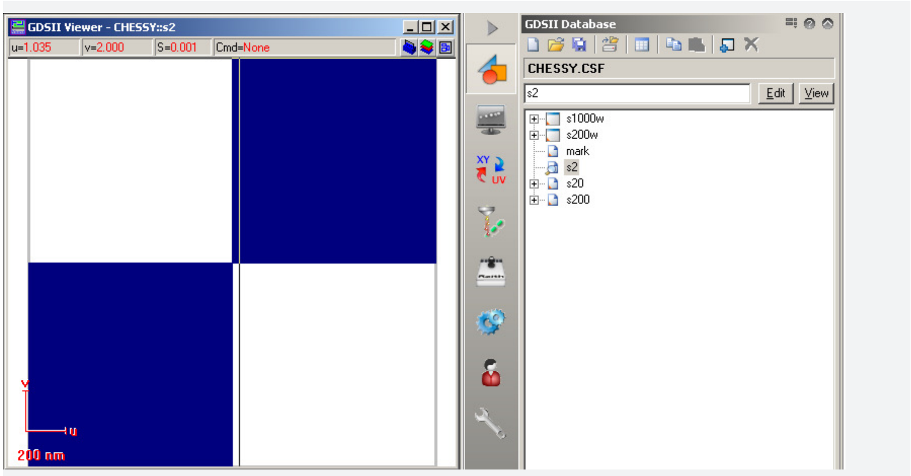


Figure 6-7 Studying chessy in the **GDSII Viewer**.

STEP 2 ▶

Now open the pattern **S20** using the **GDSII Viewer** and select the **Fill** icon. 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.

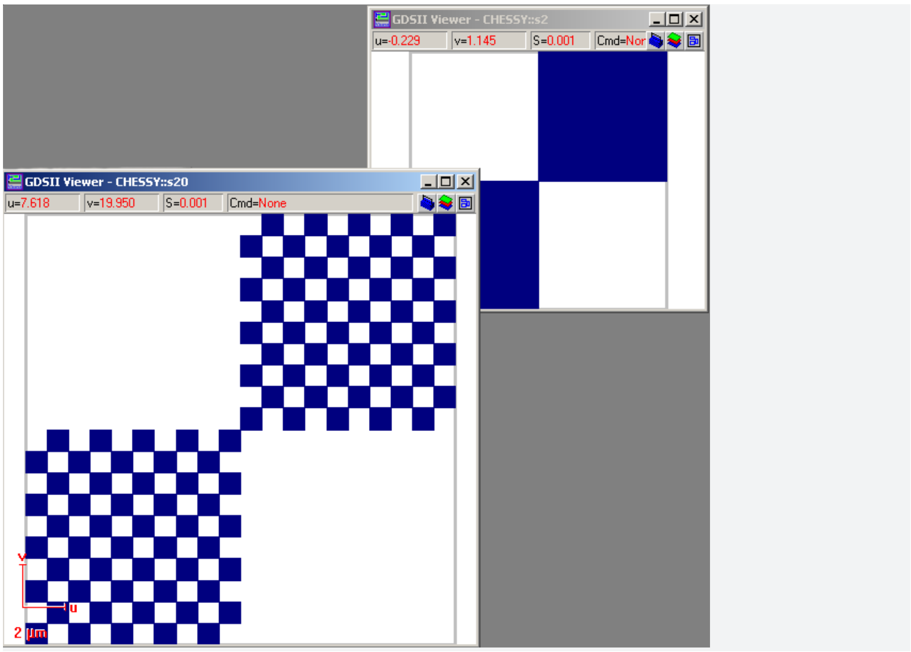
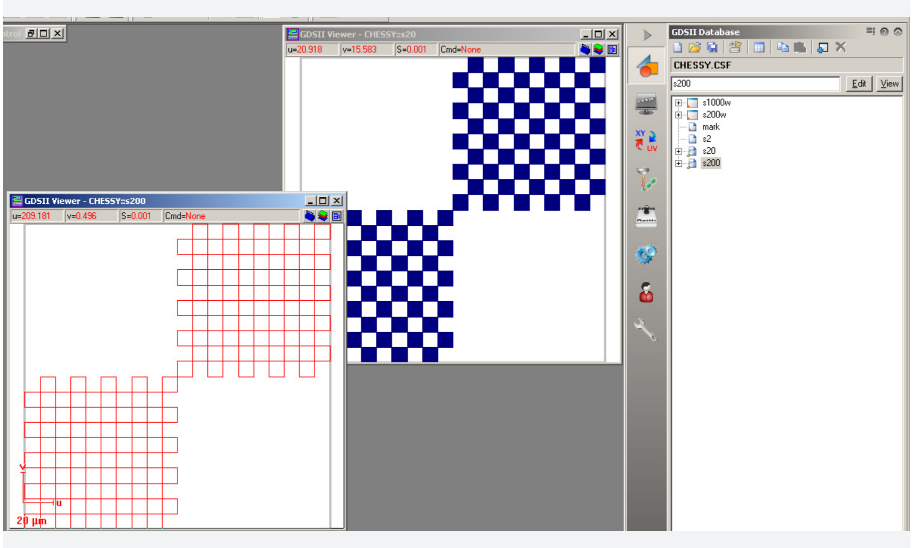


Figure 6-8 Selecting different **Hierarchy** levels for viewing.

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.

Figure 6-9 Studying a different pattern in the **GDSII Viewer**.



GDSII Viewer - CHESSY::s20

u=20.918 v=15.583 S=0.001 Cmd=None

GDSII Viewer - CHESSY::s200

u=209.181 v=0.496 S=0.001 Cmd=None

GDSII Database

CHESSY.CSF

s200

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.

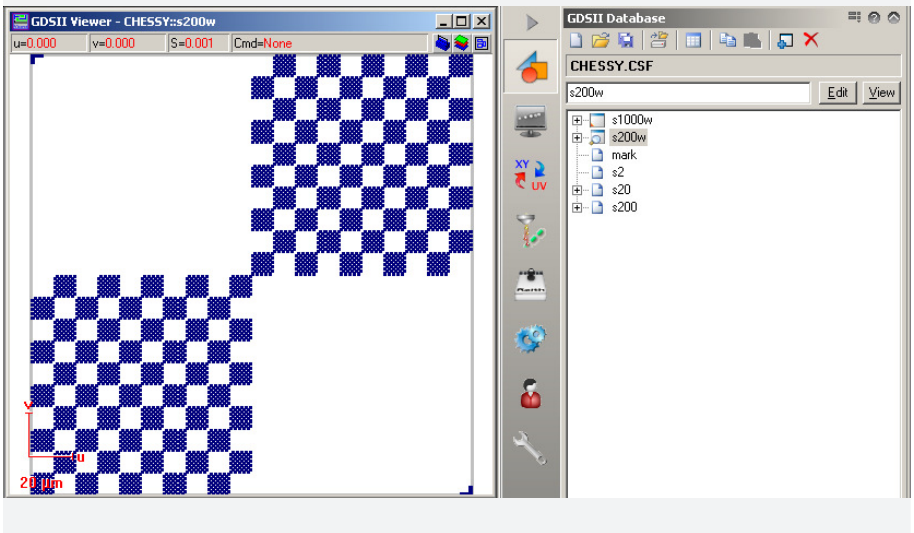


Figure 6-10 Studying the pre-defined pattern **Mark**.

HINT

The same process of hierarchy levels design can be continued from one hierarchy level to the next. For example, the pattern S1000w already includes 125,000 squares. Even though 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.

6.2 Advanced Pattern Design using FBMS

If you do not have the FBMS option installed, you can proceed straight to the next chapter.

Task 1 Designing FBMS elements

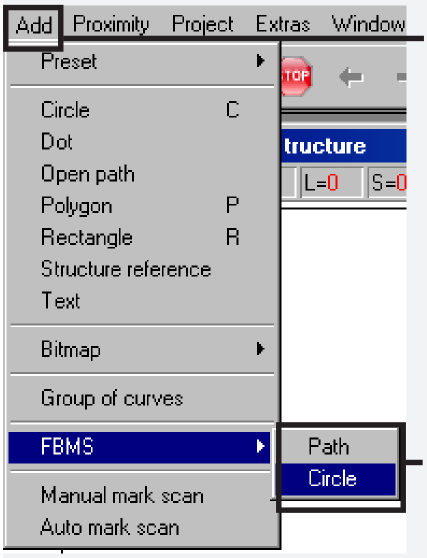
HINT

It is possible for the user to mix standard GDSII elements without FBMS, together with elements in which the FBMS technique is used.

STEP 1 ▶

To design **FBMS elements** in the GDSII structure, you can choose between a path or circle, e.g. **Add > FBMS > Circle**.

Figure 6-11 Designing FBMS elements.



Select the Add tab from the menu bar.

Either a Path or a Circle can be selected to create an FBMS element.

To insert a path or a circle, click on either **Path** or **Circle** in the **Add** menu, then click into the GDSII Editor at the position you wish to place the structure.

HINT

The pattern design is exactly the same as in the standard version.

HINT

FBMS is particularly useful for users who want to create large designs without the use of stitching.

The limitation of FBMS is that due to the movement of the stage, additional periodic non-linear components can be introduced to the structure.

HINT

FBMS can be interspersed with standard structures in the same structure. When the procedure is executed, the standard structures will be exposed first and then the FBMS structures.

In this way, the user has the freedom to choose the speed advantages of the FBMS patterning, as well as the higher accuracy of the standard structure.

7 Patterning

AIM

The aim of this chapter is to guide the user through the steps needed to carry out a patterning task.

7.1 Patterning (Standard)

Chapter 7.1 explains the patterning for a standard pattern.

Task 1 Familiarization with demo pattern
Task 2 Measuring the beam current
Task 3 Patterning
Task 4 Developing the sample
Task 5 Multiple patterning

7.2 Patterning for FBMS Elements (Option)

Chapter 7.2 is only applicable if the option for FBMS is installed on the Turnkey system.

Task 1 Patterning parameters for FBMS

7.1 Patterning

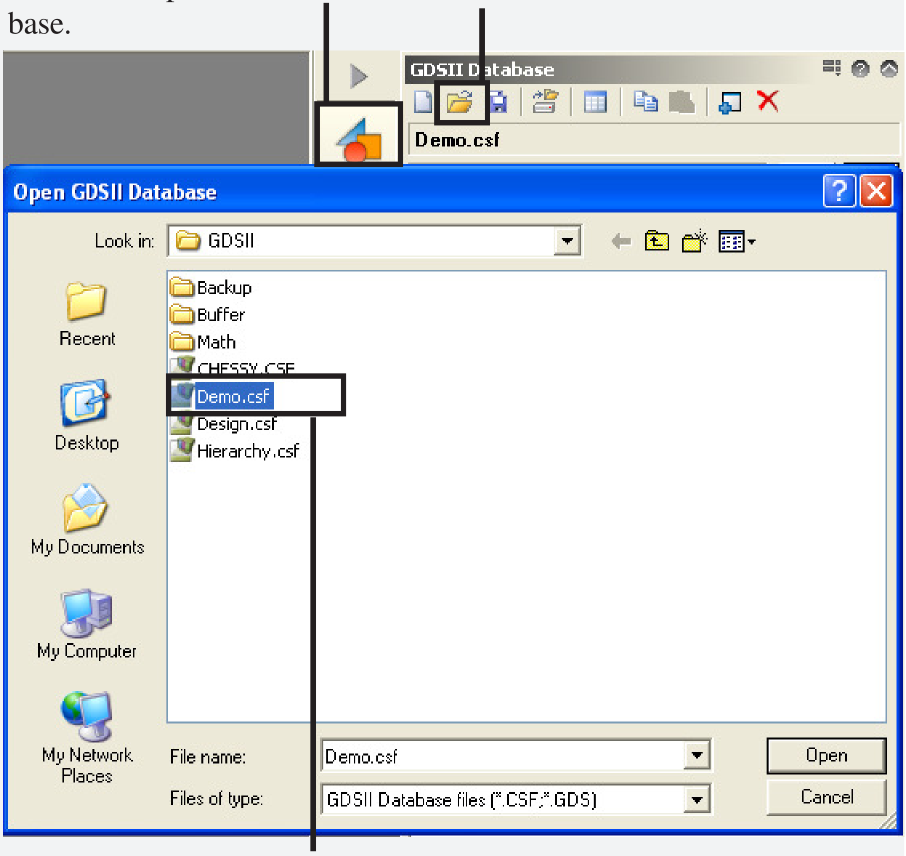
Task 1 Familiarization with demo pattern

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

STEP 1 ▶

Click on the **Design** icon in the **control** bar to open the **GDSII Database**. Then click on the **Open** icon to open another GDSII data file. A dialog box opens with a list of file names and folder options. Select **Demo.csf**.

Figure 7-1 Opening the Demo Pattern.



Select the **Design** icon from the **control** bar to open the GDSII Database.

Click on this icon to open a database file.

Select the file Demo.csf.

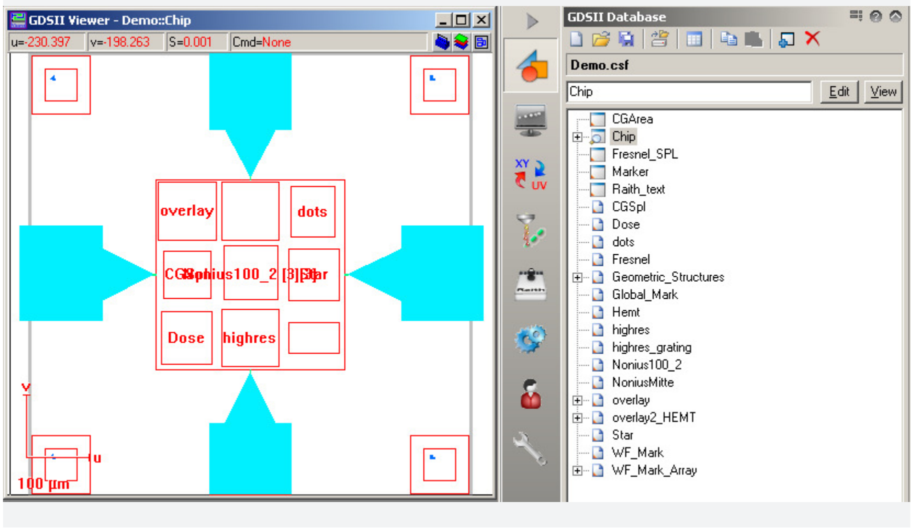
Look in: GDSII

File name: Demo.csf

Files of type: GDSII Database files (*.CSF;*.GDS)

STEP 2 ▶ Highlight the pattern **Chip**, then double click on it to open the Chip pattern in the GDSII Viewer. The GDSII Viewer will now display the hierarchical structure of the selected pattern.

Figure 7-2 Studying the pattern **Chip**.



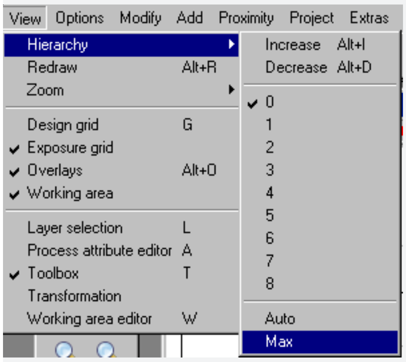
GDSII Viewer - Demo::Chip

u=-230.397 v=-198.263 S=0.001 Cmd=None

STEP 3 ▶

While the viewer is activated, choose **View > Hierarchy > Max** from the menu bar.

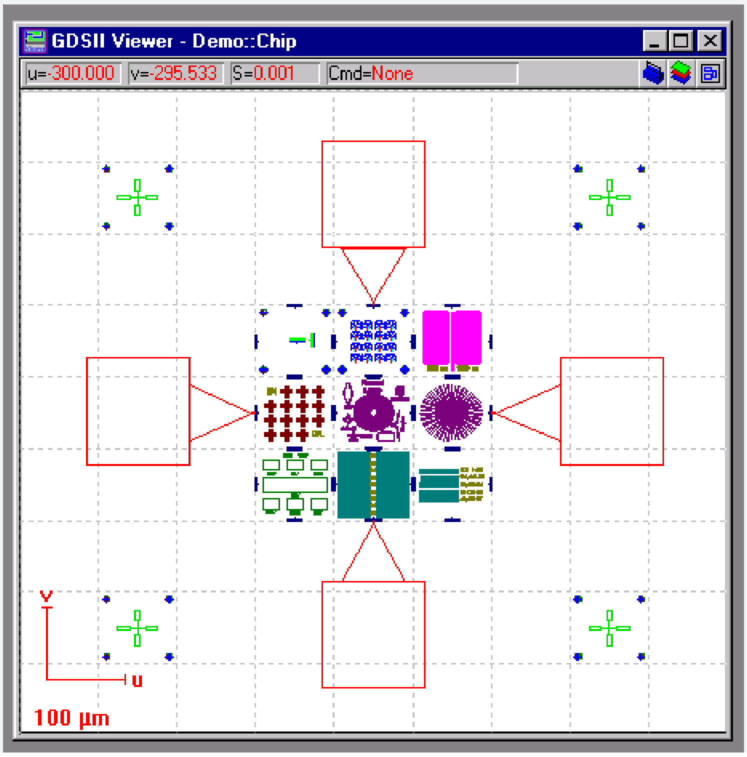
Figure 7-3 Select Hierarchy via the menu bar.



STEP 4 ▶

The full structure is now displayed, showing various test patterns, as described in detail within **Raith_Demo_Pattern.pdf**, which is located in each GDSII folder of every user.
System route **User > GDSII > Raith_Demo_Pattern.pdf**.

Figure 7-4 Displaying the full structure in the GDSII Viewer.


GDSII Viewer - Demo::Chip

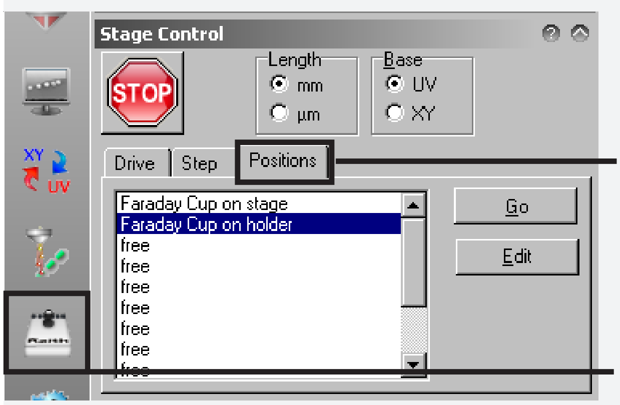
u=-300.000 v=-295.533 S=0.001 Cmd=None

Task 2 Measuring the beam current

STEP 1 ▶

Open the **Stage Control** window by clicking the corresponding icon on the **control** bar and drive to one of the Faraday cups. Its position may already be stored as one of the **Positions**.

Figure 7-5 Opening the Stage Control window.



Click the Positions tab to select one of the pre-defined positions.

Click the Stage icon in the control bar to open the Stage Control.

STEP 2 ▶

When the stage is at the **Faraday cup**, toggle the **beamblanker** to switch on the beam. In the Raith EO 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 ▶

Take note of the current.

Figure 7-6 Measuring the Current.

HINT

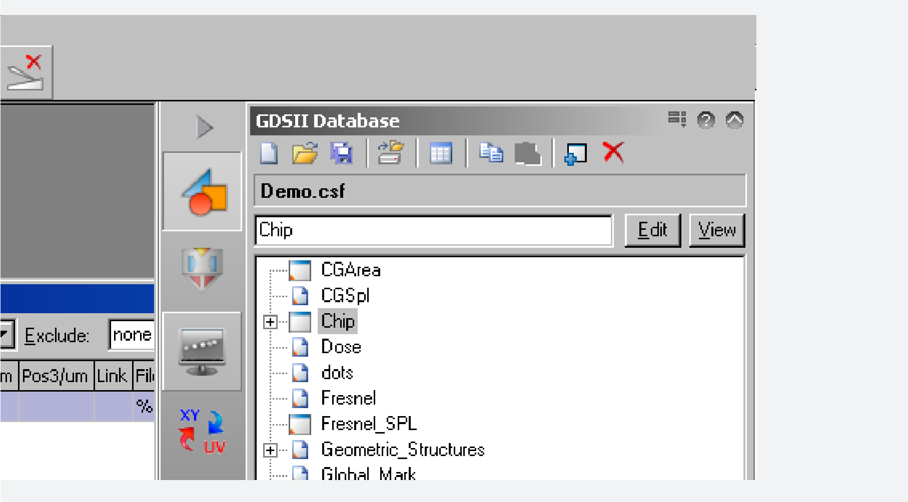
To avoid complication with defocussing, the beam current should also be measured using the same working distance as used for the patterning.

Task 3 Patterning

STEP 1 ▶ Make sure that the Writefield size is set to 100 µm in the **Writefield Manager** window.

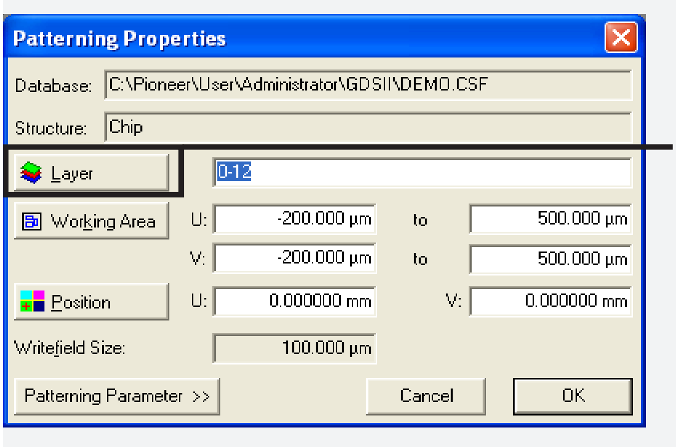
STEP 2 ▶ Open a New Positionlist via the **menu** bar, **File > New Positionlist**. Drag and drop the design **Chip** into the positionlist.

Figure 7-7 Open a New Positionlist.



STEP 3 ▶ By default, the Patterning is scheduled for the current sample position. The next step is to change the Patterning 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 Patterning 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 in the positionlist and a cascading menu will be displayed. Click on **Properties**. Enter the position to U=V=2 mm.

Figure 7-8 Patterning Properties window.



Database: C:\Pioneer\User\Administrator\GDSII\DEMO.CSF

Structure: Chip

Layer: 0-12

Working Area U: -200.000 µm to 500.000 µm

V: -200.000 µm to 500.000 µm

Position U: 0.000000 mm V: 0.000000 mm

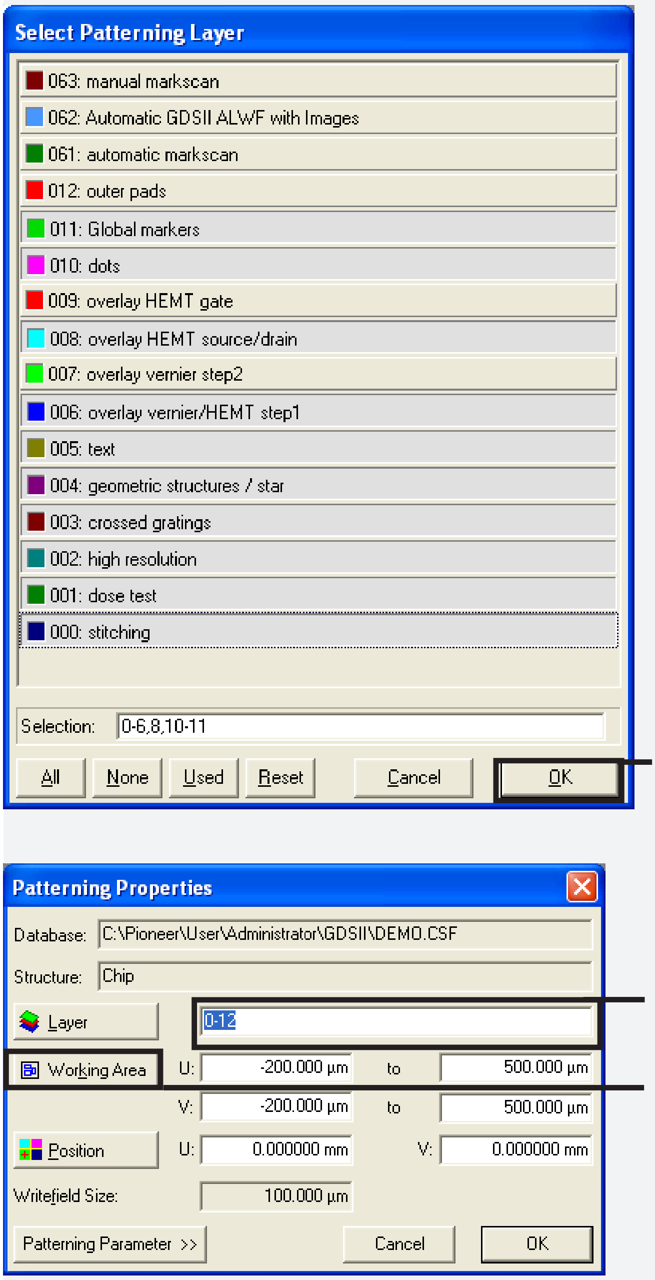
Writefield Size: 100.000 µm

Click on the Layer icon to select the patterned layer.

STEP 4 ▶

In the **Patterning Properties** dialog box, click on the **Layer** button and select layers 0-6 as well as layers 8, 10 and 11. Confirm with **OK**.

Figure 7-9 Select Patterning Layer.



Select Patterning Layer

- 063: manual markscan
- 062: Automatic GDSII ALWF with Images
- 061: automatic markscan
- 012: outer pads
- 011: Global markers
- 010: dots
- 009: overlay HEMT gate
- 008: overlay HEMT source/drain
- 007: overlay vernier step2
- 006: overlay vernier/HEMT step1
- 005: text
- 004: geometric structures / star
- 003: crossed gratings
- 002: high resolution
- 001: dose test
- 000: stitching

Selection: 0-6,8,10-11

Confirm with OK.

Patterning Properties

Database: C:\Pioneer\User\Administrator\GDSII\DEMO.CSF

Structure: Chip

Layer: 0-12

Working Area U: -200.000 µm to 500.000 µm

V: -200.000 µm to 500.000 µm

Position U: 0.000000 mm V: 0.000000 mm

Writefield Size: 100.000 µm

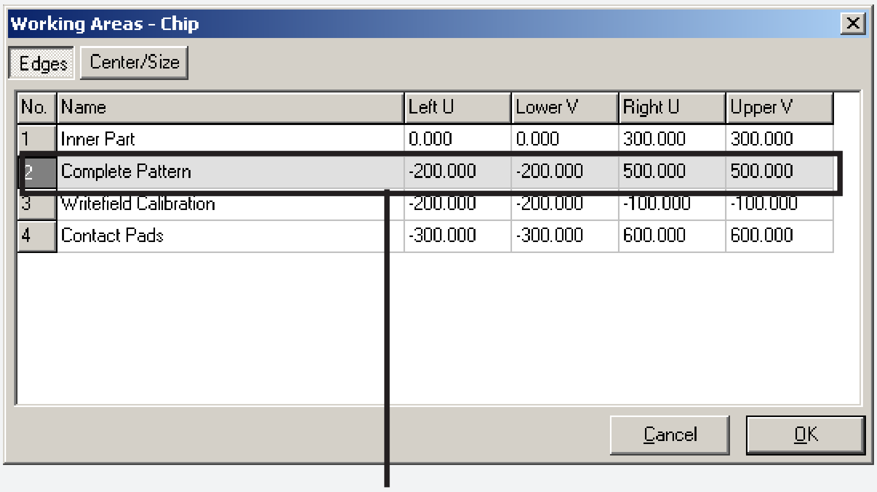
Selected Layers are now displayed.

Click on the Select Working Area icon. Patterning Properties can be edited

STEP 5 ▶

In the same dialog box click on the **Select Working Area** icon this will open a new dialog box. Select the working area named **Complete Pattern**. Confirm both windows with **OK**.

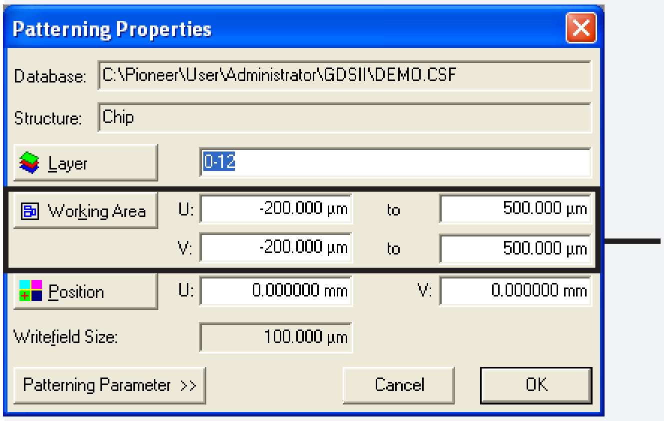
Figure 7-10 Working Area parameters.



No.	Name	Left U	Lower V	Right U	Upper V
1	Inner Part	0.000	0.000	300.000	300.000
2	Complete Pattern	-200.000	-200.000	500.000	500.000
3	Writefield Calibration	-200.000	-200.000	-100.000	-100.000
4	Contact Pads	-300.000	-300.000	600.000	600.000

Select the Complete Pattern row.

Figure 7-11 Working Area defined within the Patterning Properties Dialog.



Database: C:\Pioneer\User\Administrator\GDSII\DEMO.CSF

Structure: Chip

Layer: 0-12

Working Area U: -200.000 µm to 500.000 µm

V: -200.000 µm to 500.000 µm

Position U: 0.000000 mm V: 0.000000 mm

Writefield Size: 100.000 µm

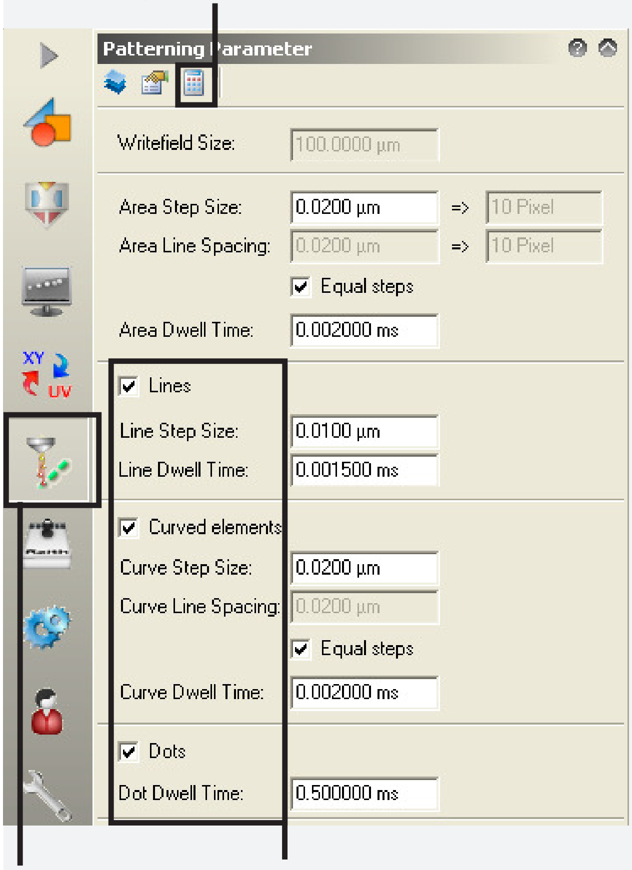
Selected Working Area is now displayed.

STEP 6 ▶

Goto the **Patterning Parameter** window by clicking on the corresponding icon in the **control** bar. Check the checkbox for **SPL Exposure**, **Curved elements** and **Dot Exposure**. Click the **calculator** icon.

Figure 7-12 Opening the **Patterning Parameter** Window.

Click on the calculator icon to open the Patterning Parameter Calculation.



Patterning Parameter

Writefield Size: 100.0000 µm

Area Step Size: 0.0200 µm => 10 Pixel

Area Line Spacing: 0.0200 µm => 10 Pixel

☑ Equal steps

Area Dwell Time: 0.002000 ms

☑ Lines

Line Step Size: 0.0100 µm

Line Dwell Time: 0.001500 ms

☑ Curved elements

Curve Step Size: 0.0200 µm

Curve Line Spacing: 0.0200 µm

☑ Equal steps

Curve Dwell Time: 0.002000 ms

☑ Dots

Dot Dwell Time: 0.500000 ms

Select the Paterning icon.

Check the checkbox for **Lines**, **Curved elements** and **Dots**.

HINT

The **Beam Current** in the **Patterning Parameter Calculation** window shows the same value as measured before. There are different tabs assigned for **Areas,Curved elements, 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 7 ▶

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 **Curved Elements** tab and enter the same dose value. Click on the **Line** tab and enter the corresponding **Line Dose** of 300 pAs/cm. Then click on the **Dot** tab and enter 0.01 pAs for the **Dose**.

HINT

After you have entered the appropriate dose, the corresponding tab title (Area, Curved Elements, 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 gray, since the parameters are no longer consistent.

STEP 8 ▶

Switch back to the Area tab and enter the step size and line spacing 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 9 ▶

Select the **Curved Elements** tab and enter the step size and line spacing of 0.020 µm. Click the **Calculator** button next to the Dwell time.

STEP 10 ▶

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.

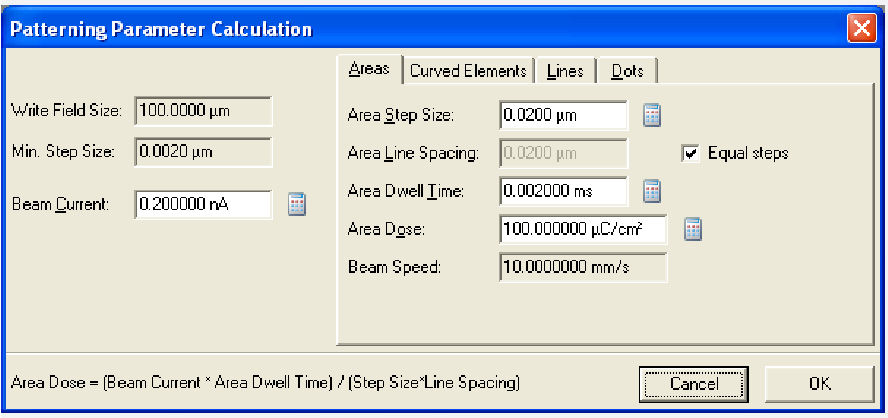
STEP 11 ▶

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

STEP 12 ▶

Now all four tab titles, **Area**, **Curved Elements**, **Line** and **Dot** should be in black and the **OK** button is now enabled. Click on OK.

Figure 7-13 Opening the **Patterning Parameter Calculation** window.



Write Field Size: 100.0000 µm
Min. Step Size: 0.0020 µm
Beam Current: 0.200000 nA
Area Step Size: 0.0200 µm
Area Line Spacing: 0.0200 µm
☑ Equal steps
Area Dwell Time: 0.002000 ms
Area Dose: 100.000000 µC/cm²
Beam Speed: 10.0000000 mm/s

Area Dose = (Beam Current * Area Dwell Time) / (Step Size*Line Spacing)

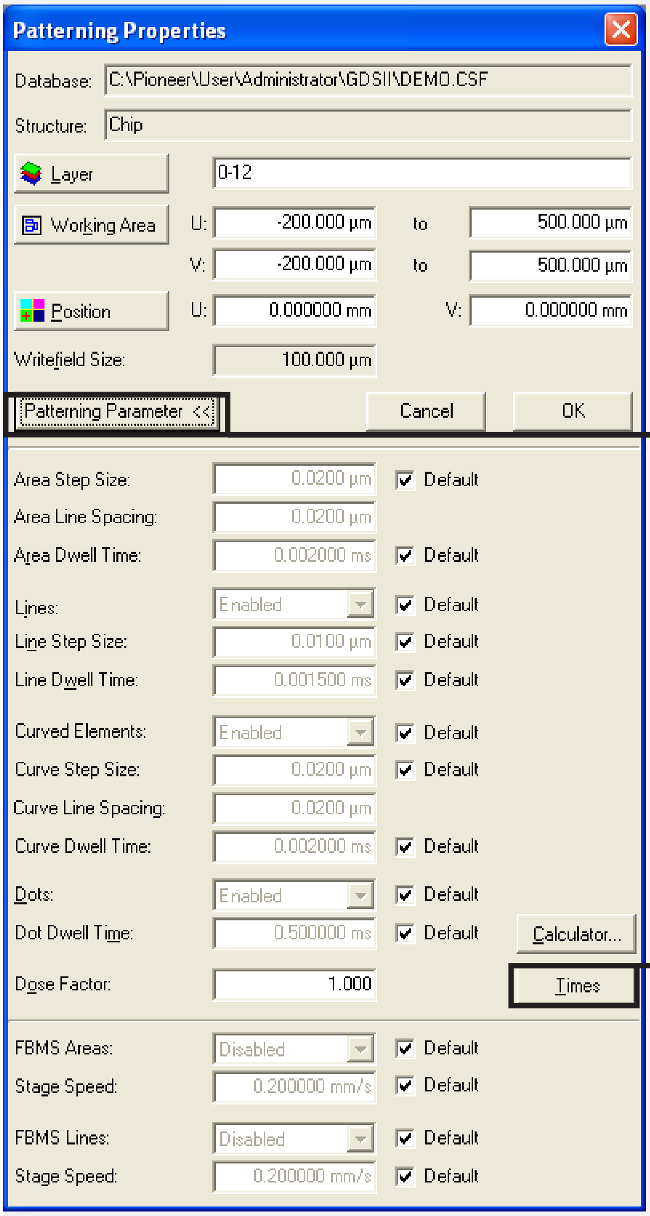
HINT

It is possible to individually evaluate Dot, Line, Curved Elements in conjunction with the Area.

STEP 13 ▶

Go to the **Positionlist** window. Highlight the corresponding line with the right mouse button, select **Properties**. The dialog box, **Patterning Properties** will open. Click on the **Patterning Parameter** button to display the exposure values. Click on the **Times** button to obtain the **Estimated Patterning Time**.

Figure 7-14 Patterning Properties displays the Estimated Patterning Times.



Database: C:\Pioneer\User\Administrator\GDSII\DEMO.CSF

Structure: Chip

Layer: 0-12

Working Area U: -200.000 µm to 500.000 µm

V: -200.000 µm to 500.000 µm

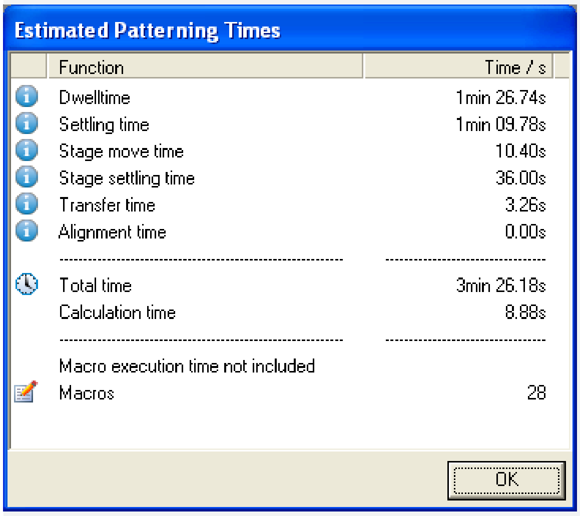
Position U: 0.000000 mm V: 0.000000 mm

Writefield Size: 100.000 µm

Area Step Size:	0.0200 µm	☑ Default
Area Line Spacing:	0.0200 µm	
Area Dwell Time:	0.002000 ms	☑ Default
Lines:	Enabled	☑ Default
Line Step Size:	0.0100 µm	☑ Default
Line Dwell Time:	0.001500 ms	☑ Default
Curved Elements:	Enabled	☑ Default
Curve Step Size:	0.0200 µm	☑ Default
Curve Line Spacing:	0.0200 µm	
Curve Dwell Time:	0.002000 ms	☑ Default
Dots:	Enabled	☑ Default
Dot Dwell Time:	0.500000 ms	☑ Default
Dose Factor:	1.000	
FBMS Areas:	Disabled	☑ Default
Stage Speed:	0.200000 mm/s	☑ Default
FBMS Lines:	Disabled	☑ Default
Stage Speed:	0.200000 mm/s	☑ Default

Click on the Patterning Parameter button to display additional parameters.

Click the Times button to calculate the Estimated Patterning Times.



Estimated Patterning Times

Function	Time / s
Dwelltime	1min 26.74s
Settling time	1min 09.78s
Stage move time	10.40s
Stage settling time	36.00s
Transfer time	3.26s
Alignment time	0.00s
Total time	3min 26.18s
Calculation time	8.88s
Macro execution time not included	
Macros	28

The estimated patterning times are now displayed.

STEP 14 ▶ Activate the positionlist. Go to the **menu** bar and select **Scan > Selection**. The stage will now drive to the position to execute the patterning task.

STEP 15 ▶ If you wish to calculate the **Patterning time** for the complete Positionlist, go to menu bar **Filter>Calculate Patterning time.**

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:IPA=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.

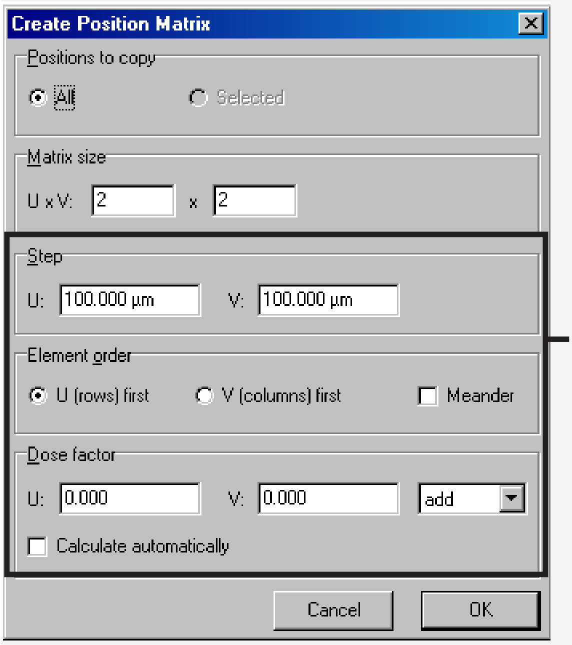
STEP 3 ▶ After you have completed the first inspection using the optical microscope, you can insert the sample into the RAITH system. Perform the stage alignment and address the corresponding sample positions. In our example U = V = 2 mm, for imaging the pattern.

Task 5 Multiple Patterning

STEP 1 ▶

We will expose a structure which has no dose variation. Highlight the line in the positionlist, select **Filter > Matrix Copy** and enter values for **Matrix size**, **Step** size and **Dose** scaling.

Figure 7-15 Create Position Matrix.



Create Position Matrix

Positions to copy

● All ○ Selected

Matrix size

U x V: 2 x 2

Step

U: 100.000 µm V: 100.000 µm

Element order

● U (rows) first ○ V (columns) first ☐ Meander

Dose factor

U: 0.000 V: 0.000 add

☐ Calculate automatically

Enter the values for the Matrix size, Step and Dose factor.

STEP 2 ▶

The structure will be exposed 4 times, each 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 > Patterning Parameters**.

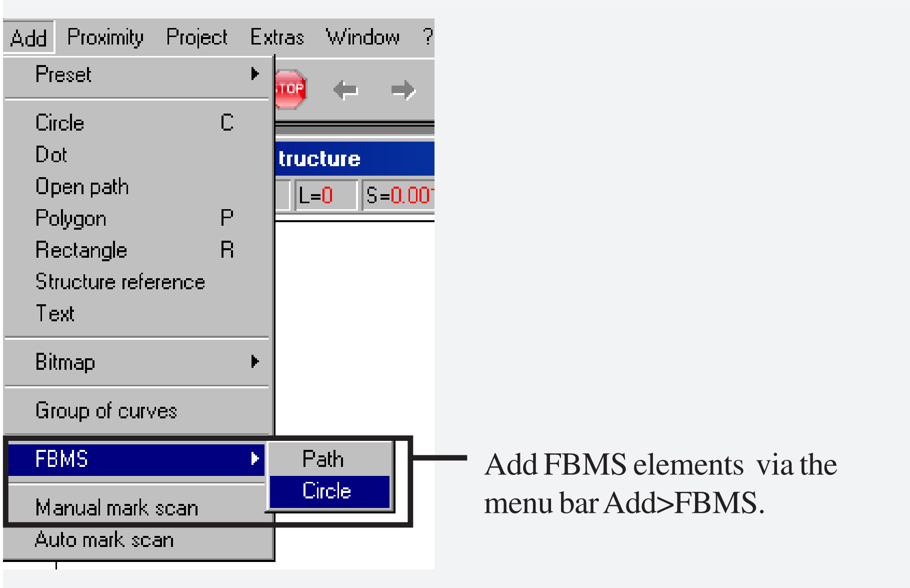
7.2 Patterning for FBMS Elements

Task 1 Patterning parameters for FBMS

STEP 1 ▶

You can add **FBMS** elements via the **menu** bar, **Add > FBMS > Path (or circle)**.

Figure 7-16 Add FBMS elements.



Add FBMS elements via the menu bar Add>FBMS.

STEP 2 ▶

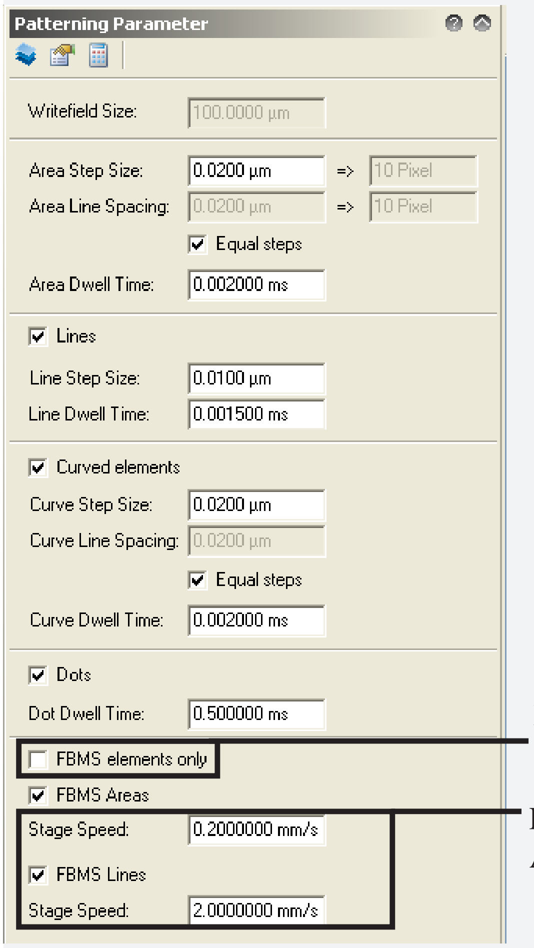
In **Patterning Parameters**, you may choose an **FBMS Area** or an **FBMS Line**, for either of which you may choose the stage speed.

If you wish to expose using FBMS, as well as the standard structures, do not check **FBMS elements only**. The only time that you should check this option is when you wish to expose FBMS elements with no standard structures.

STEP 3 ▶

Go to **Patterning Parameter Calculation**. Now there are two more tabs available. One is for **FBMS Area** and one for **FBMS Line**, in which the **Stage Speed** or the **Dose** can be calculated.

Figure 7-17 Patterning Parameter for FBMS elements.



Writefield Size: 100.0000 µm

Area Step Size: 0.0200 µm => 10 Pixel

Area Line Spacing: 0.0200 µm => 10 Pixel

☑ Equal steps

Area Dwell Time: 0.002000 ms

☑ Lines

Line Step Size: 0.0100 µm

Line Dwell Time: 0.001500 ms

☑ Curved elements

Curve Step Size: 0.0200 µm

Curve Line Spacing: 0.0200 µm

☑ Equal steps

Curve Dwell Time: 0.002000 ms

☑ Dots

Dot Dwell Time: 0.500000 ms

☐ FBMS elements only

☑ FBMS Areas

Stage Speed: 0.2000000 mm/s

☑ FBMS Lines

Stage Speed: 2.0000000 mm/s

Check FBMS elements only when you do not want to do a patterning of standard structures as well.

Enter the values for the FBMS Area and Line.

STEP 4 ▶

Within the **Patterning Parameters Calculation** window, in the **FBMS Area**, you will find the **Calculation Width**, which represents the typical width of a structure to be used for the design. This typical width can be set by the user, in **Patterning Details** within FBMS.

8 Mix and Match Patterning

AIM

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

This is a more advanced task. It is assumed that the user has carried out all previous tasks, to become familiar with the system.

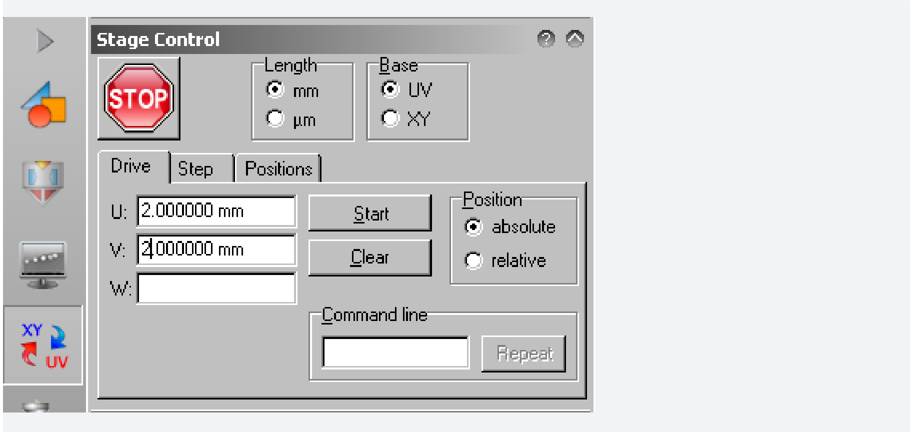
Task 1 Locating the first mark

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

STEP 1 ▶

In order to find the first mark, open the **Stage Control** window by clicking the corresponding Stage Control icon in the control bar. Enter the value 2 for U and V. Click on **Start**.

Figure 8-1 Opening the Stage Control window.



Stage Control

Length: ● mm ○ µm

Base: ● UV ○ XY

U: 2.000000 mm

V: 2.000000 mm

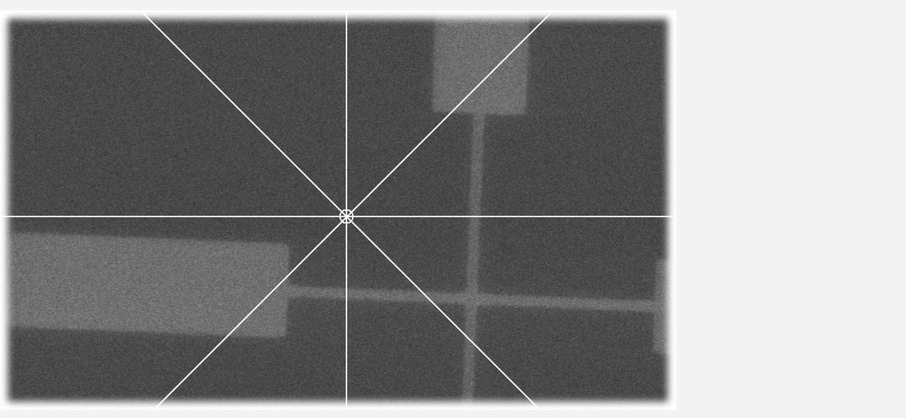
W:

Position: ● absolute ○ relative

STEP 2 ▶

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

Figure 8-2 Moving the crosshairs over the mark via the joystick.



Using the joystick, move the mark over the crosshairs and switch off the beam. The next task is to define a local coordinate system based on the design coordinates of the marks.

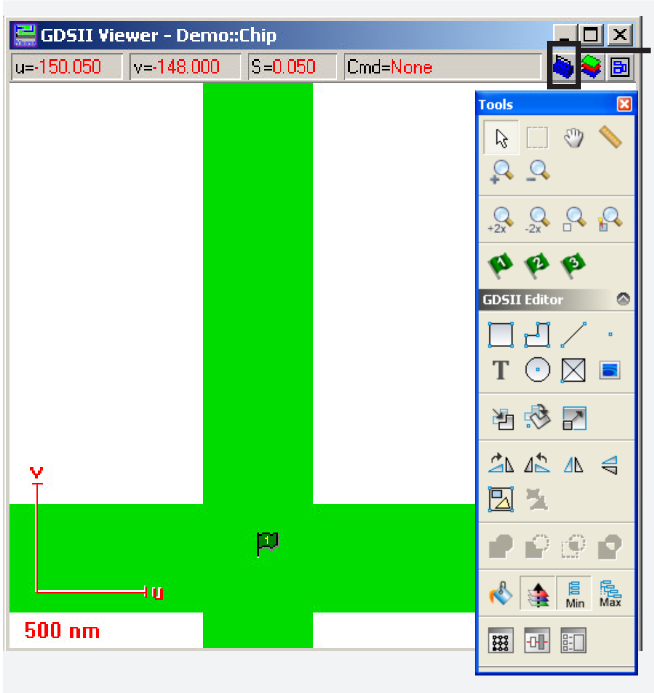
Task 2 Defining local UV positions of marks

STEP 1 ▶

In the **Adjust UVW** window, switch to **Local** coordinates.

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 tool box, by clicking on the **Toolbox** icon in the GDSII viewer. Drag and drop the green flag 1 onto your mark 1.

Figure 8-3 Opening the **GDSII Viewer** and the GDSII tool box.



Click on this Toolbox icon to open the GDSII tool box.

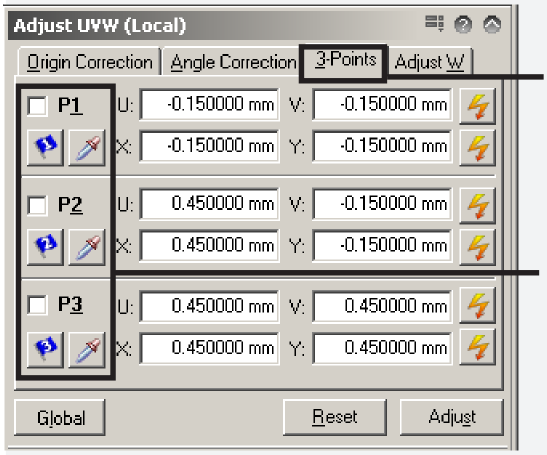
The UV coordinates for mark 1 will now be displayed in the Adjust UVW window.

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.

Figure 8-4 3-Points tab in Adjust UVW window.



Adjust UVW (Local)

☐ P1 U: -0.150000 mm V: -0.150000 mm
X: -0.150000 mm Y: -0.150000 mm

☐ P2 U: 0.450000 mm V: -0.150000 mm
X: 0.450000 mm Y: -0.150000 mm

☐ P3 U: 0.450000 mm V: 0.450000 mm
X: 0.450000 mm Y: 0.450000 mm

Go to 3-Point tab.

Ensure that all the mark positions are unchecked.

Task 3 3-points adjustment

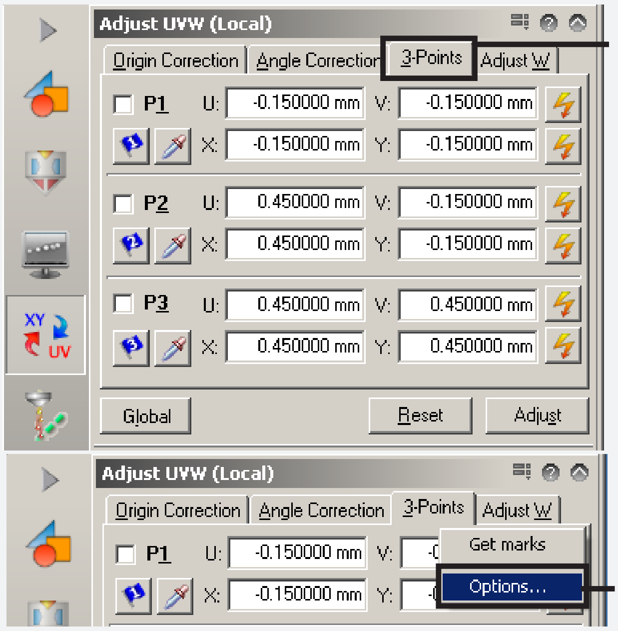
STEP 1 ▶

Open the **Adjust UVW window** and select the tab **3-Points**. Switch to **Local** coordinates.

HINT

If your sample is not leveled, it is also possible to read in the focus value together with the coordinates for all three marks. In this case, you would move the stage to all the three marks and re-adjust the focus on each mark before reading in the coordinates. In the **Adjust UV** window, the message **Focus!** will be displayed at the bottom of the window. The focus will now be changed for each digitally addressed UV location.

Figure 8-5 Select Options within the Adjust UVW window.



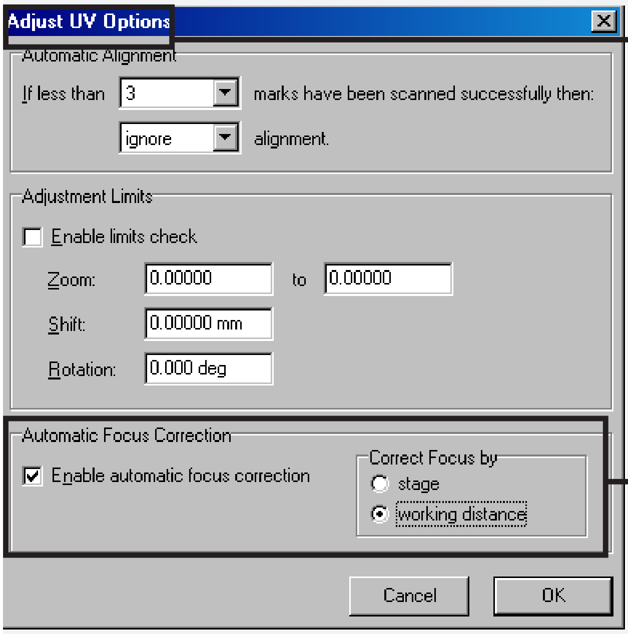
Adjust UVW (Local)

☐ P1	U:	-0.150000 mm	V:	-0.150000 mm
	X:	-0.150000 mm	Y:	-0.150000 mm
☐ P2	U:	0.450000 mm	V:	-0.150000 mm
	X:	0.450000 mm	Y:	-0.150000 mm
☐ P3	U:	0.450000 mm	V:	0.450000 mm
	X:	0.450000 mm	Y:	0.450000 mm

Select the 3-Points tab.

Select Options via right mouse click.

Figure 8-6 Adjust UV Options with Automatic Focus Correction .


Adjust UV Options

Automatic Alignment

If less than 3 marks have been scanned successfully then: ignore alignment.

Adjustment Limits

☐ Enable limits check

Zoom: 0.00000 to 0.00000

Shift: 0.00000 mm

Rotation: 0.000 deg

Automatic Focus Correction

☑ Enable automatic focus correction

Correct Focus by
○ stage
● working distance

The Adjust UV Options will now be displayed.

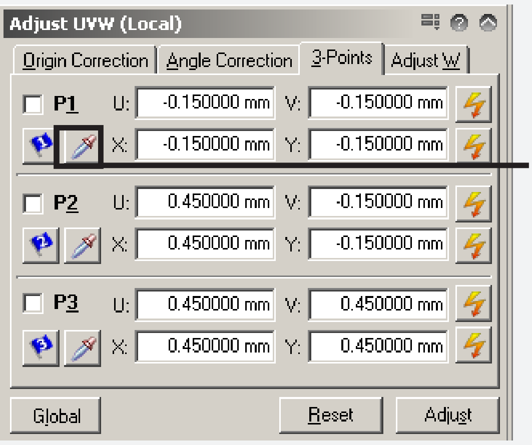
The focus can be automatically adjusted either via the stage or working distance.

In **Adjust UV Options**, we will now check the box to **Enable automated focus correction**. The focus can be corrected in two ways. Here we will select the working distance. The lens will now move to adjust for the new focus settings. It is also possible to select **Stage** to adjust the automatic focus. In this case, the stage will be moved to re-adjust the focus.

STEP 2 ▶

Check the focus level. Click on the **Pipette** icon to update the current XY coordinates of the first mark position. Activate checkbox P1 and click on **Adjust**.

Figure 8-7 Reading in the current coordinates via the Pipette icon.


Adjust UVW (Local)

☐ P1 U: -0.150000 mm V: -0.150000 mm
X: -0.150000 mm Y: -0.150000 mm

☐ P2 U: 0.450000 mm V: -0.150000 mm
X: 0.450000 mm Y: -0.150000 mm

☐ P3 U: 0.450000 mm V: 0.450000 mm
X: 0.450000 mm Y: 0.450000 mm

Click on the Pipette icon to read in the current coordinates.

In this step, we have performed, in principle, an origin correction. This means that the origin of the local coordinate system has been redefined and is now identical to the origin of the design coordinate system (GDSII).

STEP 3 ▶

Click on the **Flash** icon related to the UV coordinates of P2. 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 crosshairs is situated over the mark. Check the focus level.

Click on the **Pipette** icon of P2. The XY coordinates in the Adjust UVW window will be updated.

Click the checkbox of P2 and click **Adjust**. Please note that the UV coordinates have been updated after the adjustment has been performed.

STEP 5 ▶

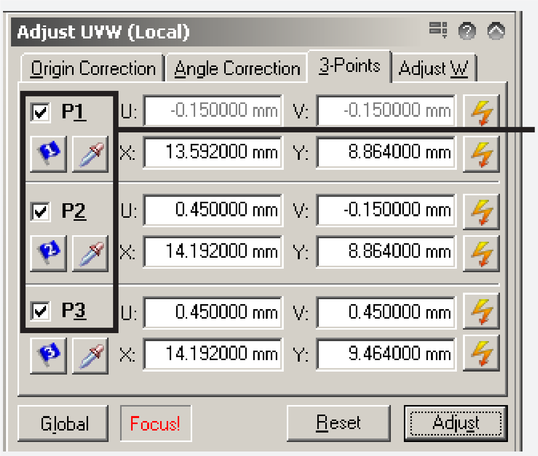
Click on the **Flash** icon related to the UV coordinates of P3 to move the stage to mark 3.

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 crosshairs is situated above the mark. Check the focus level. Click on the **Pipette** icon of P3. The XY coordinates will be updated.

Check P3 and click **Adjust**.

Figure 8-8 All three Marks have now been checked.



☑ P1	U:	-0.150000 mm	V:	-0.150000 mm
	X:	13.592000 mm	Y:	8.864000 mm
☑ P2	U:	0.450000 mm	V:	-0.150000 mm
	X:	14.192000 mm	Y:	8.864000 mm
☑ P3	U:	0.450000 mm	V:	0.450000 mm
	X:	14.192000 mm	Y:	9.464000 mm

All three marks have now been checked for focus level and have been updated.

The local coordinates system is now identical to the GDSII design coordinate system.

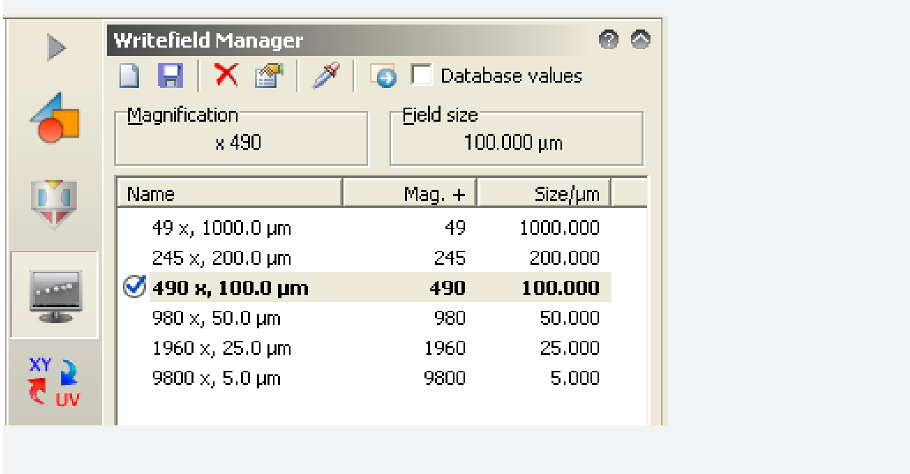
Task 4 Semi-automated Writefield alignment

In certain cases, it may not be required to set-up a fully automated Writefield 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 clicking the corresponding icon:

STEP 2 ▶ Open the **Writefield Manager** window, select 100 µm Writefield from the list and click on .

Figure 8-9 Open the **Writefield Manager** dialog.



☐ Database values

Magnification: x 490

Field size: 100.000 µm

Name	Mag. +	Size/µm
49 x, 1000.0 µm	49	1000.000
245 x, 200.0 µm	245	200.000
✓ **490 x, 100.0 µm**	**490**	**100.000**
980 x, 50.0 µm	980	50.000
1960 x, 25.0 µm	1960	25.000
9800 x, 5.0 µm	9800	5.000

STEP 3 ▶ Open a new **positionlist**.

STEP 4 ▶

Select the structure **Chip** from the database **Demo.csf** and drag and drop it into the positionlist.

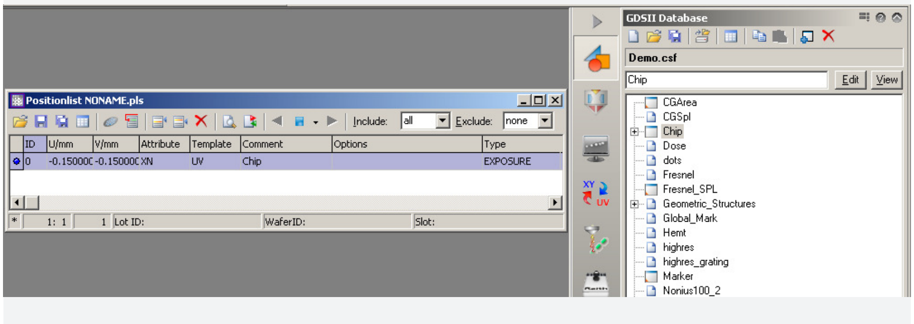


Figure 8-10 Open a new **Positionlist**.

Positionlist NONAME.pls

Include: all Exclude: none

ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type
0	-0.15000(	-0.15000(	XN	UV	Chip		EXPOSURE

* 1: 1 1 Lot ID: WaferID: Slot:

GDSII Database

Demo.csf

Chip

STEP 5 ▶

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

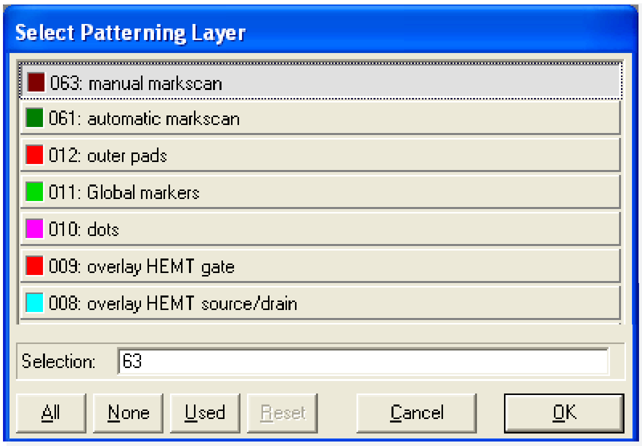


Figure 8-11 **Select Patterning Layer**.

Select Patterning Layer

- 🟥 063: manual markscan
- 🟩 061: automatic markscan
- 🟥 012: outer pads
- 🟩 011: Global markers
- 🟪 010: dots
- 🟥 009: overlay HEMT gate
- 🟦 008: overlay HEMT source/drain

Selection: 63

STEP 6 ▶

Click on the **Working area** icon and select the Working area **Writefield Calibration** and confirm with **OK**. Adjust the UV position by clicking the corresponding icon.

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

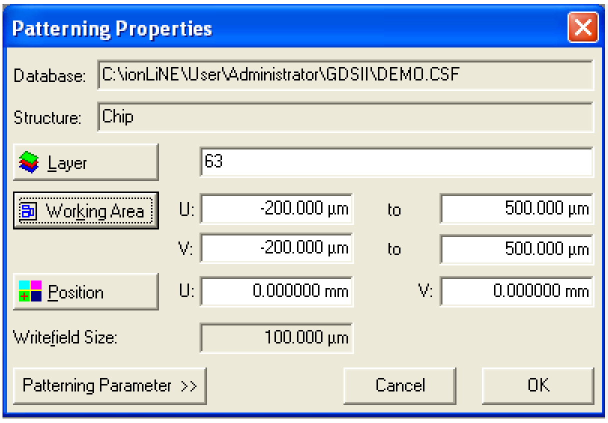


Figure 8-12 Viewing the **Patterning Properties**.

Patterning Properties

Database: C:\ionLiNE\User\Administrator\GDSII\DEMO.CSF

Structure: Chip

Layer: 63

Working Area U: -200.000 µm to 500.000 µm

V: -200.000 µm to 500.000 µm

Position U: 0.000000 mm V: 0.000000 mm

Writefield Size: 100.000 µm

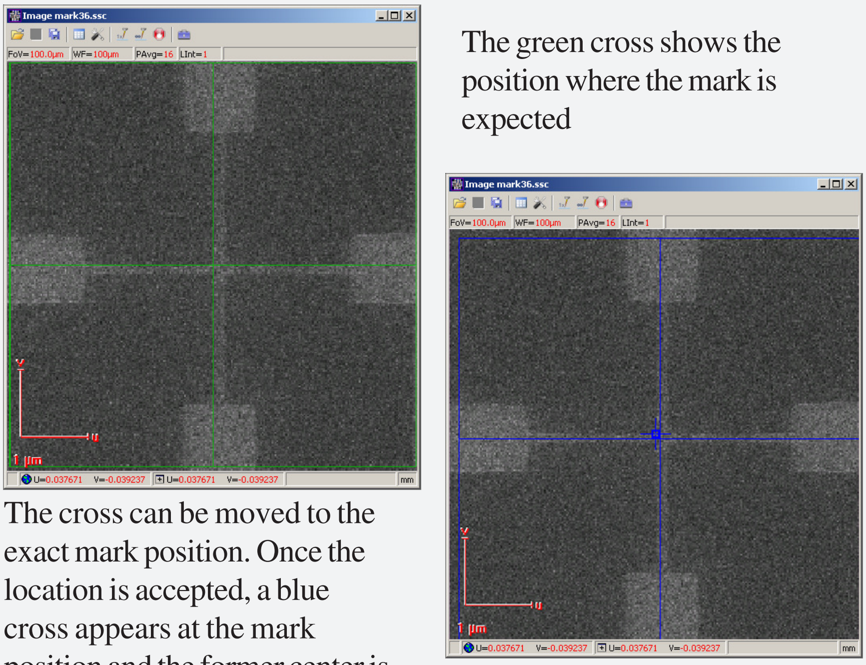
STEP 7 ▶

Activate the **Positionlist**. Select **Scan > All** from the menu bar. The stage will now drive to the corresponding position and the manual mark scan during patterning 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 pressed and the left mouse button pressed while moving the mouse cursor to the real mark position. Once you have reached the new position, release the Ctrl key and a blue cross will be displayed at the selected position.

Figure 8-13 Green cross positioning in the images.



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 Writefield alignment

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

STEP 2 ▶ Open the **Writefield Manager** window, select 100 µm Writefield from the list.

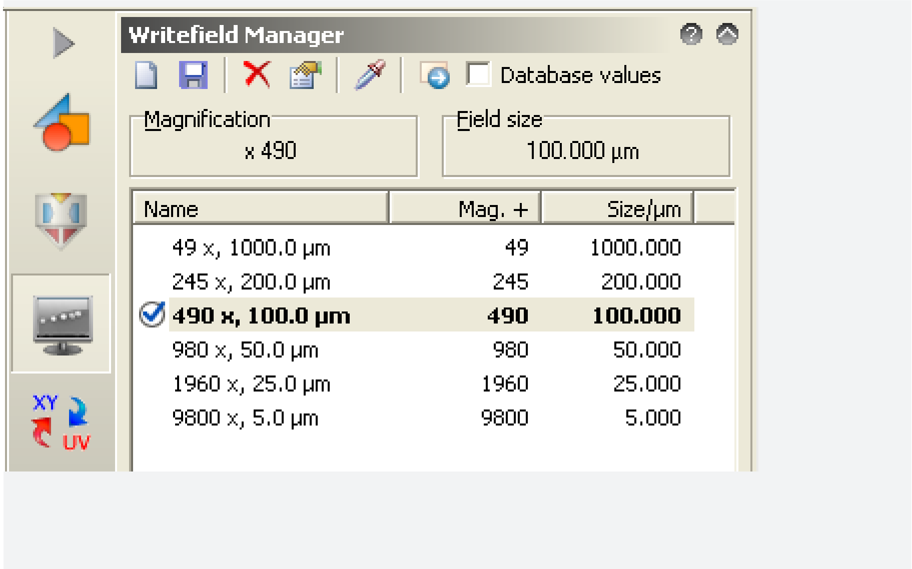


Figure 8-14 Select the **Writefield Manager** window.

☐ Database values

Magnification: x 490

Field size: 100.000 µm

Name	Mag. +	Size/µm
49 x, 1000.0 µm	49	1000.000
245 x, 200.0 µm	245	200.000
✓ **490 x, 100.0 µm**	**490**	**100.000**
980 x, 50.0 µm	980	50.000
1960 x, 25.0 µm	1960	25.000
9800 x, 5.0 µm	9800	5.000

STEP 3 ▶ Open a new **Positionlist**.

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

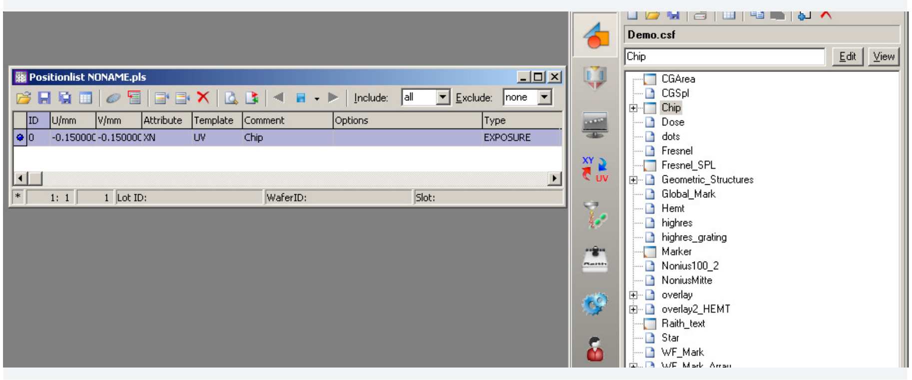


Figure 8-15 Opening a new **Positionlist**.

Include: all Exclude: none

ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type
0	-0.15000(	-0.15000(	XN	UV	Chip		EXPOSURE

* 1: 1 1 Lot ID: WaferID: Slot:

Chip

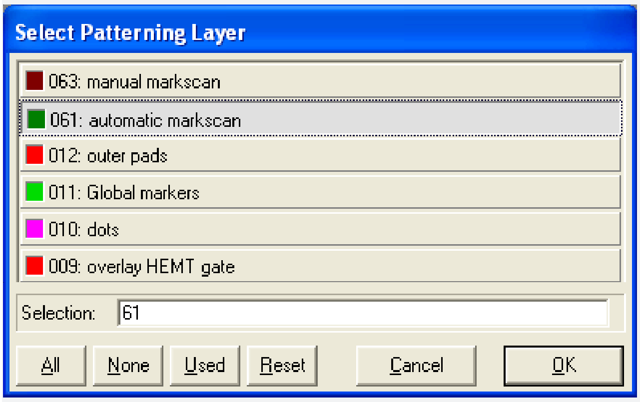
- CGArea
- CGSpl
- Chip
- Dose
- dots
- Fresnel
- Fresnel_SPL
- Geometric_Structures
- Global_Mark
- Hemt
- highres
- highres_grating
- Marker
- Nonius100_2
- NoniusMitte
- overlay
- overlay2_HEMT
- Raith_text
- Star
- WF_Mark

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**.

Figure 8-16 Select layer 61.

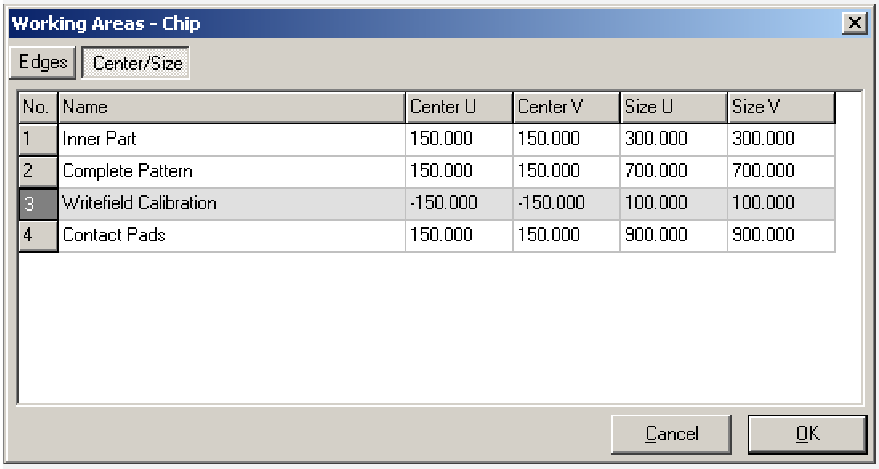


Selection: 61

STEP 6 ▶

Click on the **Working Area** icon.

Figure 8-17 Working Area.



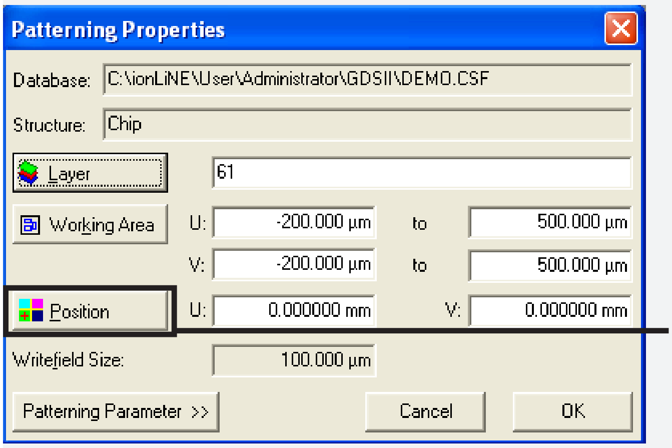
No.	Name	Center U	Center V	Size U	Size V
1	Inner Part	150.000	150.000	300.000	300.000
2	Complete Pattern	150.000	150.000	700.000	700.000
3	Writefield Calibration	-150.000	-150.000	100.000	100.000
4	Contact Pads	150.000	150.000	900.000	900.000

Select the work area **Writefield Calibration**. Confirm with **OK**.

STEP 7 ▶

Adjust the UV position by clicking the **Position** button.

Figure 8-18 Patterning Properties parameters.



Database: C:\ionLiNE\User\Administrator\GDSII\DEMO.CSF

Structure: Chip

Layer: 61

Working Area U: -200.000 µm to 500.000 µm

V: -200.000 µm to 500.000 µm

Position U: 0.000000 mm V: 0.000000 mm

Writefield Size: 100.000 µm

Press **Position** button.

This command will use the pre-defined working area and the Writefield size to calculate the correct sample UV position. It is very important to set-up the Writefield 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 auto mark scan during patterning 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.

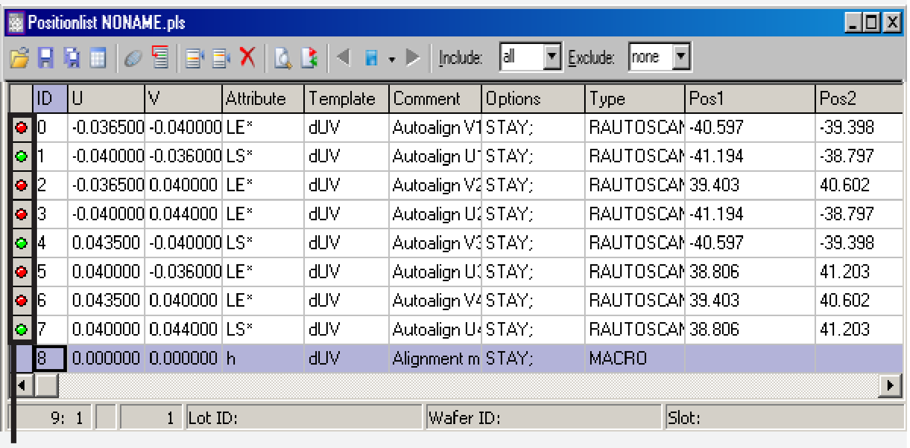
During the execution of the positionlist Align.pls we will be able to observe progress. Several line scans will be displayed, but it is unlikely that there will be a valid parameter set for mark detection within the line scanning and many errors will be shown. Once the execution of the positionlist is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

Figure 8-19 Setting up the automated writefield alignment procedure in the **Positionlist**.



Positionlist NONAME.pls

Include: all Exclude: none

	ID	U	V	Attribute	Template	Comment	Options	Type	Pos1	Pos2
🔴	0	-0.036500	-0.040000	LE*	dUV	Autoalign V1	STAY;	RAUTOSCAN	-40.597	-39.398
🟢	1	-0.040000	-0.036000	LS*	dUV	Autoalign U	STAY;	RAUTOSCAN	-41.194	-38.797
🔴	2	-0.036500	0.040000	LE*	dUV	Autoalign V2	STAY;	RAUTOSCAN	39.403	40.602
🔴	3	-0.040000	0.044000	LE*	dUV	Autoalign U	STAY;	RAUTOSCAN	-41.194	-38.797
🟢	4	0.043500	-0.040000	LS*	dUV	Autoalign V3	STAY;	RAUTOSCAN	-40.597	-39.398
🔴	5	0.040000	-0.036000	LE*	dUV	Autoalign U	STAY;	RAUTOSCAN	38.806	41.203
🔴	6	0.043500	0.040000	LE*	dUV	Autoalign V4	STAY;	RAUTOSCAN	39.403	40.602
🟢	7	0.040000	0.044000	LS*	dUV	Autoalign U	STAY;	RAUTOSCAN	38.806	41.203
	8	0.000000	0.000000	h	dUV	Alignment m	STAY;	MACRO		

9: 1 1 Lot ID: Wafer ID: Slot:

Green indicator light = successfully executed
Blue indicator light = not used
Red indicator light = error

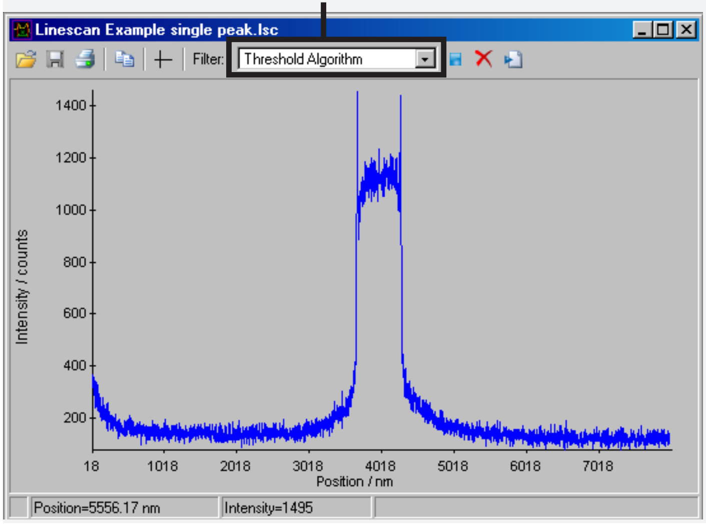
As we have not completed the optimization yet, the indicator light is displayed in red, since the Line scan could not be completed successfully. The corresponding Attributes show LE for Line scan error.

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 click on the **Apply** button.

Figure 8-20 The **Linescan** is now displayed.

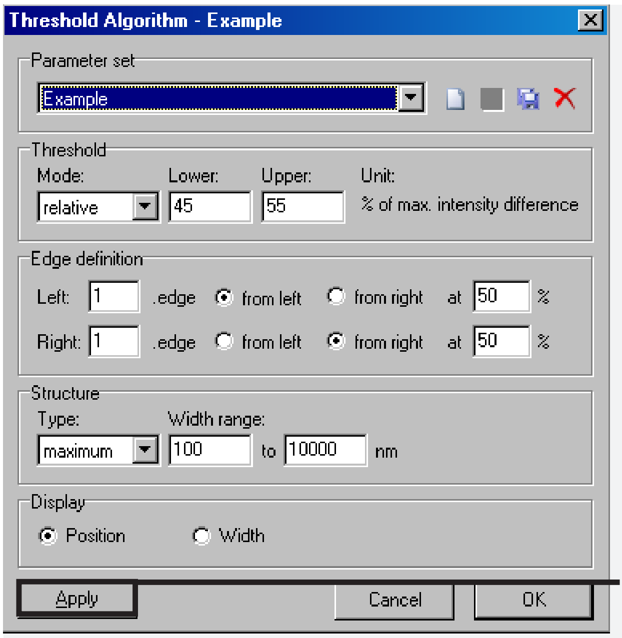
Select the Threshold Algorithm.



Linescan Example single peak.lsc

Filter: Threshold Algorithm

Position=5556.17 nm Intensity=1495



Threshold Algorithm - Example

Parameter set

Example

Threshold

Mode: relative

Lower: 45

Upper: 55

Unit: % of max. intensity difference

Edge definition

Left: 1 .edge ● from left ○ from right at 50 %

Right: 1 .edge ○ from left ● from right at 50 %

Structure

Type: maximum

Width range: 100 to 10000 nm

Display

● Position ○ Width

Click on the Apply button.

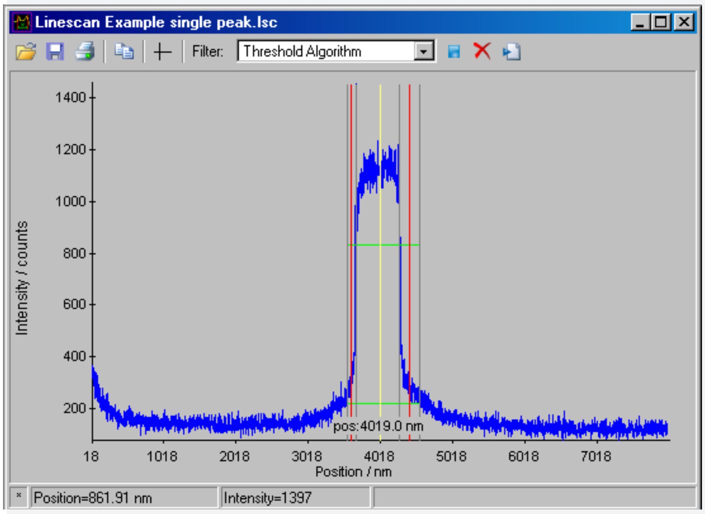
STEP 11 ▶

Select the parameter called **Writefield alignment** 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.

Figure 8-21 Applying the **Threshold** Algorithm.

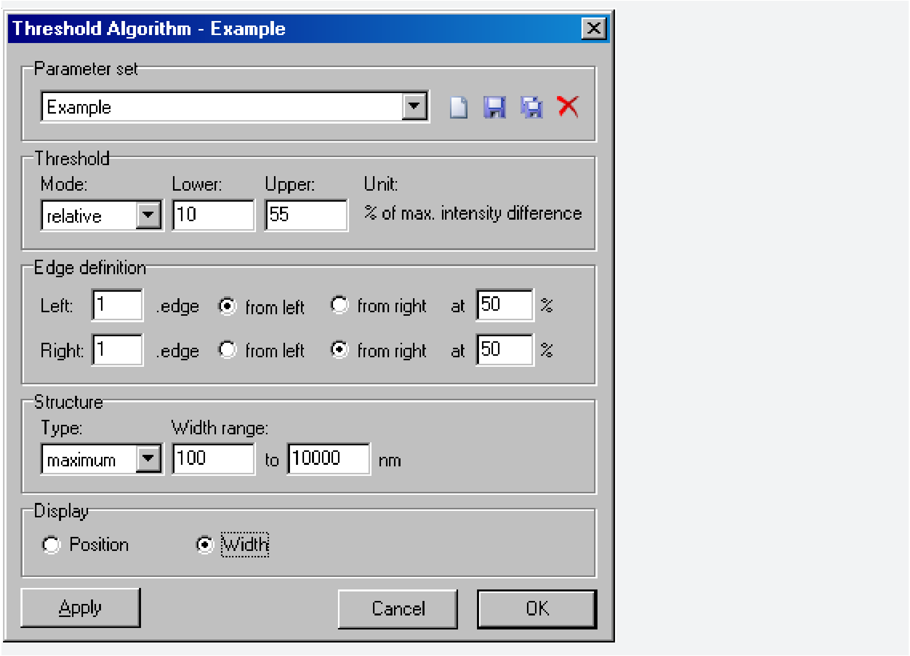


Linescan Example single peak.lsc

Filter: Threshold Algorithm

Position=861.91 nm

Intensity=1397



Threshold Algorithm - Example

Parameter set

Example

Threshold

Mode: relative

Lower: 10

Upper: 55

Unit: % of max. intensity difference

Edge definition

Left: 1 .edge ● from left ○ from right at 50 %

Right: 1 .edge ○ from left ● from right at 50 %

Structure

Type: maximum

Width range: 100 to 10000 nm

Display

○ Position ● Width

STEP 13 ▶

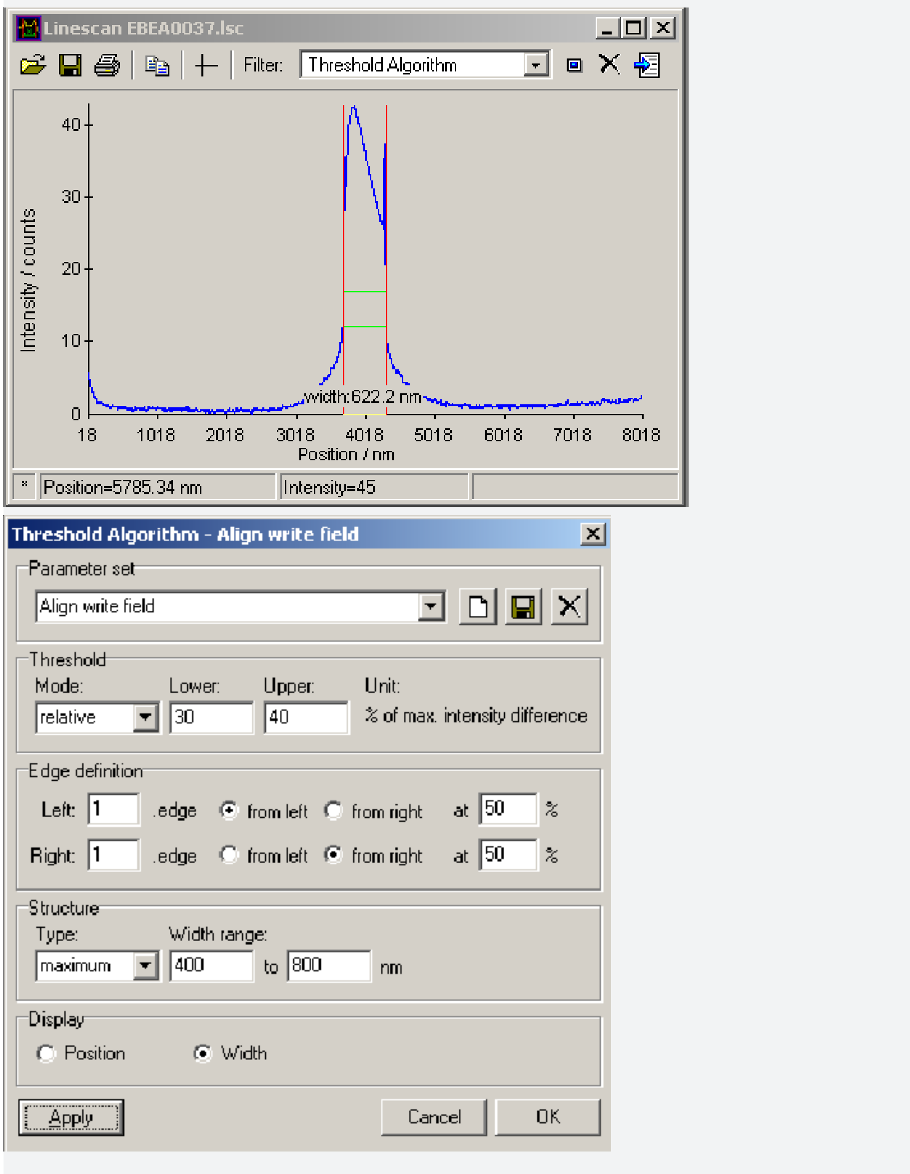
The next step is to optimize the parameter set. In our example, increase the **Lower Threshold** value.

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 thresholds of 50% and 70% were not well selected. By reducing both thresholds to 30% to 40%, improved results were achieved.

Figure 8-22 Changing the **Threshold** values.



Parameter set: Align write field

Threshold — Mode: relative; Lower: 30; Upper: 40; Unit: % of max. intensity difference

Edge definition — Left: 1 .edge ● from left ○ from right at 50 %

Right: 1 .edge ○ from left ● from right at 50 %

Structure — Type: maximum; Width range: 400 to 800 nm

Display — ○ Position ● Width

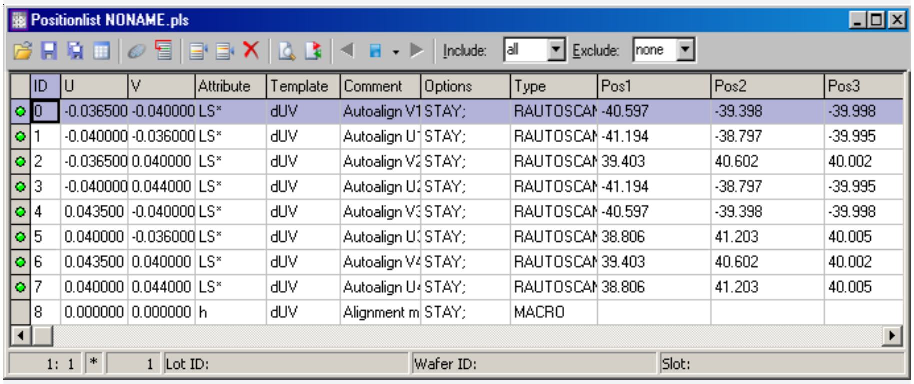
Since we have now defined the parameter set, the software will be able to detect the line successfully in the **Threshold Algorithm** window. Save the parameters and close the window with **OK**. In addition, close only the Line scan window but leave the positionlist **Align.pls** open.

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.

Figure 8-23 Scan All positions in the Positionlist.



Include: all Exclude: none

	ID	U	V	Attribute	Template	Comment	Options	Type	Pos1	Pos2	Pos3
●	0	-0.036500	-0.040000	LS*	dUV	Autoalign V1	STAY;	RAUTOSCAN	-40.597	-39.398	-39.998
●	1	-0.040000	-0.036000	LS*	dUV	Autoalign U	STAY;	RAUTOSCAN	-41.194	-38.797	-39.995
●	2	-0.036500	0.040000	LS*	dUV	Autoalign V	STAY;	RAUTOSCAN	39.403	40.602	40.002
●	3	-0.040000	0.044000	LS*	dUV	Autoalign U	STAY;	RAUTOSCAN	-41.194	-38.797	-39.995
●	4	0.043500	-0.040000	LS*	dUV	Autoalign V	STAY;	RAUTOSCAN	-40.597	-39.398	-39.998
●	5	0.040000	-0.036000	LS*	dUV	Autoalign U	STAY;	RAUTOSCAN	38.806	41.203	40.005
●	6	0.043500	0.040000	LS*	dUV	Autoalign V	STAY;	RAUTOSCAN	39.403	40.602	40.002
●	7	0.040000	0.044000	LS*	dUV	Autoalign U	STAY;	RAUTOSCAN	38.806	41.203	40.005
	8	0.000000	0.000000	h	dUV	Alignment m	STAY;	MACRO			

1: 1 * 1 Lot ID: Wafer ID: Slot:

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 Patterning

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 **Patterning 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 µAs/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 on your selection of either automated procedure (Layer 61) or semi-automated procedure (63), user interactions may be required. After completion, the sample can be developed and inspected.

9 Automation

AIM

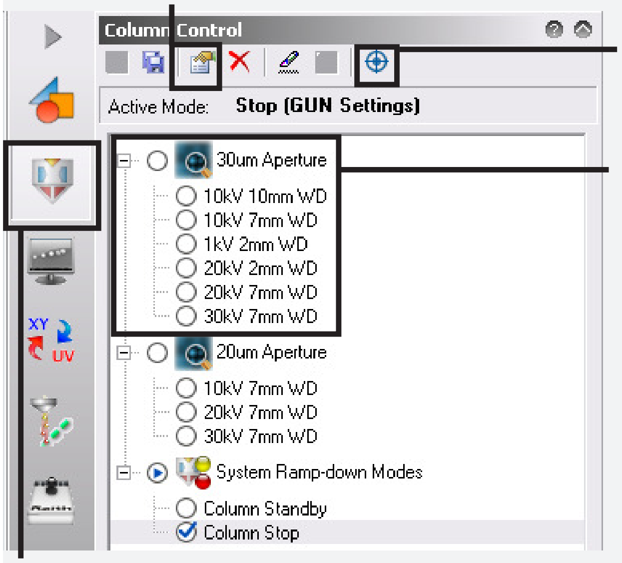
The aim of this chapter is to explain the automated features within the Column Control. The parameters voltage, aperture and working distances can be selected and automatically initiated from the positionlist. It is also possible to start and stop the Column or to select standby from the positionlist. In addition, an automated Writefield alignment can be initiated from the positionlist.

Task 1 Setting Column Control parameters

STEP 1 ▶

Click on the **Column Control** icon in the **control** bar to open the **Column Control** window. In Column Control you can **Start** and **Stop** the column and the **EHT**. You can also set **Voltages** with **Apertures**, as well as the **Working distances** automatically.

Click on this icon to Edit Dataset Values.



Column Control

Active Mode: **Stop (GUN Settings)**

- ○ 30um Aperture
  - ○ 10kV 10mm WD
  - ○ 10kV 7mm WD
  - ○ 1kV 2mm WD
  - ○ 20kV 2mm WD
  - ○ 20kV 7mm WD
  - ○ 30kV 7mm WD
- ○ 20um Aperture
  - ○ 10kV 7mm WD
  - ○ 20kV 7mm WD
  - ○ 30kV 7mm WD
- ● System Ramp-down Modes
  - ○ Column Standby
  - ☑ Column Stop

Click on this icon to activate the selected mode.

You can select different Apertures with Voltages and Working Distance combinations.

Select the Column Control icon.

Figure 9-1 Column Control Parameters.

HINT

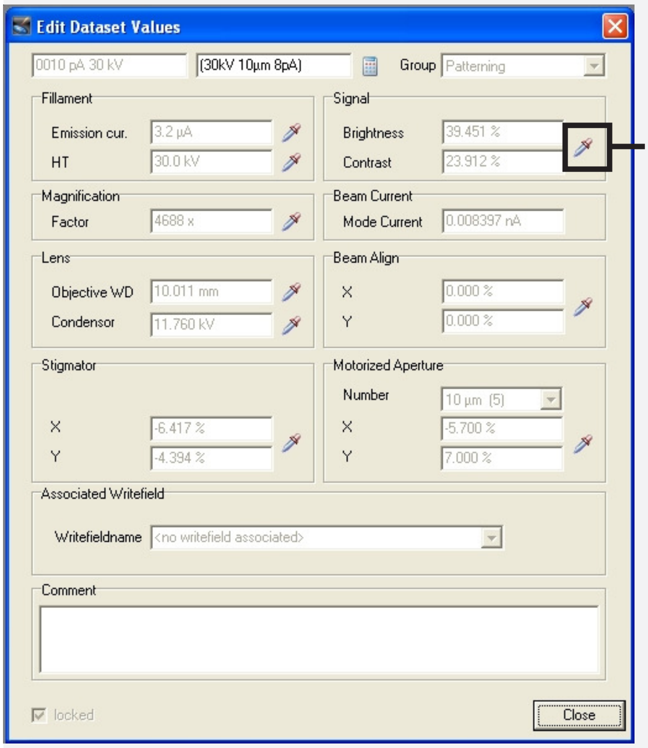
It is highly recommended to edit these parameters only via Column Control, not via the Raith EO software, since the Column Control automatically ramps up to the selected setting, thus avoiding any damage to the system,

In Column Control, if you check **Column Standby**, the vacuum will be maintained, but the EHT will be switched off and the column will be kept running.

Column Stop will stop the column.

STEP 2 ▶

To **Edit Dataset Values**, you can either double click on the selected voltage and working distance or alternatively you can click the **Edit Dataset Values** icon. This will open a new window, Edit Dataset Values, in which all of the values for acceleration, voltages, detector, apertures, magnification etc. can be set.



Edit Dataset Values

0010 pA 30 kV
(30kV 10µm 8pA)
Group: Patterning

Fillament
Emission cur.: 3.2 µA
HT: 30.0 kV

Signal
Brightness: 39.451 %
Contrast: 23.912 %

Magnification
Factor: 4688 x

Beam Current
Mode Current: 0.008397 nA

Lens
Objective WD: 10.011 mm
Condensor: 11.760 kV

Beam Align
X: 0.000 %
Y: 0.000 %

Stigmator
X: -6.417 %
Y: -4.394 %

Motorized Aperture
Number: 10 µm (5)
X: -5.700 %
Y: 7.000 %

Associated Writefield
Writefieldname: <no writefield associated>

Comment

☑ locked

Click on the Current Values button to transfer the currently active values from the Raith EO software to the dataset.

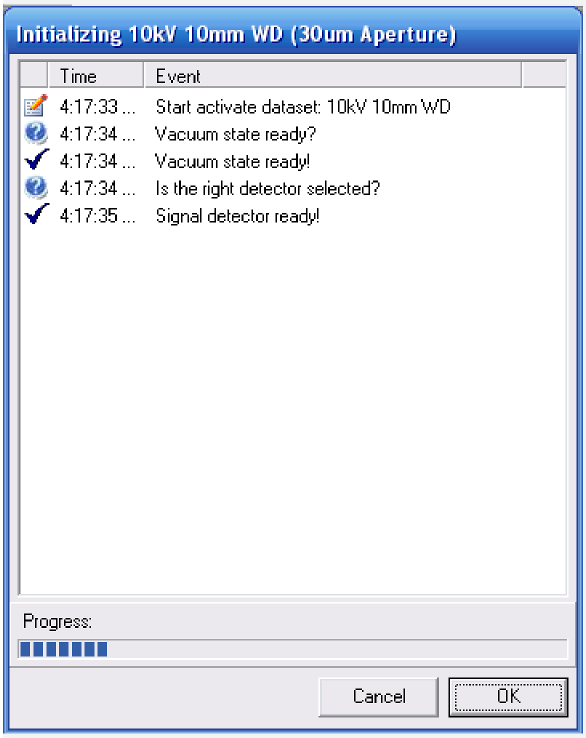
Figure 9-2 Edit Dataset Values Parameters.

HINT

Once you have edited all of your typical values, it is important to click the **Current Values** button. The values from the Column Control will be taken over into the dataset automatically.

To activate the settings, click on the icon **Activate Selected Mode** in the **Column Control** window.

Figure 9-3 Clicking on the icon **Activate Selected Mode** in the **Column Control** window will initiate this process.



Initializing 10kV 10mm WD (30um Aperture)

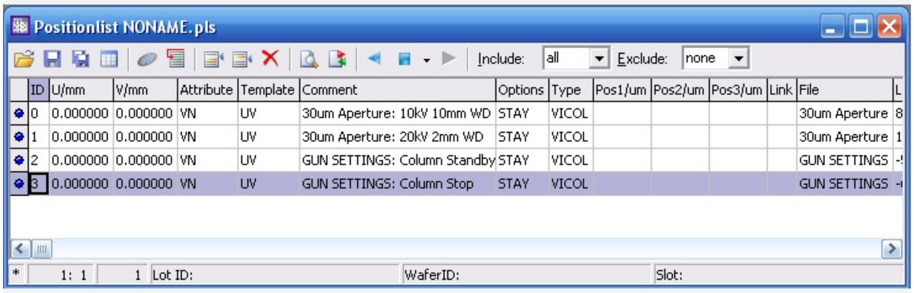
Time	Event
4:17:33 ...	Start activate dataset: 10kV 10mm WD
4:17:34 ...	Vacuum state ready?
4:17:34 ...	Vacuum state ready!
4:17:34 ...	Is the right detector selected?
4:17:35 ...	Signal detector ready!

Task 2 Activating Column Control in a Positionlist

STEP 1 ▶

To use these **Column Control** parameters, you can drag and drop them into the positionlist.

Figure 9-4 Drag & Drop **Column Control** parameters into **Positionlist**.



Positionlist NONAME.pls

Include: all Exclude: none

	ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type	Pos1/um	Pos2/um	Pos3/um	Link	File	L
●	0	0.000000	0.000000	VN	UV	30um Aperture: 10kV 10mm WD	STAY	VICOL					30um Aperture	8
●	1	0.000000	0.000000	VN	UV	30um Aperture: 20kV 2mm WD	STAY	VICOL					30um Aperture	1
●	2	0.000000	0.000000	VN	UV	GUN SETTINGS: Column Standby	STAY	VICOL					GUN SETTINGS	-
●	3	0.000000	0.000000	VN	UV	GUN SETTINGS: Column Stop	STAY	VICOL					GUN SETTINGS	-

* 1: 1 1 Lot ID: WaferID: Slot:

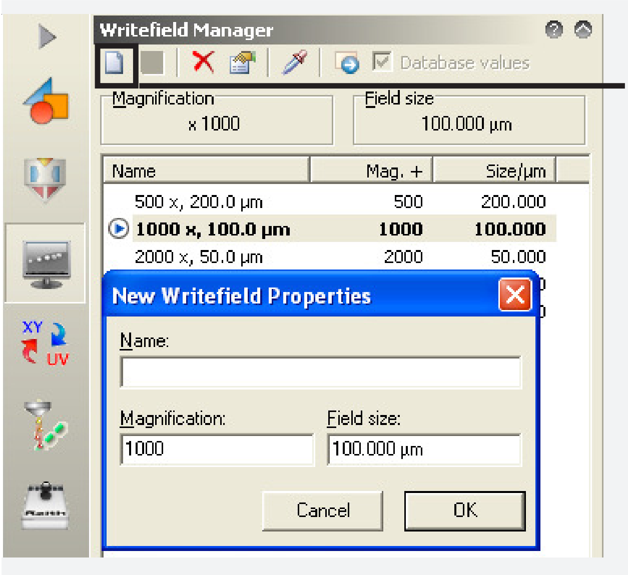
In the **Positionlist** shown in this example, we dragged and dropped the **10kV30umaperture** command into the first row. For the second position in the positionlist, we chose to use a higher voltage of 20 kV. It is also possible to include **Column Standby** and **Column Stop** into the positionlist. When the positionlist is executed, the column parameters will be changed accordingly.

Task 3 Automated Writefield Alignment

STEP 1 ▶

Go to the **Writefield Manager** window. We will now enter new Writefield properties. Click on the icon **Register new Writefield** and a new dialog box, **New Writefield properties,** will be displayed in which you can enter **Name**, **Magnification** and **Field size**.

Figure 9-5 New Writefield properties dialog.



Magnification: x 1000
Field size: 100.000 µm

Name	Mag. +	Size/µm
500 x, 200.0 µm	500	200.000
1000 x, 100.0 µm	**1000**	**100.000**
2000 x, 50.0 µm	2000	50.000

New Writefield Properties

Name:
Magnification: 1000
Field size: 100.000 µm

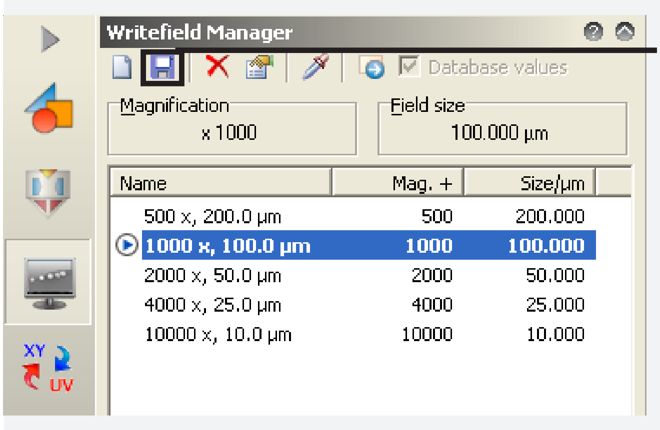
Click on the Register new Writefield icon.

At first, this new Writefield property value will be shown in **red**. This is to prompt the user that the new value has not been saved yet.

STEP 2 ▶

The next step is to **Save** the **New Writefield properties** to the database. The writefield definition will be taken over, as well as automatically saving the corresponding writefield alignment parameters. To save, click on the **Save** icon .

Figure 9-6 Saving the values.

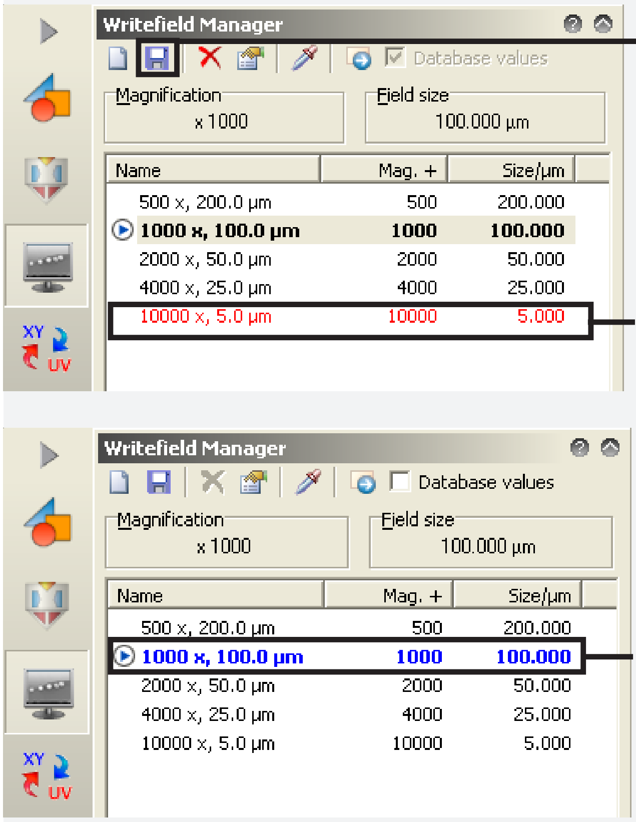


Magnification: x 1000
Field size: 100.000 µm

Name	Mag. +	Size/µm
500 x, 200.0 µm	500	200.000
1000 x, 100.0 µm	**1000**	**100.000**
2000 x, 50.0 µm	2000	50.000
4000 x, 25.0 µm	4000	25.000
10000 x, 10.0 µm	10000	10.000

Click on this icon to save the parameters.

Figure 9-7 Saving the data to **Database**.



Writefield Manager

☑ Database values

Magnification: x 1000

Field size: 100.000 µm

Name	Mag. +	Size/µm
500 x, 200.0 µm	500	200.000
1000 x, 100.0 µm	**1000**	**100.000**
2000 x, 50.0 µm	2000	50.000
4000 x, 25.0 µm	4000	25.000
10000 x, 5.0 µm	10000	5.000

Click on the Save icon to save the parameters.

Values are displayed in red to prompt user to save them .

Writefield Manager

☐ Database values

Magnification: x 1000

Field size: 100.000 µm

Name	Mag. +	Size/µm
500 x, 200.0 µm	500	200.000
1000 x, 100.0 µm	**1000**	**100.000**
2000 x, 50.0 µm	2000	50.000
4000 x, 25.0 µm	4000	25.000
10000 x, 5.0 µm	10000	5.000

Blue values alert the user that the values in the Writefield Manager differ from the saved Database file.

HINT

If the position in the **Writefield Manager** window is displayed in blue, it alerts the user to the fact that the values in the saved Database file differ from those in the **Writefield Alignment** window.

HINT

It is always possible to work within the Writefield Alignment window, to carry out Writefield Alignment and to use the correct values for **Zoom**, **Shift** and **Rotation** in the window, without saving the parameters. To save them, click the **Save** icon in the Writefield Manager window.

STEP 3 ▶

You can drag & drop the **Writefield Alignment** into the **positionlist**. Executing the positionlist will set the Writefield Alignment values.

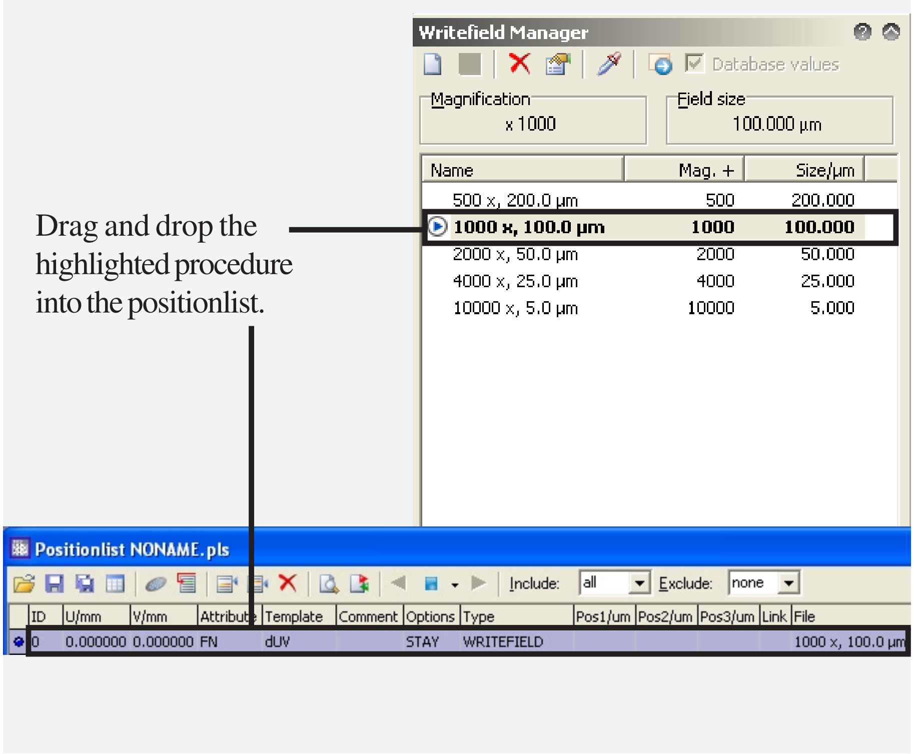


Figure 9-8 Drag & drop **Writefield Alignment** into positionlist.

Drag and drop the highlighted procedure into the positionlist.

Writefield Manager

☑ Database values

Magnification: x 1000

Field size: 100.000 µm

Name	Mag. +	Size/µm
500 x, 200.0 µm	500	200.000
1000 x, 100.0 µm	**1000**	**100.000**
2000 x, 50.0 µm	2000	50.000
4000 x, 25.0 µm	4000	25.000
10000 x, 5.0 µm	10000	5.000

Positionlist NONAME.pls

Include: all Exclude: none

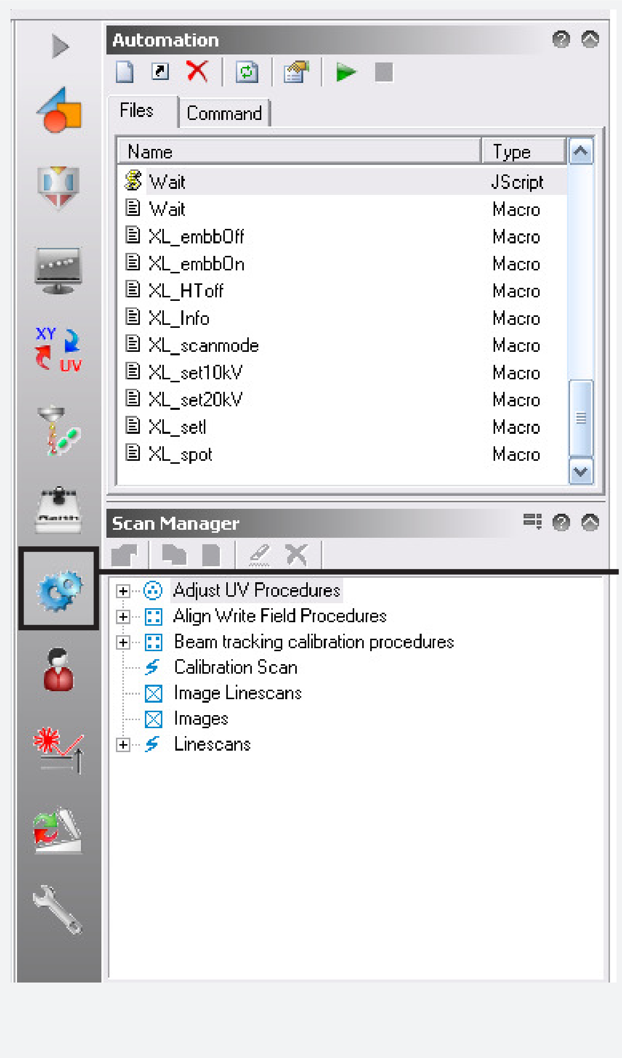
ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type	Pos1/um	Pos2/um	Pos3/um	Link	File
0	0.000000	0.000000	FN	dUV		STAY	WRITEFIELD					1000 x, 100.0 µm

Task 4 Further automation

STEP 1 ▶

It is also possible to drag and drop scripts into the positionlist.

If you open the **Automation** icon , a list of pre-written scripts and records will be displayed. These are saved in the **User Script** folder. The scripts and records can be dragged and dropped into the **positionlist**.



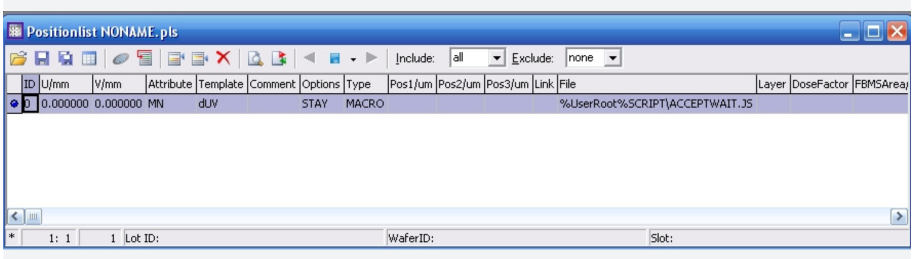
Name	Type
Wait	JScript
Wait	Macro
XL_embbOff	Macro
XL_embbOn	Macro
XL_HToff	Macro
XL_Info	Macro
XL_scanmode	Macro
XL_set10kV	Macro
XL_set20kV	Macro
XL_setI	Macro
XL_spot	Macro

Scan Manager

- Adjust UV Procedures
- Align Write Field Procedures
- Beam tracking calibration procedures
- Calibration Scan
- Image Linescans
- Images
- Linescans

Click on this icon to open the Automation window.

Figure 9-9 Automated window.



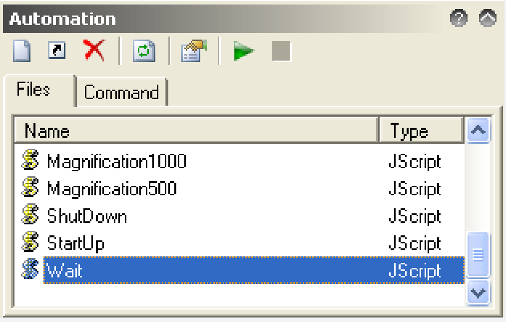
ID	U/mm	V/mm	Attribute	Template	Comment	Options	Type	Pos1/um	Pos2/um	Pos3/um	Link	File	Layer	DoseFactor	FBMSArea
0	0.000000	0.000000	MN	dUV		STAY	MACRO					%UserRoot%SCRIPT\ACCEPTWAIT.JS			

Figure 9-10 Drag and Drop Automation into position list.

STEP 2 ▶

To open the **Scripting Editor**, click on **Files** in the Automation window. Select the file script you wish to open and double click on it.

Figure 9-11 Drag and Drop **Automation** into position list.



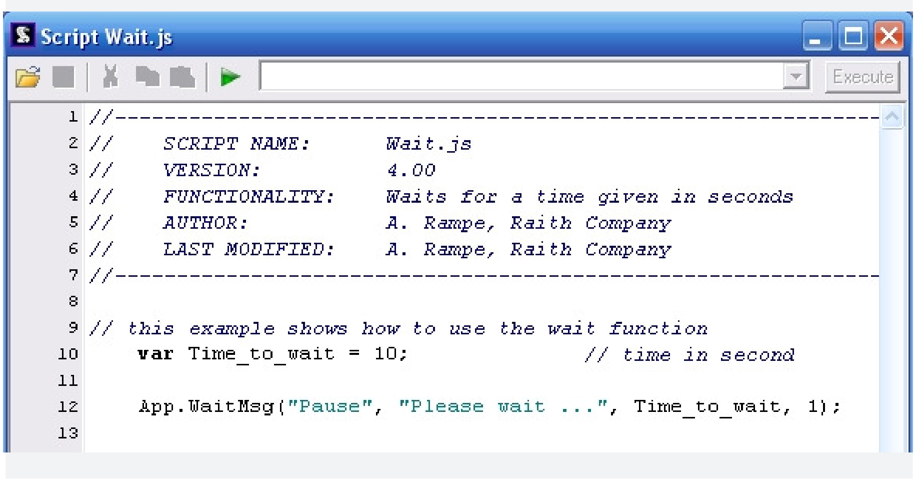
Name	Type
Magnification1000	JScript
Magnification500	JScript
ShutDown	JScript
StartUp	JScript
Wait	JScript

HINT

If you want to create or edit a script, you can open the **Scripting Editor** in the software. You can also create record files within the same editor. Any changes must be saved into the **User** folder. This will update the list in the **Automation** window, and saved items will become available for drag & drop into the positionlist.

Double click on the required script and a new window will open, displaying the details of the script.

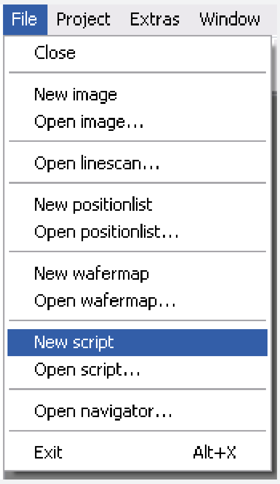
Figure 9-12 Viewing a Script.



```
//----------------------------------------------------------------
//    SCRIPT NAME:      Wait.js
//    VERSION:          4.00
//    FUNCTIONALITY:    Waits for a time given in seconds
//    AUTHOR:           A. Rampe, Raith Company
//    LAST MODIFIED:    A. Rampe, Raith Company
//----------------------------------------------------------------

// this example shows how to use the wait function
    var Time_to_wait = 10;                // time in second

    App.WaitMsg("Pause", "Please wait ...", Time_to_wait, 1);

```

To open a new script, go to **File > New Script**.

Figure 9-13 Opening a New Script.



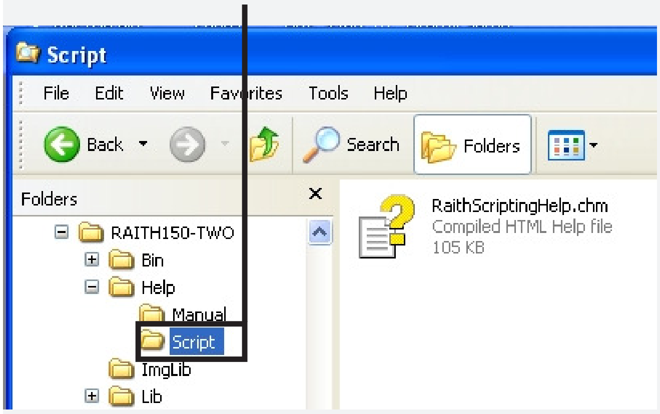
A new script can now be created.

HINT

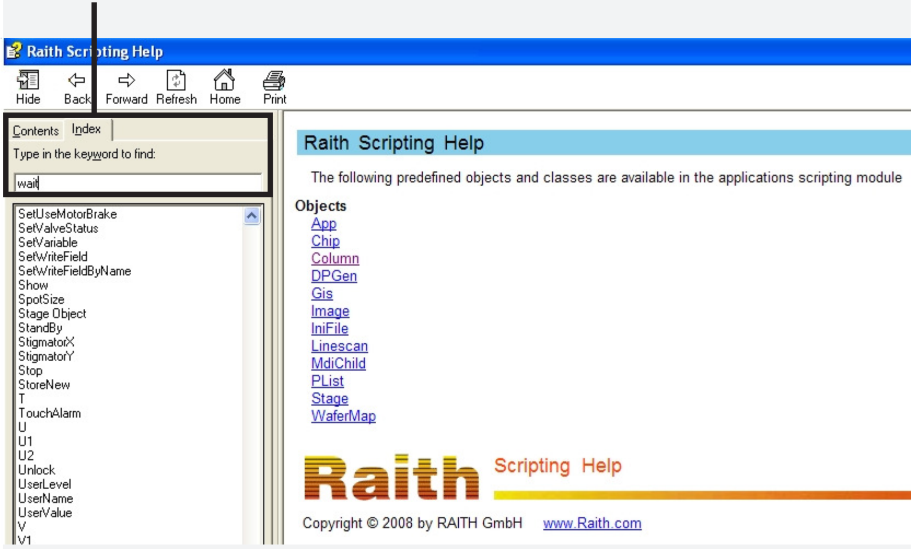
The **RAITH Scripting Help** teaches you all of the special commands for the RAITH software. The full scripting is based on Java script for internet files. In addition, you have the records files, into which commands from the **Command** tab can be dropped. These can be written and used for programming.

Figure 9-14 RAITH Scripting Help.

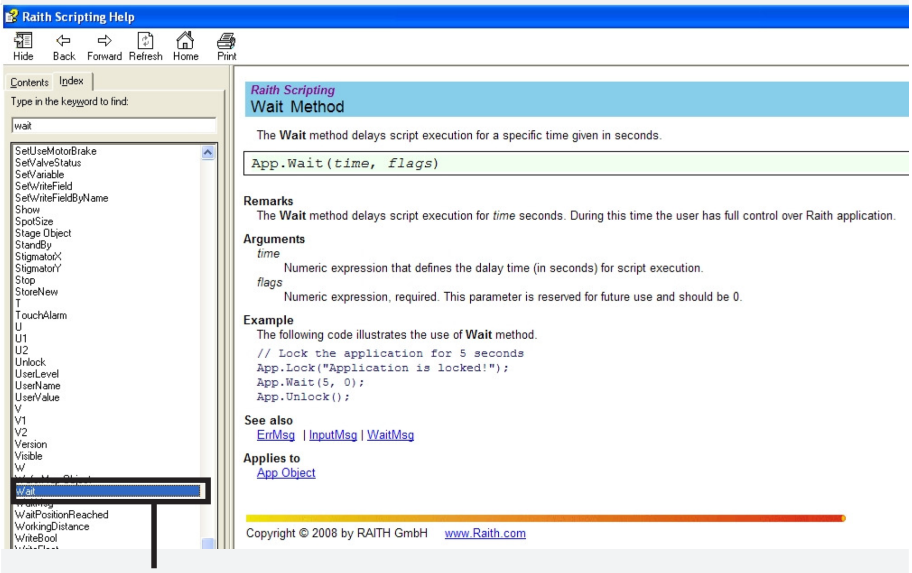
In order to access the **RAITH Scripting Help**, go to **Windows Explorer**, select your RAITH product. Then select Help then Script. Within the **Script** folder, double-click on **RaithScriptingHelp.chm**.



Click on the **Index** tab and then type **wait** into the keyword text field.



The following predefined objects and classes are available in the applications scripting module



Type in the keyword to find: wait

Raith Scripting

Wait Method

The **Wait** method delays script execution for a specific time given in seconds.

```
App.Wait(time, flags)
```

Remarks

The **Wait** method delays script execution for *time* seconds. During this time the user has full control over Raith application.

Arguments

time

Numeric expression that defines the dalay time (in seconds) for script execution.

flags

Numeric expression, required. This parameter is reserved for future use and should be 0.

Example

The following code illustrates the use of **Wait** method.

```
// Lock the application for 5 seconds
App.Lock("Application is locked!");
App.Wait(5, 0);
App.Unlock();
```

See also

ErrMsg | InputMsg | WaitMsg

Applies to

App Object

Copyright © 2008 by RAITH GmbH www.Raith.com

The word **wait** will then be highlighted in the list. Double click on the word **wait** to access the information.

10 Patterning on wafer

AIM

Before a patterning on wafer can be carried out, the user has to create a new wafer layout and carry out the wafer orientation. This tutorial will take the user through the steps required for creating a new, unpatterned wafer, performing the wafer adjustment using a flat on the side of the wafer and finally the Deskew procedure. Afterwards, the wafer exposure can be carried out.

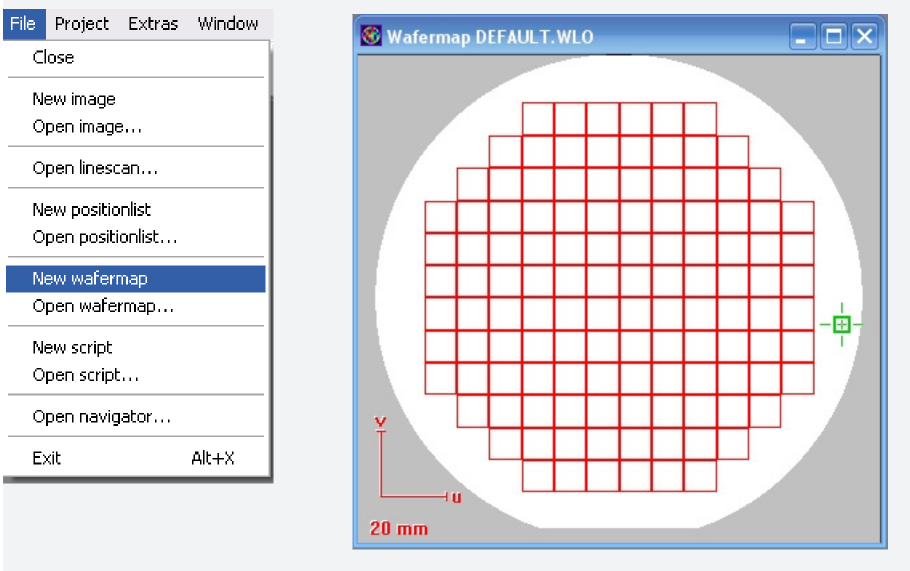
Task 1 Creating a Wafermap
Task 2 Performing the Wafer adjustment
Task 3 Performing the Deskew

Task 1 Creating a Wafermap

STEP 1 ▶

The first step is to either open or create a new wafermap. Go to **File>New wafermap**. A default wafermap window will open.

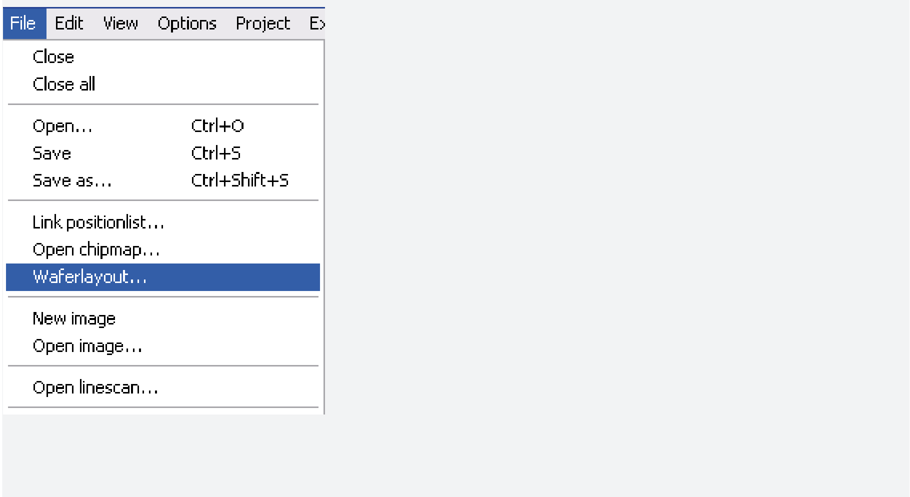
Figure 10-1 Opening the default **Wafermap.**



STEP 2 ▶

The next step is to define the wafer layout. Go to **File>Waferlayout**. A new window, **Edit Waferlayout**, is now displayed, in which all parameters can be edited.

Figure 10-2 Opening the **Waferlayout.**



HINT

If you want to use an unpatterned wafer, you must check the checkbox **Unpatterned wafer** in the **Edit Waferlayout** window. In our example, we will start with an unpatterned wafer.

STEP 3 ▶

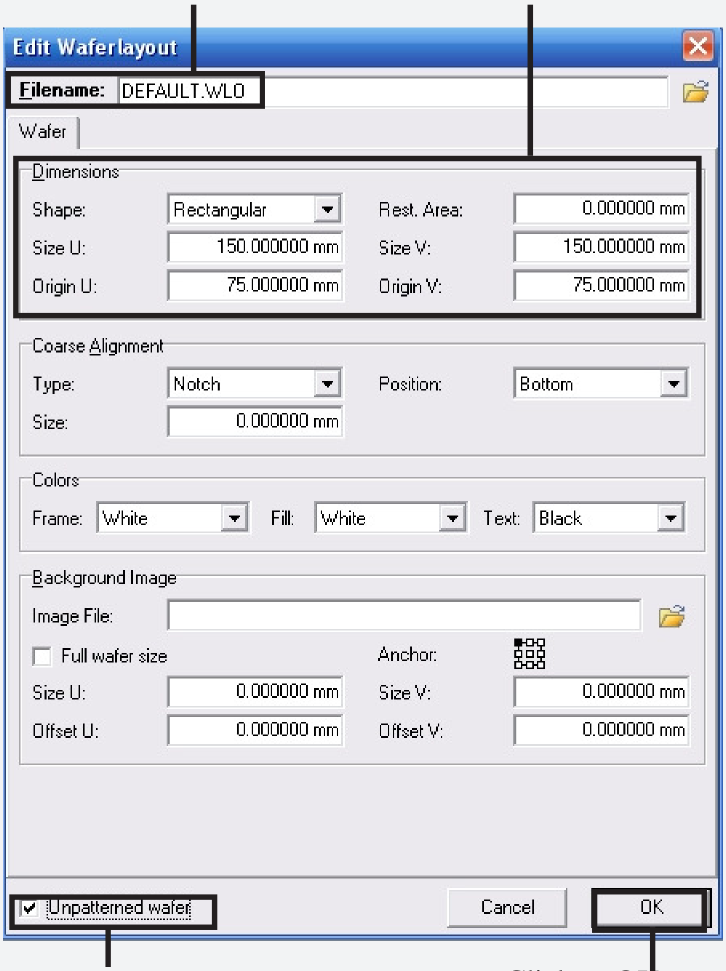
Check the checkbox **Unpatterned wafer.**
In the **Dimension** fields you must enter the dimensions of your unpatterned wafer.

You can enter the **Filename** at the top. Click **OK** to confirm.

Figure 10-3 Edit Waferlayout.

Enter a Filename.

Enter the dimensions of the unpatterned wafer. The size of 150 mm indicates that an unpatterned 6 inch wafer is being used.



Edit Waferlayout

Filename: DEFAULT.WLO

Wafer

Dimensions
Shape: Rectangular
Rest. Area: 0.000000 mm
Size U: 150.000000 mm
Size V: 150.000000 mm
Origin U: 75.000000 mm
Origin V: 75.000000 mm

Coarse Alignment
Type: Notch
Position: Bottom
Size: 0.000000 mm

Colors
Frame: White
Fill: White
Text: Black

Background Image
Image File:
☐ Full wafer size
Anchor:
Size U: 0.000000 mm
Size V: 0.000000 mm
Offset U: 0.000000 mm
Offset V: 0.000000 mm

☑ Unpatterned wafer

Check the checkbox for Unpatterned Wafer if you wish to work with an unpatterned wafer.

Click on OK to confirm.

A blank wafermap will now be displayed, showing a white field.

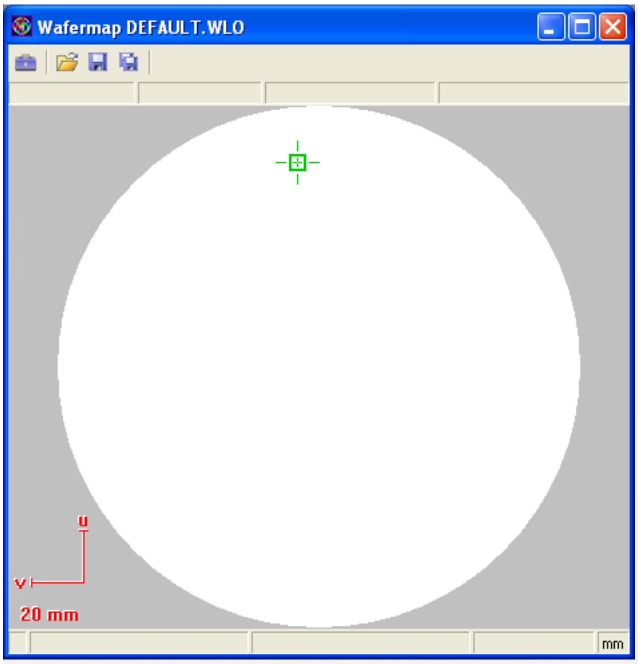


Figure 10-4 Showing the **Unpatterned Wafer.**

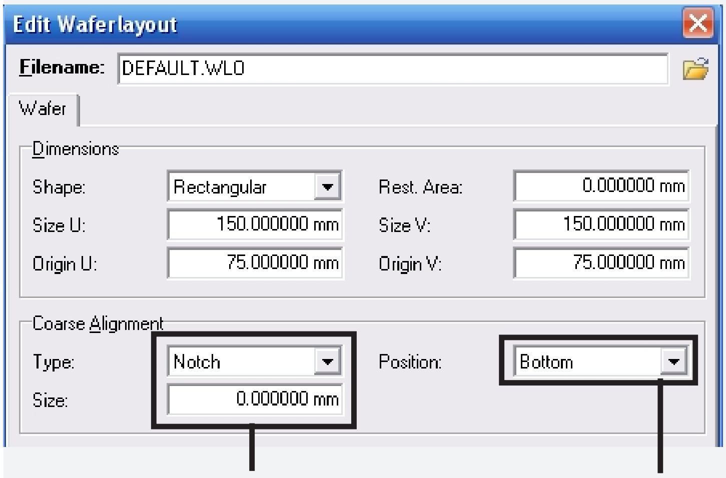
STEP 4 ▶

For easier wafer orientation, it is often useful to either create a Flat, Square or Notch on your wafer. Go back to **Edit Waferlayout**, **Coarse Alignment** and click on the downward arrow of the **Type** field. Select either **Major Flat**, **Square** or **Notch**. In our example, we will create a **Major Flat** on the left hand side of the wafer. The **Position** can be chosen by clicking on the downward arrow of the Position field. The selected Coarse Alignment will be shown on the waferlayout.

HINT

The flat helps with the orientation of a round wafer, to define the center of the wafer and for general orientation.

Figure 10-5 Creating a **Flat** on the **Unpatterned Wafer.**



Edit Waferlayout

Filename: DEFAULT.WLO

Wafer

Dimensions

Shape: Rectangular

Rest. Area: 0.000000 mm

Size U: 150.000000 mm

Size V: 150.000000 mm

Origin U: 75.000000 mm

Origin V: 75.000000 mm

Coarse Alignment

Type: Notch

Size: 0.000000 mm

Position: Bottom

You can create a Major Flat, Square or Notch onto your unpatterned wafer.

You can select the position of your Flat, Notch or square.

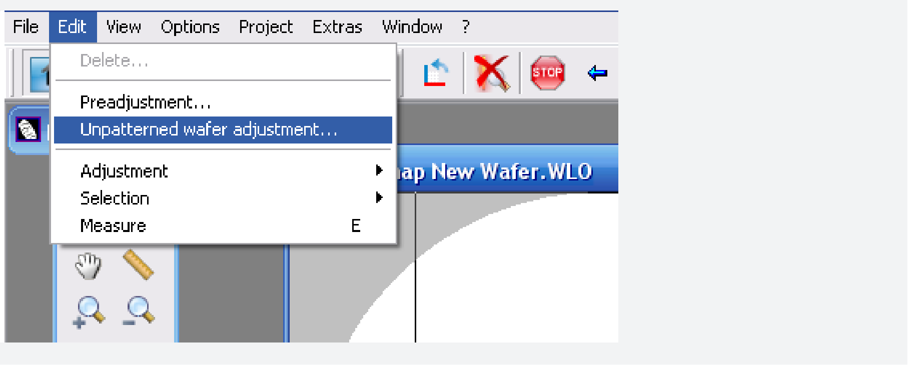
Task 2 Performing the wafer adjustment

STEP 1 ▶

Go to the **menu** bar **Edit> Unpatterned wafer adjustment.**

The **Wafer adjust** window will open, in which we can carry out the wafer adjustment using 3 marks.

Figure 10-6 Performing the **Wafer Adjustment.**



STEP 2 ▶

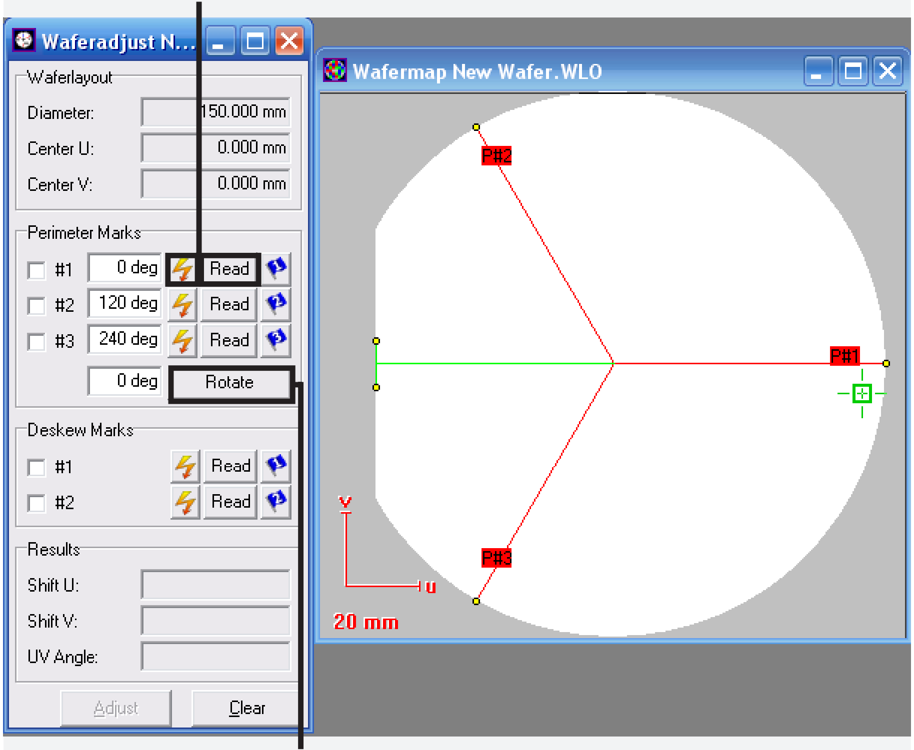
First go to the position using the **Flash** icon for **Perimeter Mark #1**, then activate the Column Control software. Using the **joystick**, locate the edge of the sample, then save this first mark position by clicking on the **Read** button.

STEP 3 ▶

Repeat the same procedure for the **Perimeter Marks #2** and **#3** positions.

Figure 10-7 Reading in the **Coordinates.**

Use the Flash icon to move to the Perimeter mark. Move to the selected position on your sample using the joystick. Confirm the coordinates by clicking the Read button.



Waferadjust N...

Waferlayout

Diameter: 150.000 mm

Center U: 0.000 mm

Center V: 0.000 mm

Perimeter Marks

☐ #1 0 deg

☐ #2 120 deg

☐ #3 240 deg

0 deg

Deskew Marks

☐ #1

☐ #2

Results

Shift U:

Shift V:

UV Angle:

Wafermap New Wafer.WLO

Click the Rotate button to obtain the center of your wafer.

STEP 4 ▶

Next, you will obtain the **Rotation**, which will now give you the center of your wafer. In this way, all of your structures will be positioned correctly on your wafer.

By entering a value next to the **Rotate** button, the three arms of your parameter marks will rotate.

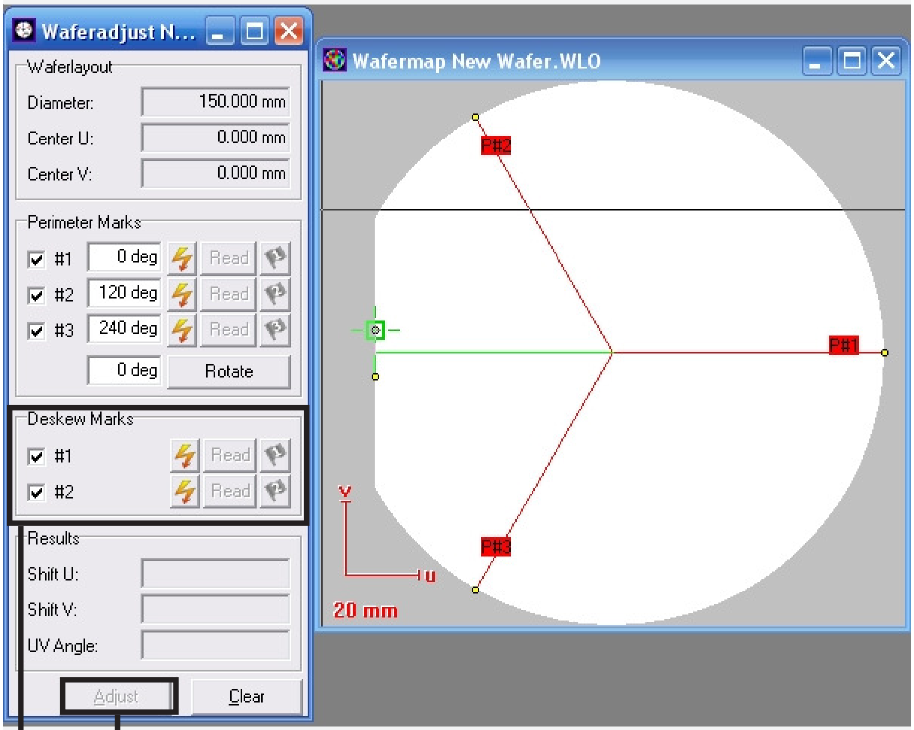
Task 3 Performing the Deskew

STEP 1 ▶

Finally, to give your structure an orientation on the wafer, and to be able to find this orientation again using the flat of the wafer, the so-called Deskew marks are saved by the software. The term Deskew refers to the process of correcting for the non-horizontal orientation of the surface of the wafer.

Check the checkboxes for **Deskew Marks**.

Figure 10-8 Performing the **Deskew.**



Waferadjust N...

Waferlayout
Diameter: 150.000 mm
Center U: 0.000 mm
Center V: 0.000 mm

Perimeter Marks
☑ #1 0 deg
☑ #2 120 deg
☑ #3 240 deg
0 deg

Deskew Marks
☑ #1
☑ #2

Results
Shift U:
Shift V:
UV Angle:

Click on Adjust to confirm the deskew.

Check the checkboxes for Deskew Marks.

To use the parameters again, you need to go through a location procedure using the parameter marks.

Drive, using the joystick, to locate the real edge position of your sample, then **Read** in the coordinates. Drive to the second position and then click Read to read in the coordinates.

Click on the **Adjust** button to confirm the Deskew.

The wafer layout and orientation are now completed.

Index