HNU-EBL

Manual

HNU-EBL Manual

Electron beam lithography simulation calculation EDA software

Developer: School of electrical and information engineering, Hunan University
Contact: Liu Jie , Jie_ liu@hnu.edu.cn , wechat: liujieuw

Contents

1 Overview

1.1 Introduction

Electron beam lithography (EBL) is an important technology for micro-nano fabrication of high-resolution nano-level lithography layouts, and it has a wide range of application prospects. However, the proximity effect (PE) in EBL will reduce the quality of the pattern, and direct exposure without the proximity effect correction (PEC) will have a devastating effect on the resolution. Since the 1970s, the International Lithography Organization has begun to develop the PE to EBL. Hunan University has independently developed the PEC software of EBL (HNU-EBL v1.0).

Up to now, one IWAPS paper has been published on ultrafast accurate proximity effect correction of large-scale electron beam lithography based on FMM and SaaS [1], and three patents: an electron beam proximity effect correction method based on neural network [2], and a high-precision proximity effect fast correction method for large-scale electron beam exposure layout [3], A proximity effect correction method of electron beam exposure based on two-dimensional fast Fourier transform [4]. Four soft works: electron beam exposure simulation and Correction Software v1.0 [5] Based on SaaS mode, electron beam lithography proximity effect correction Software v1.0 [6], electron beam lithography proximity effect correction Software v1.0 [7], GDSII file Visualization Software v1.0[8].

1.2 System Requirements

For best performance using HNU-EBL, we recommend a state of the art computer.

We suggest that your computer has at least

- CPU: 1.7 GHz with 4 cores
- RAM: 2GB
- Video Memory: 200MB

- HDD: 500MB

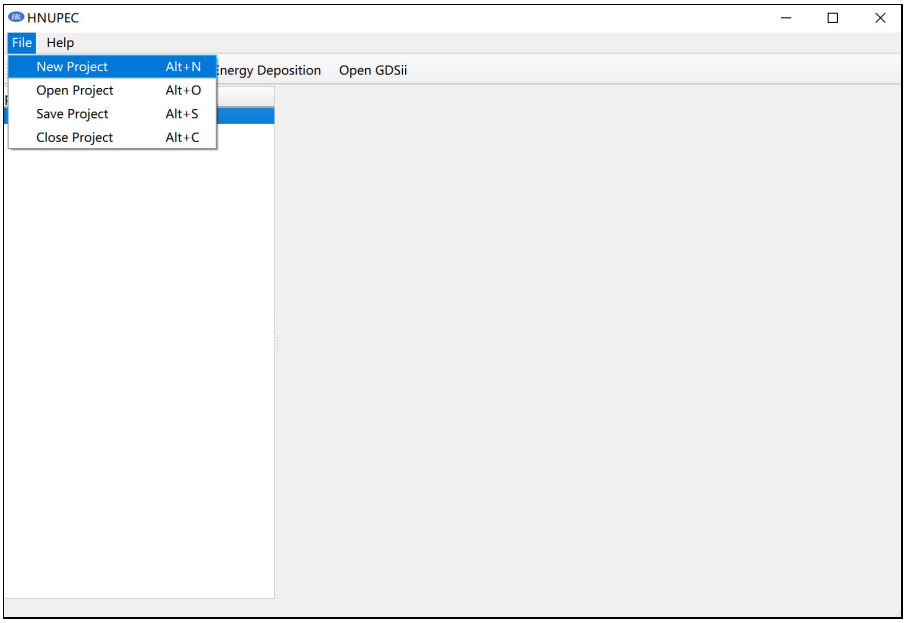
Supported platform are:

- Microsoft Windows: 7, 8, 10.

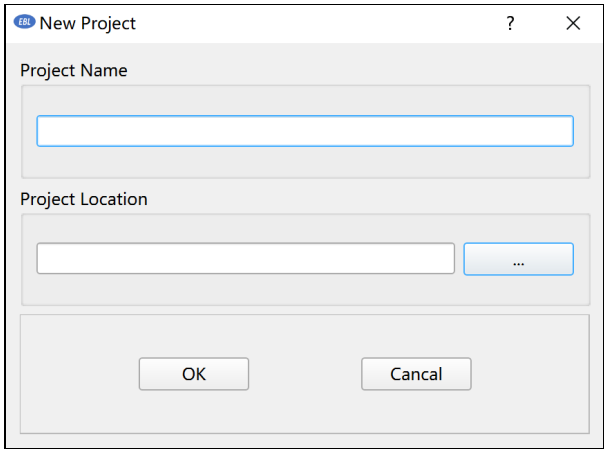
2 Calculation Module Operation Instructions

2.1 New Project

1) You can open the software and click the “File” and “New Project” to create a new project (Figure 2.1.1(a)). You need to fill in the project name and a path to save the project (Figure 2.1.1(b)). You can also click "Open Project" to open an existing project. (Note: The path can only contain English letters and symbols, not Chinese.)



(a)



(b)

Figure 2.1.1 New project

2.2 Enter Simulation Parameters

1) You can click “New Simulation” at the top left of the main interface of the software, and a dialog box will pop up (Figure 2.2.1). The Monte Carlo (MC) simulation can be performed after setting the relevant parameters.

2) You can enter material name in “Material” (Figure 2.2.1), such as “PMMA, Si” (selected in the drop-down menu), and enter the layer thickness in “Thickness”. The thickness of the "Layer n" layer should be 10nm to 5,000nm, and the thickness of the "Substrate" layer should be 1,000nm to 5mm. If there is no such material in the drop-down menu, you need to click “Edit Material” to add (Figure 2.2.2). For example, SiC is added (Figure 2.2.3).

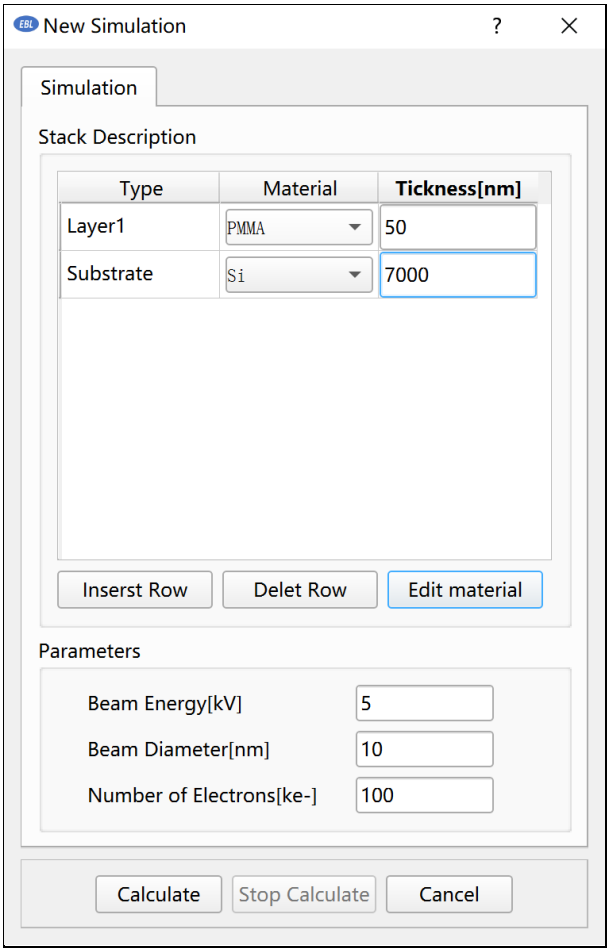


Figure 2.2.1 Enter MC simulation parameters

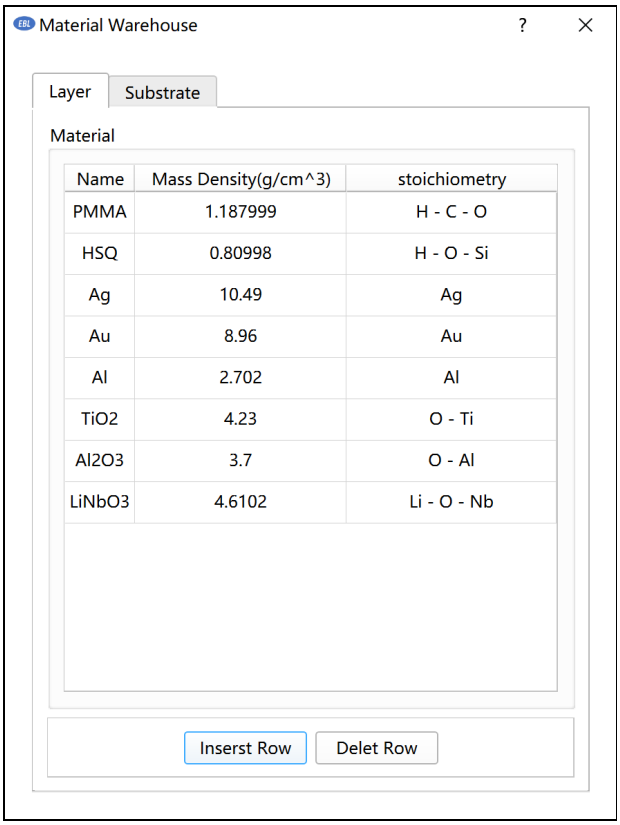


Figure 2.2.2 Edit material parameters

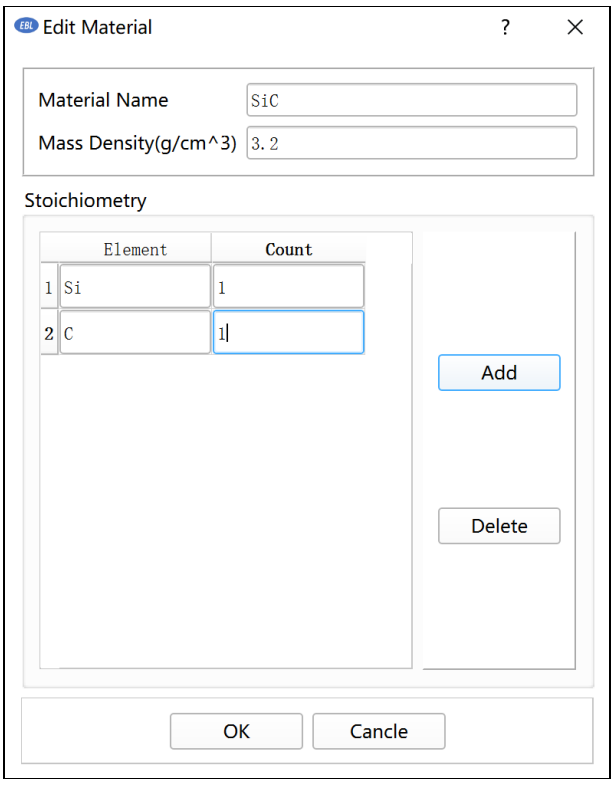


Figure 2.2.3 Add SiC and enter its parameters

3) You can click "Insert Row" to add a layer at a specific location or click "Delete Row" to delete the specified layer. The layer sequence is as shown in the table, from top to bottom Layer 1, Layer 2, ..., layer n, Substrate. (Note: The total number of layers is not less than two and the Substrate layer cannot be deleted.)

4) After adding all the layers, you need to input the simulated EBL related parameters, such as "Beam Energy, Beam Diameter, Number of Electrons" to complete the parameter input settings. If the input parameters are incomplete, the software will prompt the user to complete the parameter input.

2.3 Monte Carlo Simulation

1) You can click "Calculate" to perform MC simulation. After the calculation is completed, a dialog box will pop up to display the cost time. Click "Done" to complete the MC simulation (Figure 2.3.1).

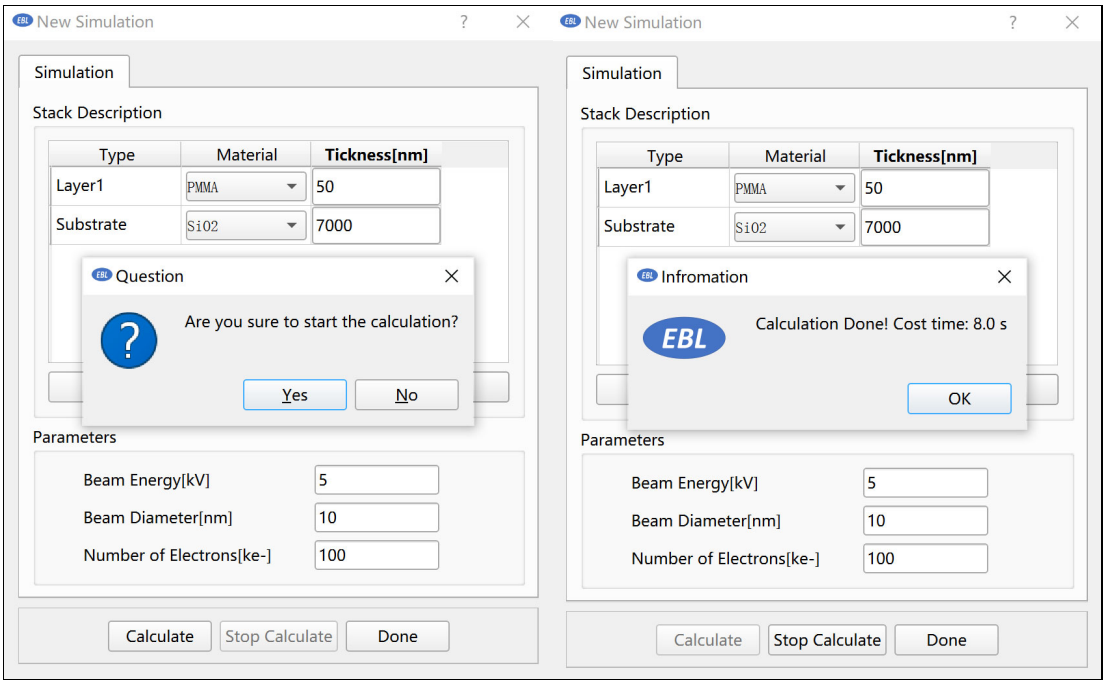


Figure 2.3.1 MC simulation

2) When the program is running, you can click "Stop Calculate" to terminate (Figure 2.3.2). You can re-enter the parameters and click "Calculate" again to start MC simulation.

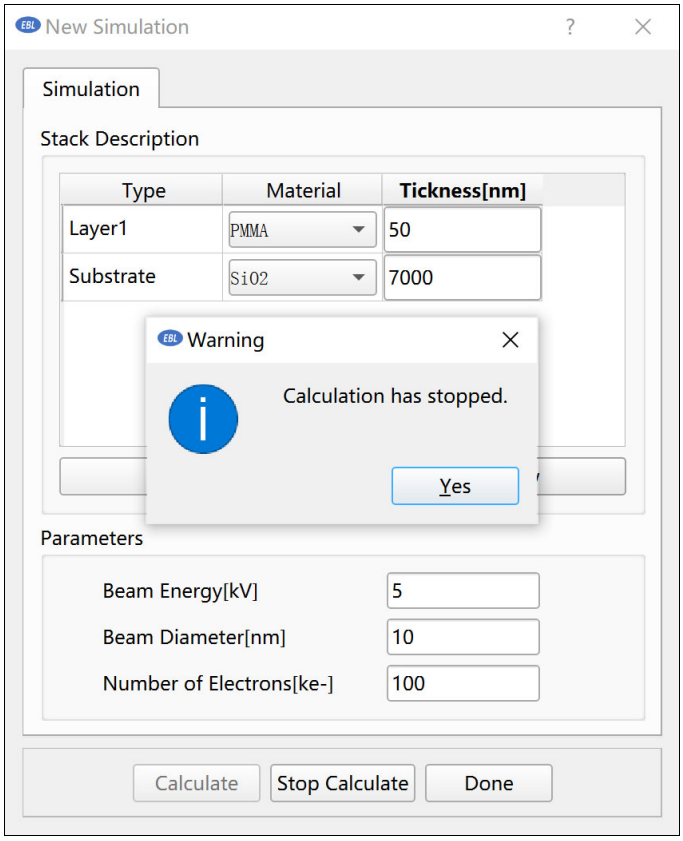


Figure 2.3.2 Stopping MC simulation

3) After the MC simulation is completed, you can click "Save Project" in the "File" at the top of the main interface of software to save the simulation results (Figure 2.3.3). (Note: Please click the "Save Project" button to save the project in time.)

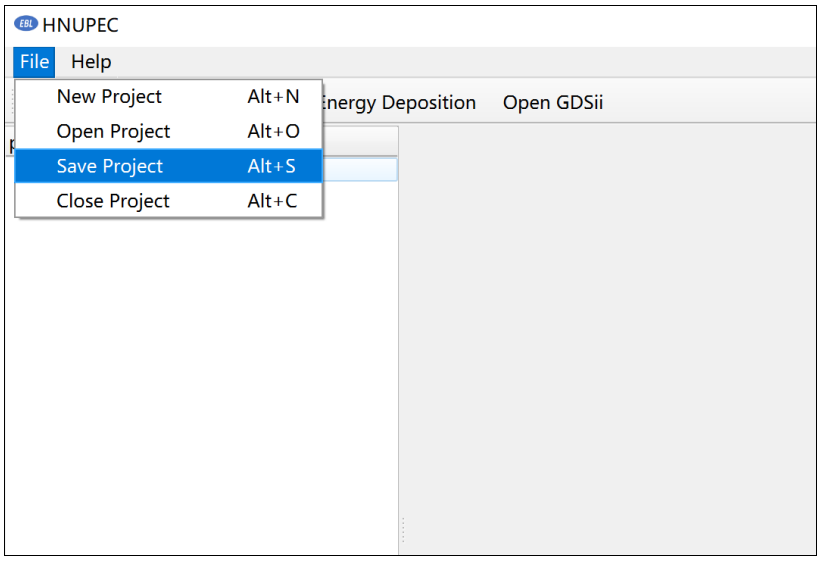


Figure 2.3.3 Save MC simulation result

4) You can right-click the MC simulation result on the left and left-click "delete" to delete the MC simulation result (Figure 2.3.4).

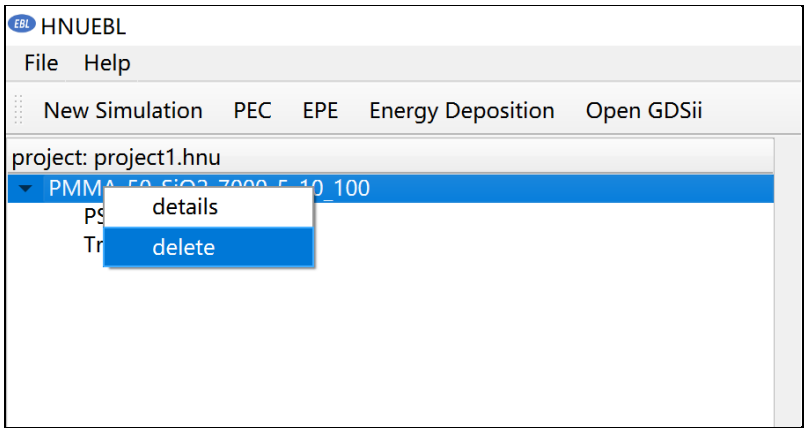


Figure 2.3.4 Delete MC simulation result

2.4 Point Spread Function

1) You can click the triangle symbol on the left of the corresponding MC simulation result to display the MC simulation result (Figure 2.4.1).

2) You can click “PSF” (Point Spread Function) to display the results of fitting MC simulation results with function. (For the convenience of observation, the X and Y coordinates are displayed in logarithm.)

3) You can click "X-linear" on the right to display the fitting result when the X coordinate is linear; click on the right "X-log" to display the fitting result when the X coordinate is logarithmic.

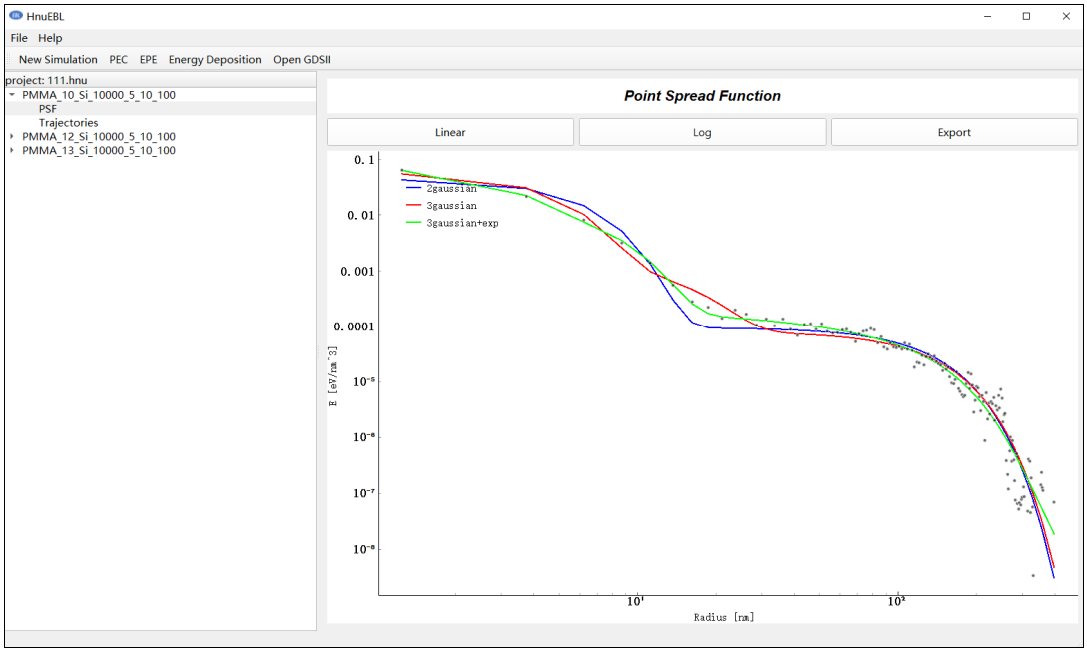


Figure 2.4.1 Fitting MC simulation

4) You can enter a specific X coordinate value or Y coordinate value on the right side and click "X-linear" or "X-log" to view the fitting curve within the specified range. (Note: The input value must be within the range of X and Y values.)

5) You can click "Numerical PSF" (Figure 2.4.1) to export the scatter point coordinate file.

6) If you want to view the fitting coefficients of the corresponding MC simulation results, you need to right-click the corresponding MC simulation result on the left and left-click "detail" to view (Figure 2.4.2)

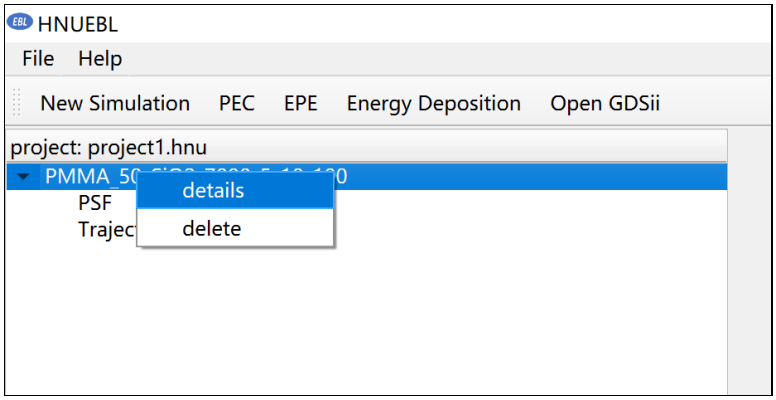


Figure 2.4.2 Coefficients of MC simulation result

7) In the interface of viewing the fitting coefficients of the corresponding MC simulation results (Figure 2.4.3), you can select a formula and change the coefficients, and click "Redraw" to view the fitting results of the PSF after changing the coefficients. (Note: The coefficient input value cannot be empty.)

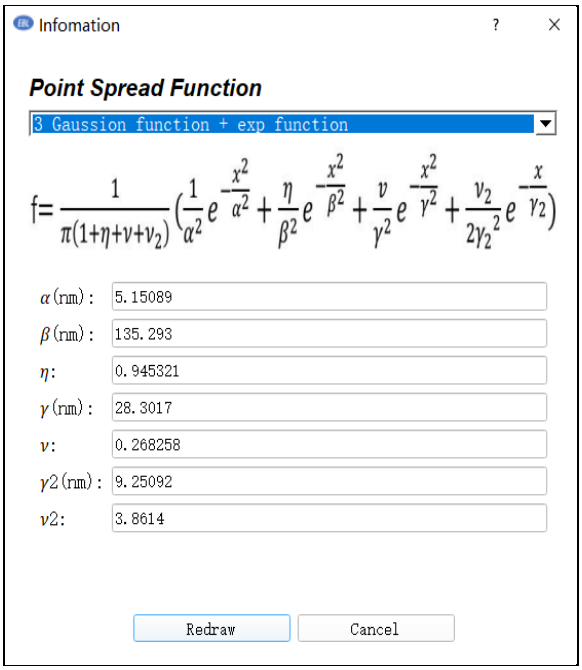


Figure 2.4.3 Change the fitting coefficient and redraw the curve

2.5 Electron Beam Scattering Simulation

1) You can click the triangle symbol on the left of the corresponding MC simulation result to display the MC simulation result.

2) You can click “Trajectories” to display the electron beam scattering simulation results (Figure 2.5.1). For the convenience of observation, the red marking lines are the layer names and thicknesses of all layers except the substrate in the corresponding MC simulation.

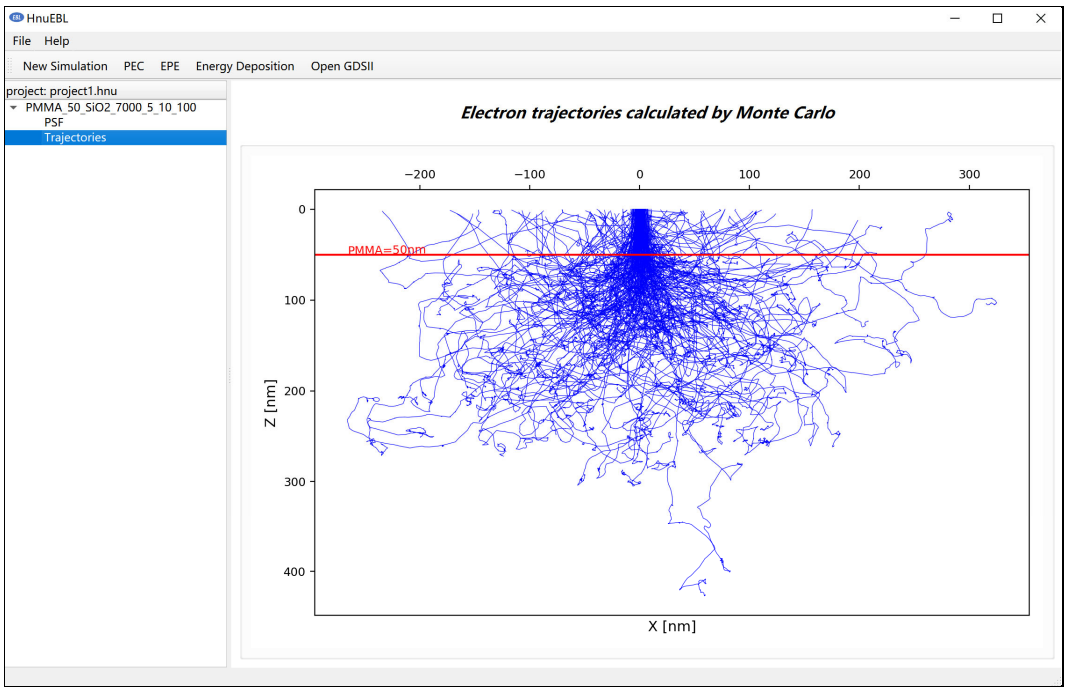


Figure 2.5.1 Electron beam scattering simulation

2.6 Proximity Effect Correction

1) You can click "PEC" on the toolbar at the top of the main interface (Figure 2.6.1) to complete the settings of PEC.

2) In the “Monte Carlo set” you can select a MC simulation result for PEC.

3) In the "fitting function", you need to select a fitting formula. The Figure 2.6.1 selects the three Gauss function for PEC.

4) In the "Matrix set" you can set the minimum size (nm) of the exposure unit. And 10nm is selected in the figure below. (Note: If the layout is large, the minimum size of the exposure unit should not be too small. If the size is too small and the computer memory is insufficient, a prompt will be given.)

5) "GDSII File" is used to open the “.gds” format layout to be corrected. (Note: The path can only contain English letters and symbols, not Chinese.)

6) "Result Folder" is used to set the output file path after the PEC completed. (Note: The path can only contain English letters and symbols, not Chinese.)

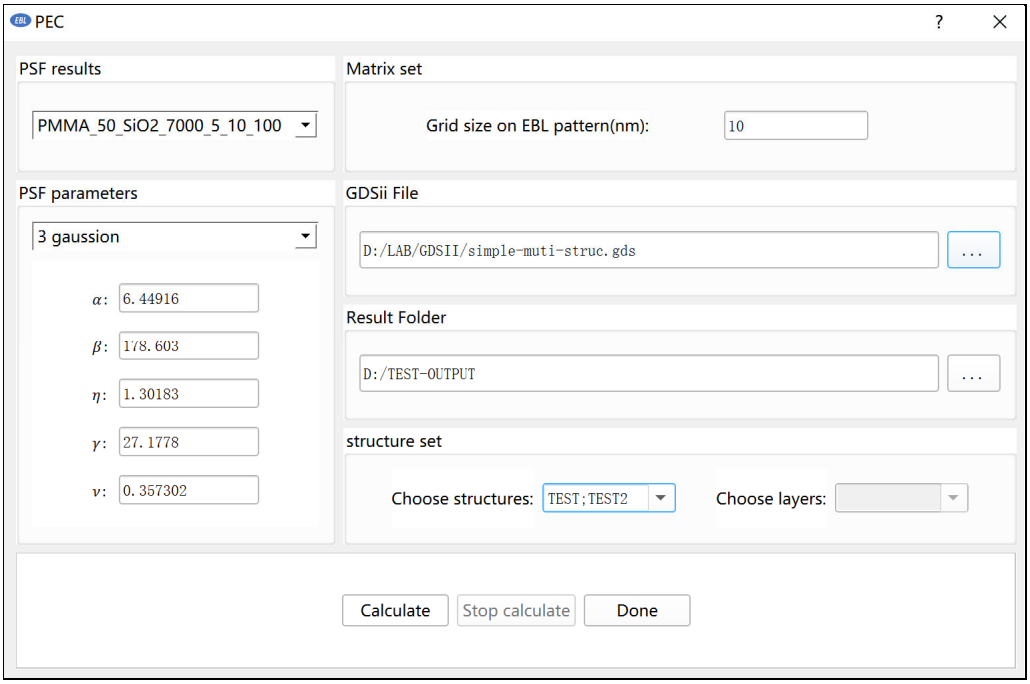


Figure 2.6.1 Set PEC

7) A GDSII file can contain multiple structures, and a structure can contain multiple layers. If users only want to perform PEC on part of the layout in the file, they can click on the "structure set" at the bottom of the page to further set the layout to be corrected. But you must select at least one for "Structure" and at least one for "Layer" (Figure 2.6.2 ((top)). If multiple “Structure” has been selected, you cannot select “Layer” again (Figure 2.6.2 (bottom)).

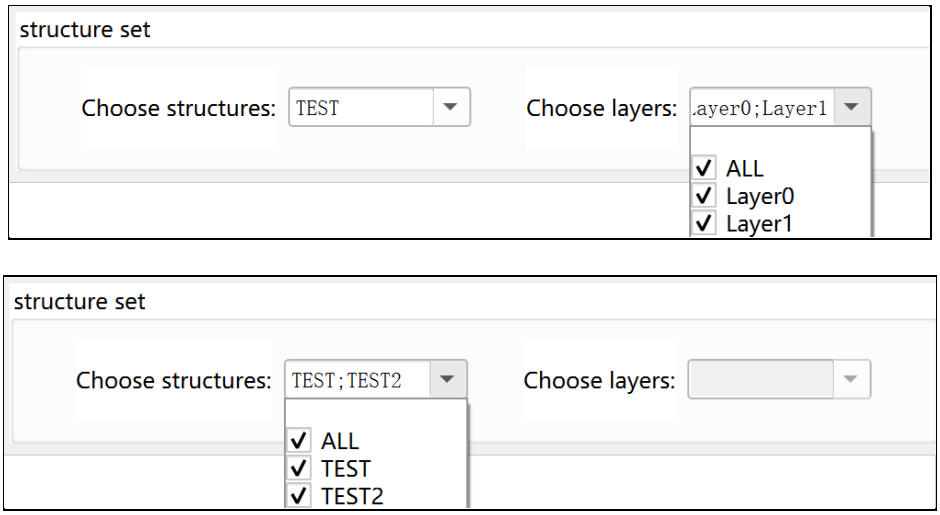


Figure 2.6.2 Set the layout to be corrected

8) You can click "Calculation" (Figure 2.6.3) to perform PEC. If the input parameters are incomplete, the calculation cannot be started. The calculation time is displayed after the calculation is completed (Figure 2.6.4). After the calculation is completed, you need to click "Done" to close the current window, and you can save the output file to the specified folder.

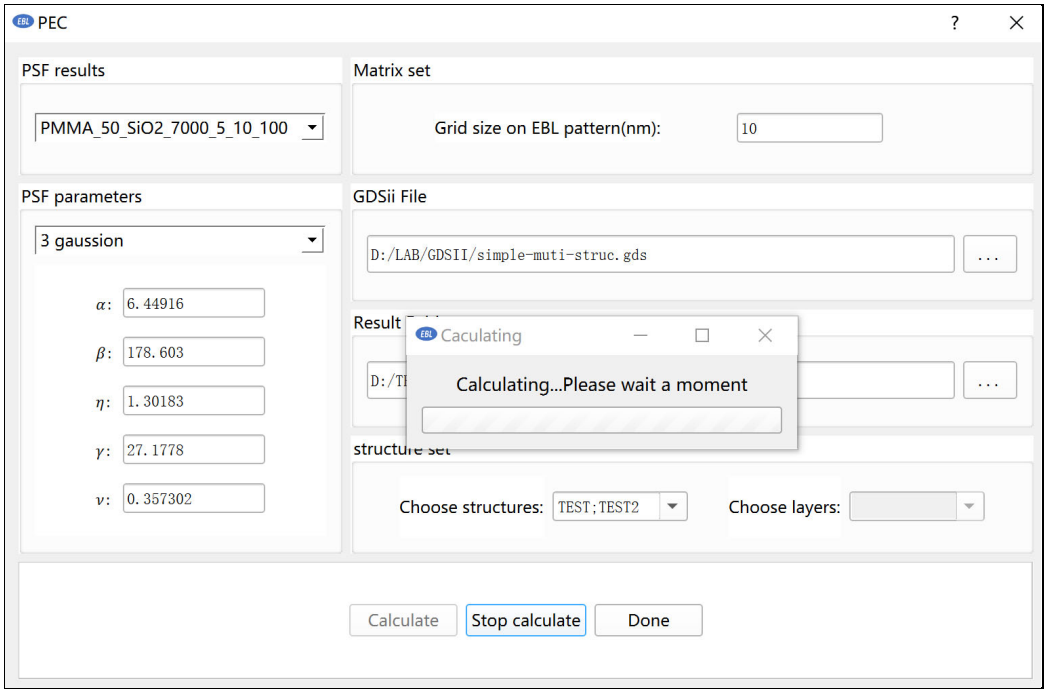


Figure 2.6.3 PEC calculating

9) You can click "Stop Calculation" to stop the current calculation (Figure 2.6.5).

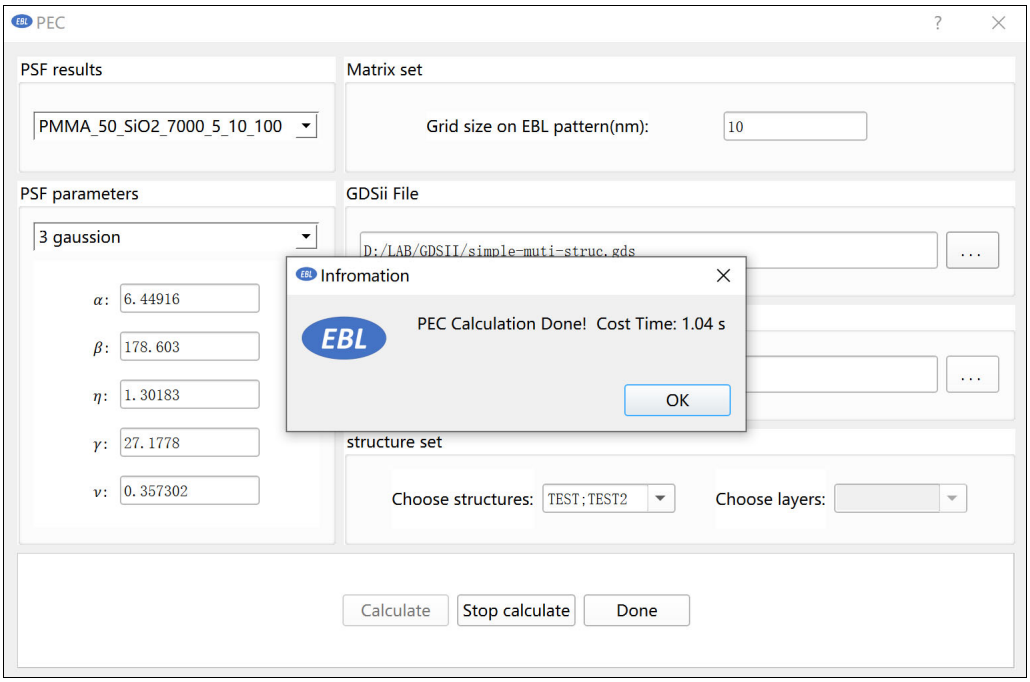


Figure 2.6.4 PEC done

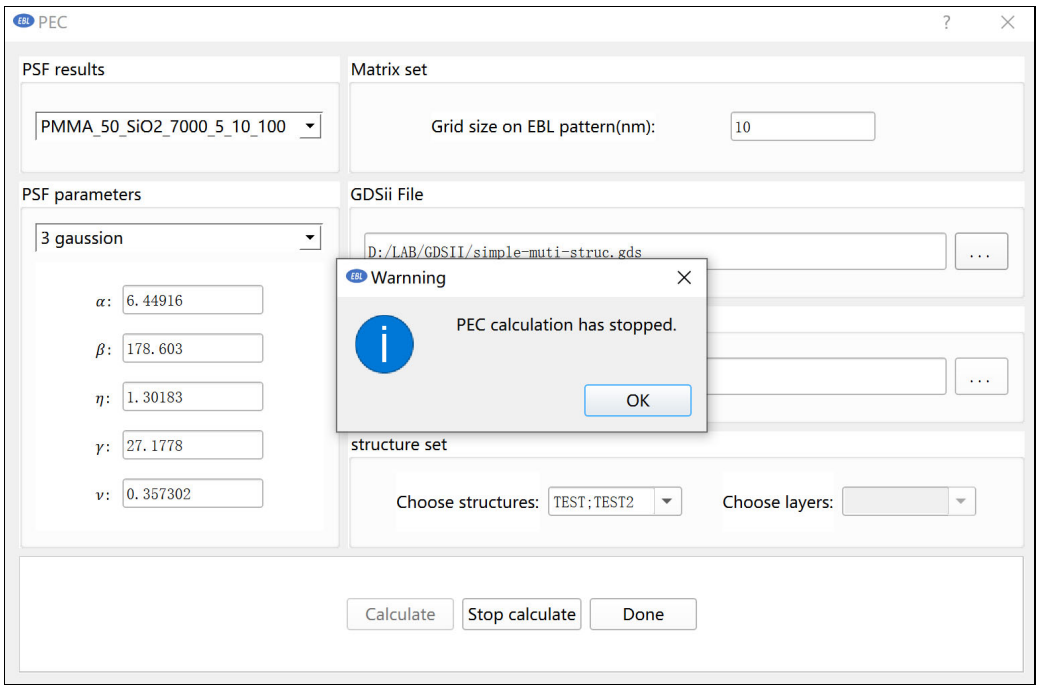


Figure 2.6.5 Stopping PEC

2.7 Edge Placement Error

1) You can click "EPE" on the toolbar at the top of the main interface to start the settings related to calculating the edge placement error (EPE) (Figure 2.7.1).

2) In the "Monte Carlo set" you need to select a MC simulation result for calculating the EPE.

3) In the "fitting function", you need to select the fitting formula. And the Figure 2.7.1 selects the three Gauss function to calculate the EPE.

4) "Matrix set" is set to calculate the minimum size (nm) of the exposure unit, and 5nm is selected in the figure 2.7.1. (Note: If the layout is large, the minimum size of the exposure unit should not be too small. If the size is too small and the computer memory is insufficient, a prompt will be given.)

5) "GDSII File" is used to open the calculated layout in ".gds" format. (Note: The path can only contain English letters and symbols, not Chinese.)

6) You can click "Calculation" (Figure 2.7.1) to calculate the EPE. If the input parameters are incomplete, the calculation cannot be started. There will be a prompt box after the calculation is completed (Figure 2.7.2). After the calculation is completed, you need to click "Done" to close the current window.

7) You can click "Stop Calculation" to stop the current calculation (Figure 2.7.3).

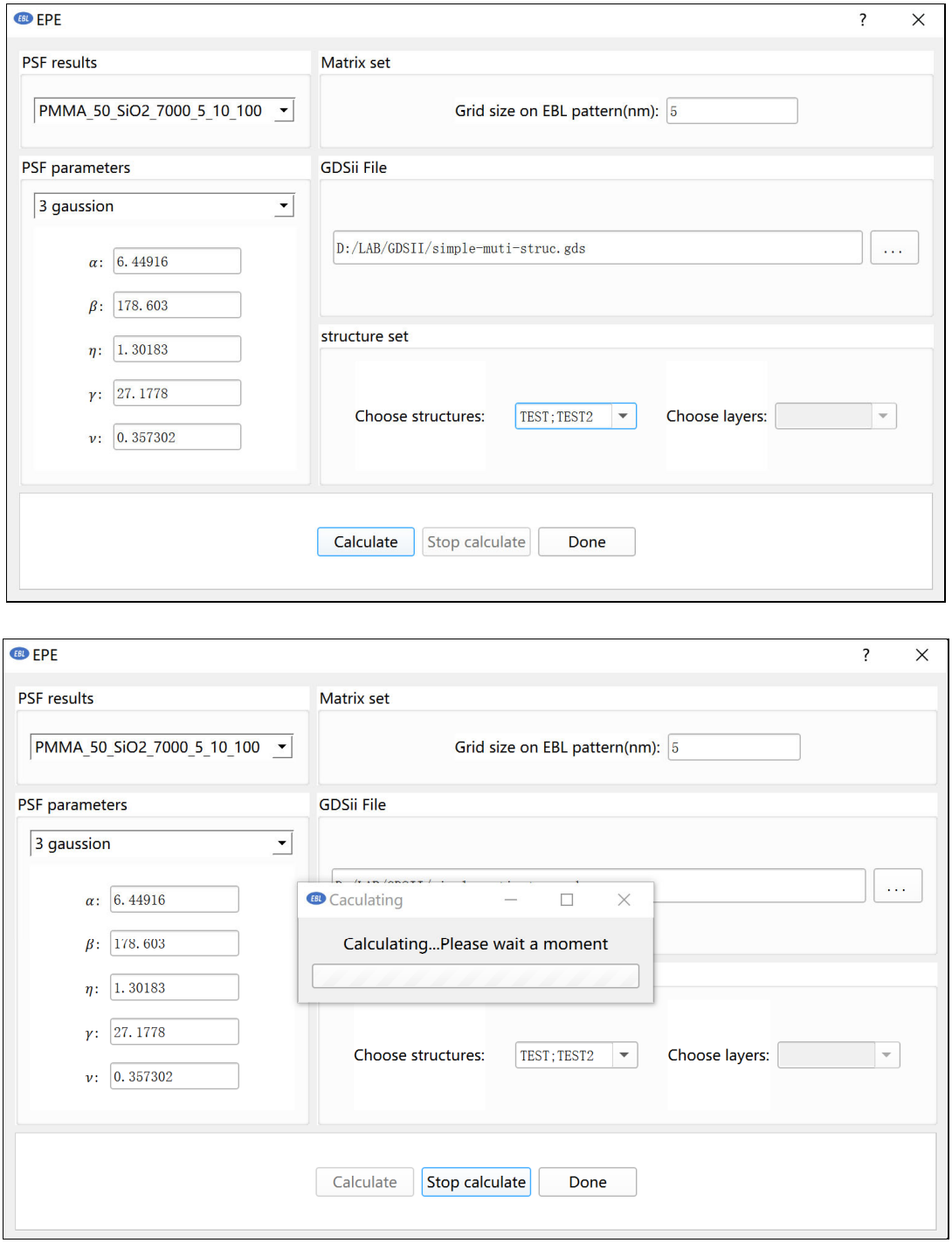


Figure 2.7.1 Calculating EPE

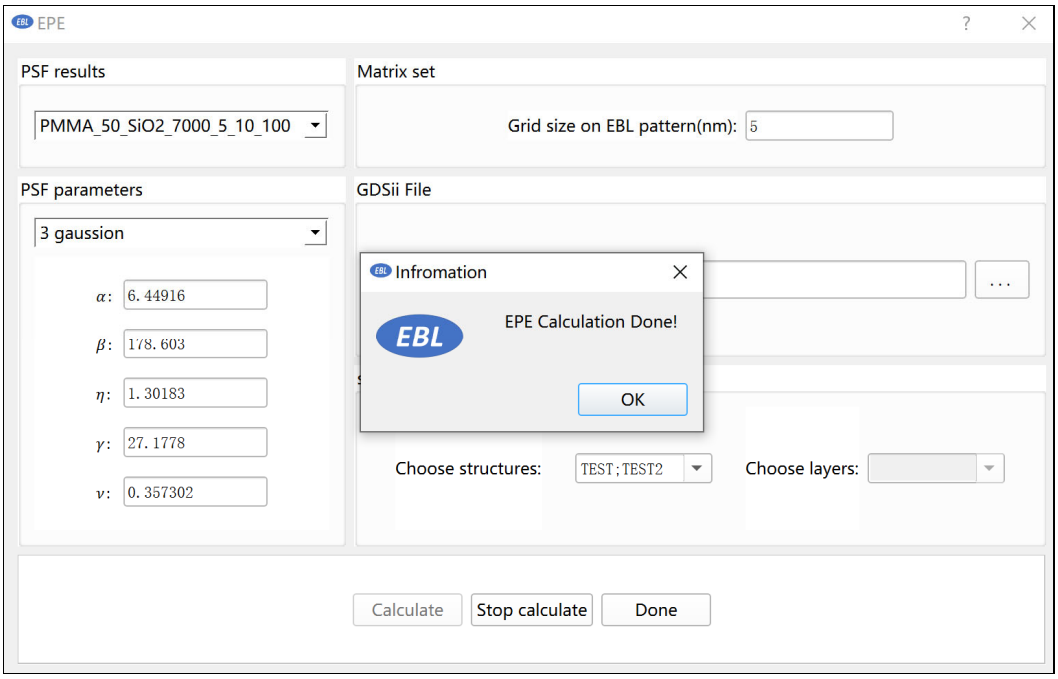


Figure 2.7.2 EPE done

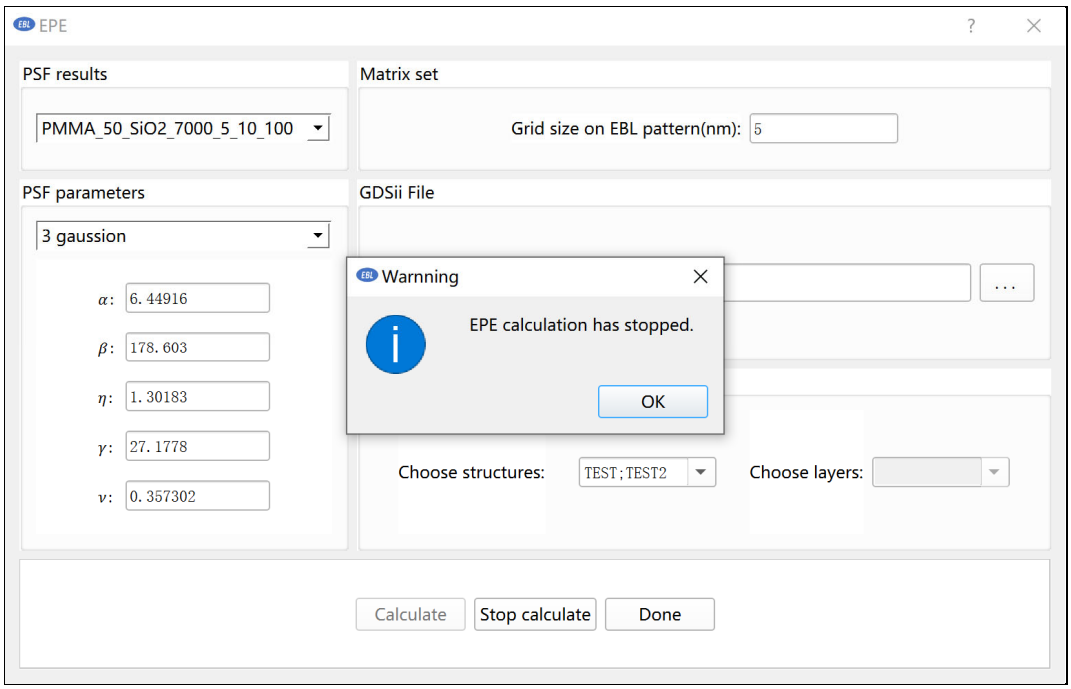


Figure 2.7.3 Stopping calculation of the EPE

8) After the EPE calculation is completed, you can click "OK" in the dialog box in Figure 2.7.3, and a dialog box showing the results of the layout EPE calculation will automatically pop up (Figure 2.7.4). (Note: the layout only contains TEXT pixels can not calculate the edge position error.)

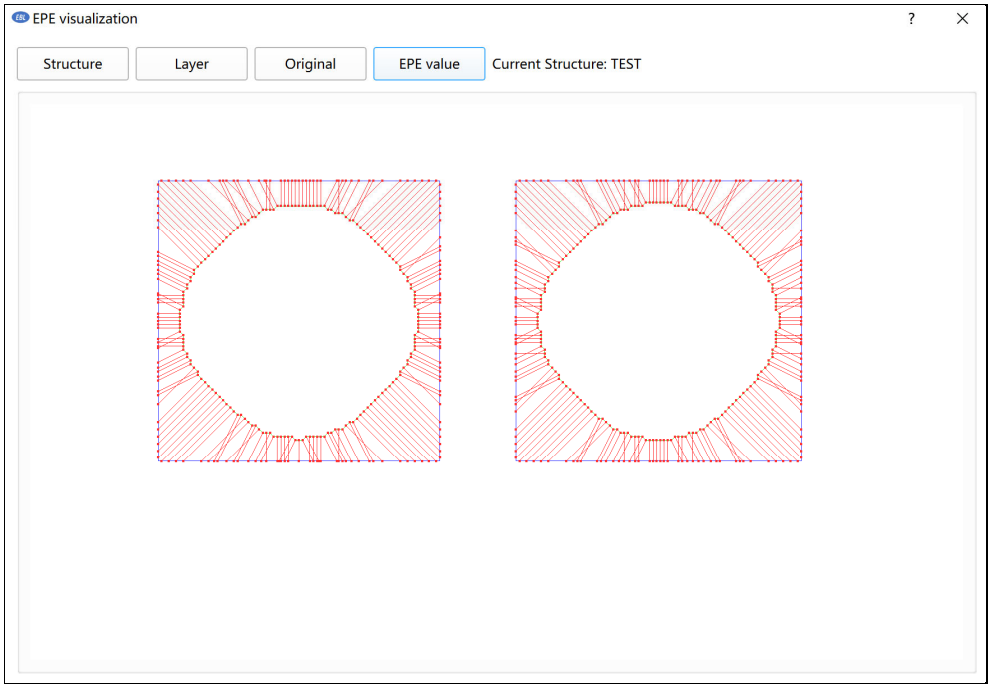


Figure 2.7.4 EPE

9) You can click the "Structure" above to select the EPE of any structure (Figure 2.7.5), and the first structure is displayed by default. It is not possible to select multiple structures for display.

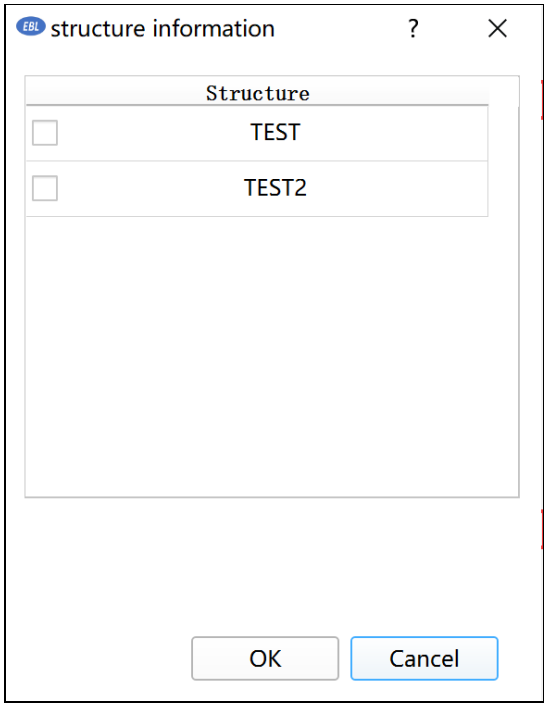


Figure 2.7.5 Select strcuture

10) You can click the "Layer" above to select the EPE of any layer (Figure 2.7.6). All layers are displayed by default. Multiple layers can be selected for display.

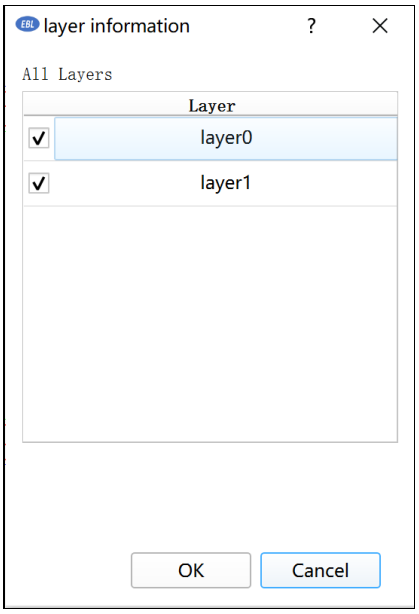


Figure 2.7.6 Select layer

11) You can click the "EPE value" above to display the value of the EPE (Figure 2.7.7).

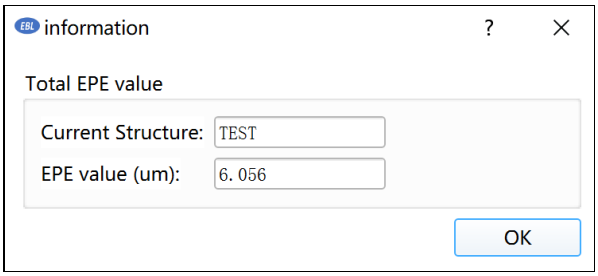


Figure 2.7.7 EPE

12) In the image display area, you can slide the mouse wheel to zoom in and out the image. You can hold down the left mouse button to drag the image and click the right-click of mouse to select a part of the image to partially zoom. You can click "Original" to return the image to the original position. (Figure 2.7.8)

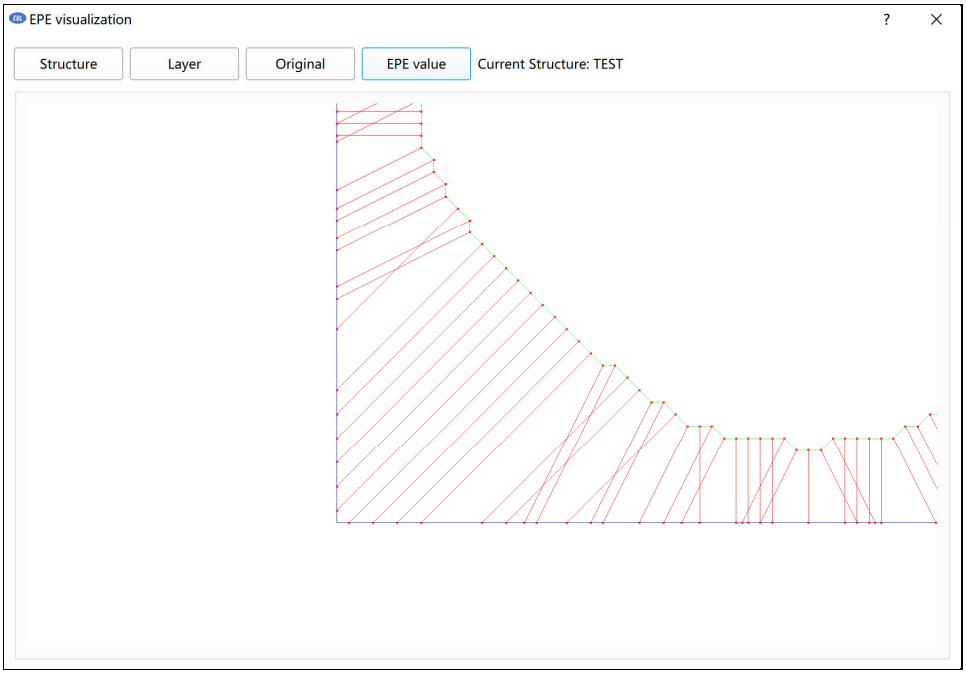


Figure 2.7.8 Partial zoom

2.8 Energy Deposition

1) You can click "Energy Deposition" on the toolbar at the top of the main interface to calculate the energy deposition (Figure 2.8.1).

2) In the "Monte Carlo set" you need to select a MC simulation result for calculating energy deposition.

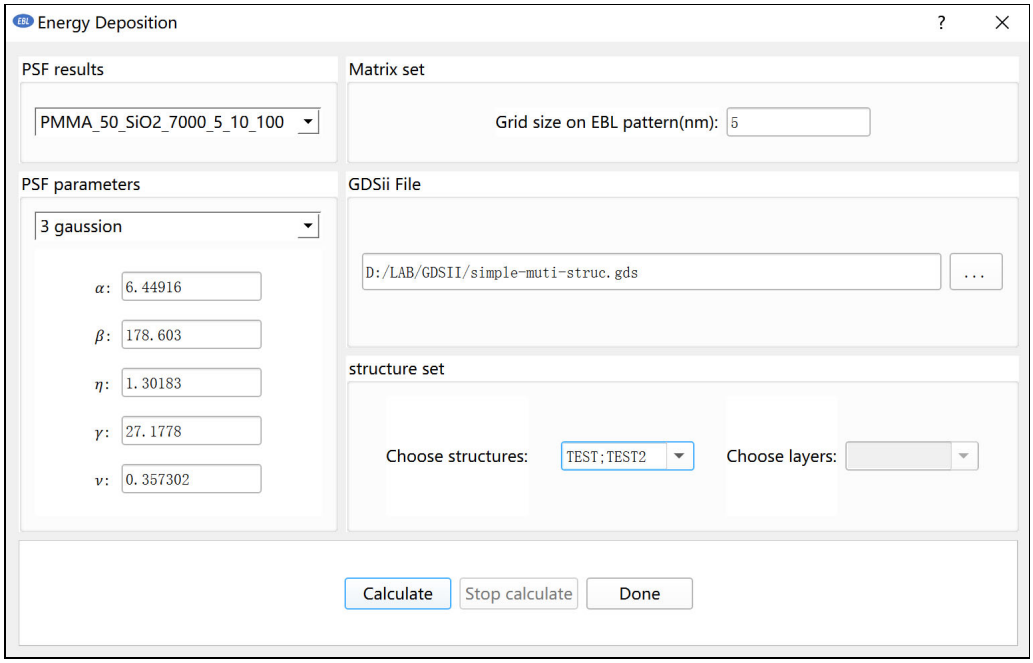
3) In the “fitting function”, you need to select a fitting formula. the three Gauss function for energy deposition calculation is selected in the figure 2.8.1.

4) "Matrix set" is set to calculate the minimum size (nm) of the exposure unit, and 5nm is selected in the figure 2.8.1. (Note: If the layout is large, the minimum size of the exposure unit should not be too small. If the size is too small and the computer memory is insufficient, a prompt will be given.)

5) "GDSII File" is used to open a layout in “.gds” format. (Note: The path can only contain English letters and symbols, not Chinese.)

6) You can click "Calculation" (Figure 2.8.1) to perform energy deposition calculation. If the input parameters are incomplete, the calculation cannot be started. There will

be a prompt box when the calculation is completed (Figure 2.8.2). After the calculation is completed, you need to click "Done" to close the current window.



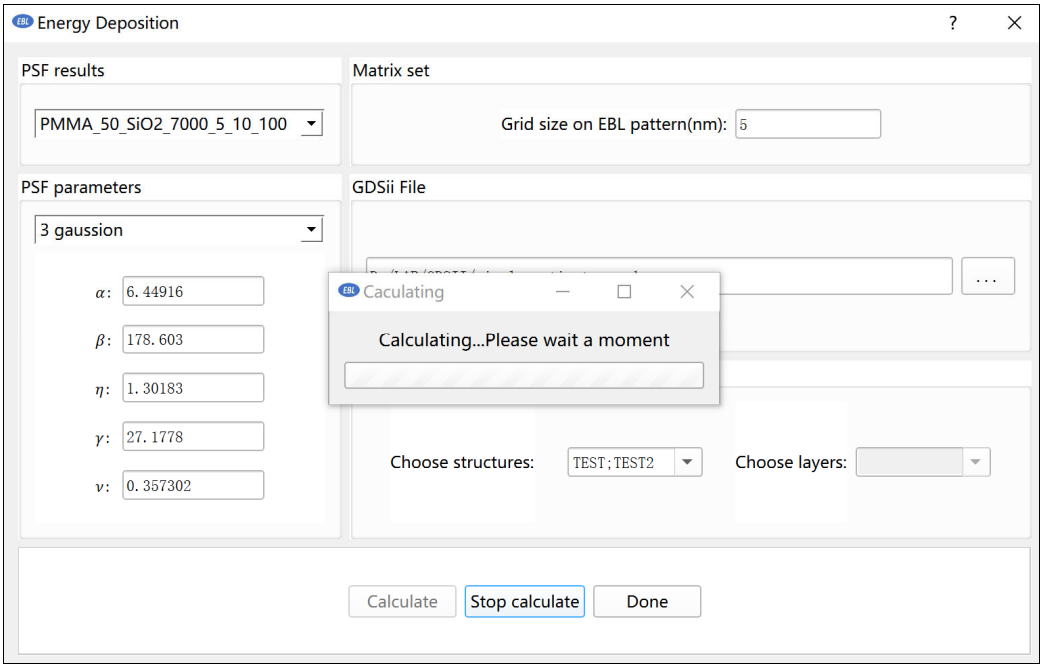


Figure 2.8.1 Energy deposition calculation

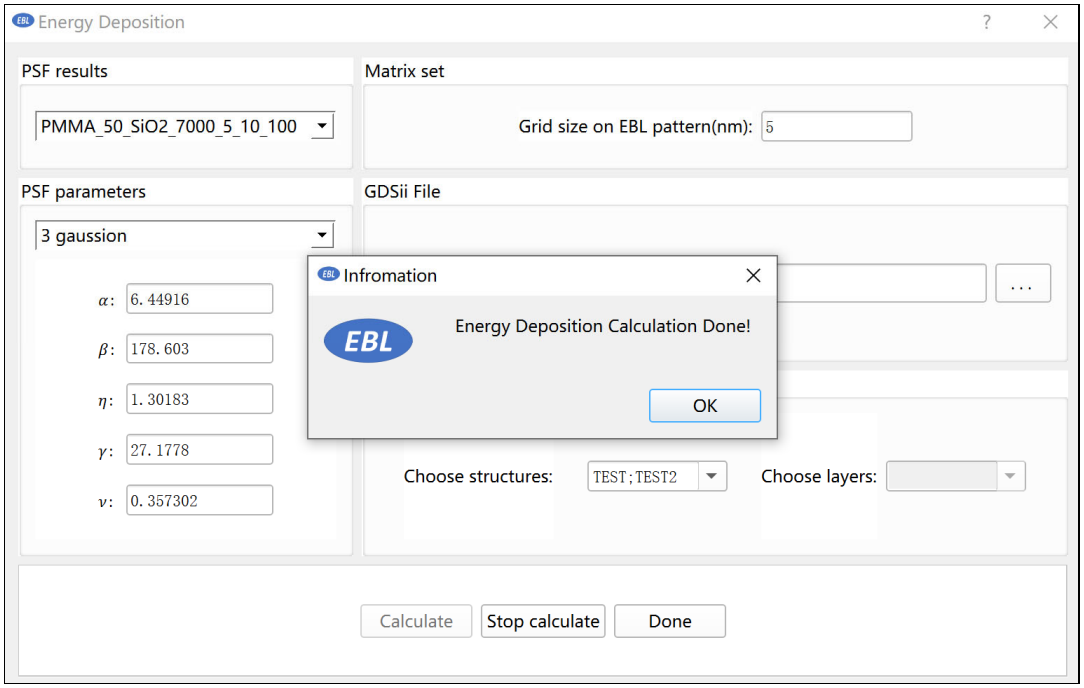


Figure 2.8.2 Energy deposition calculation completed

7) You can click "Stop Calculation" to stop the current calculation (Figure 2.8.3).

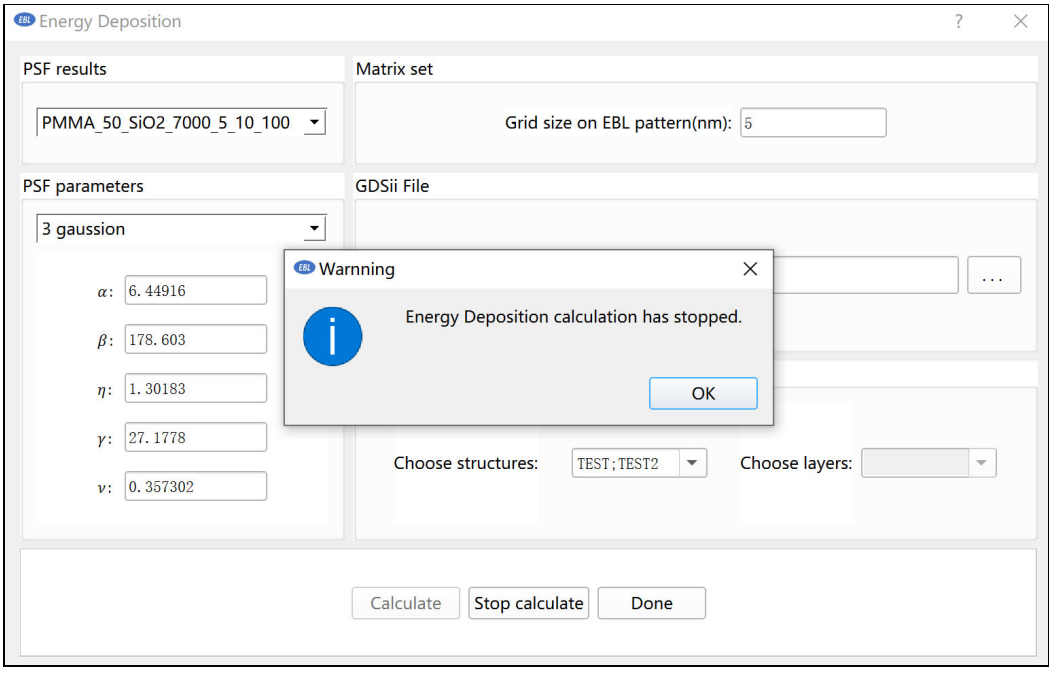


Figure 2.8.3 Stopping energy deposition calculation

8) After the energy deposition calculation is completed, you can click "OK" in the dialog box shown in Figure 2.8.3, and a dialog box showing the results of the layout energy deposition calculation will pop up automatically (Figure 2.8.4). (Note: the

layout with only TEXT pixels cannot be performed energy deposition calculation; the calculation result cannot be displayed when the dose corresponding to the graph is all 0.)

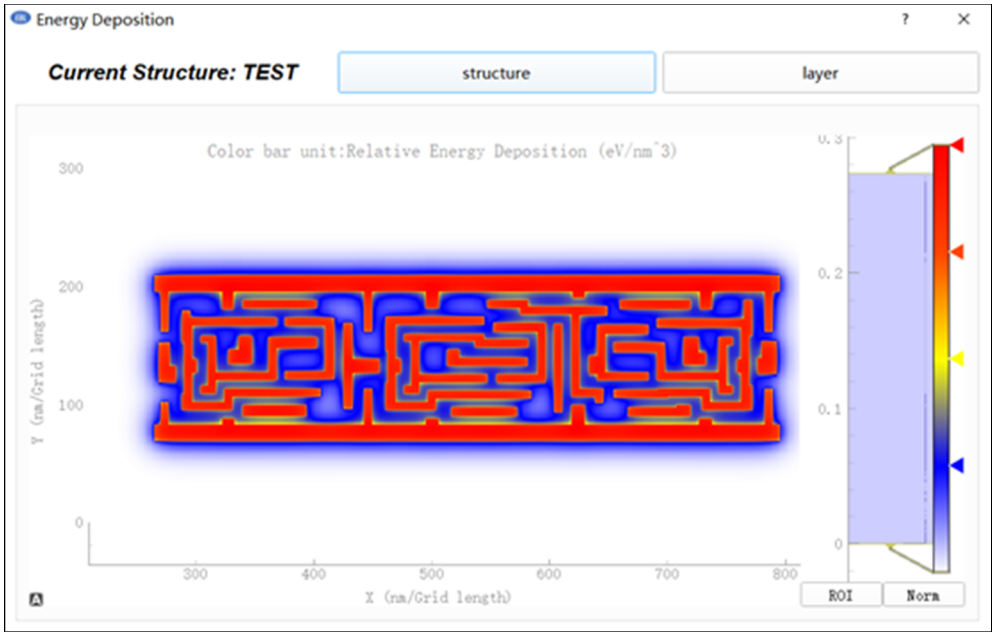


Figure 2.8.4 Energy deposition result

9) You can click “Structure” at the top right to display the energy deposition of any structure (Figure 2.8.5), and the first structure is displayed by default. Multiple structures cannot be selected.

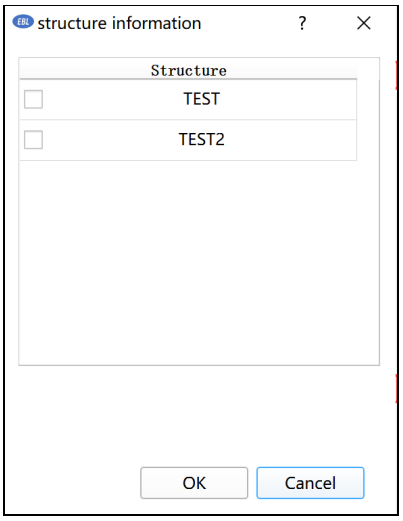


Figure 2.8.5 Select structure

10) You can click “Layer” on the upper right to select the energy deposition of any layer (Figure 2.8.6). The first layer is displayed by default. Only a single layer can be displayed.

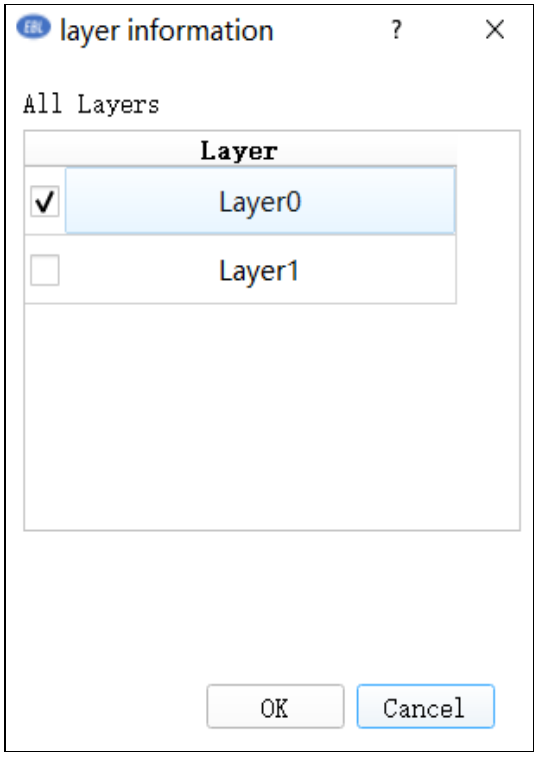


Figure 2.8.6 Select layer

3 GDSII Visualization Module Operation Instructions

3.1 Open File

1) You can click "Open GDSII" in the toolbar at the top of the main interface to open the GDSII visualization module interface (Figure 3.1.1) and click "Open" to open a GDSII file (Figure 3.1.2). If the GDSII file is large and takes a long time to load, there will be a prompt at the bottom right of the software interface (Figure 3.1.3).

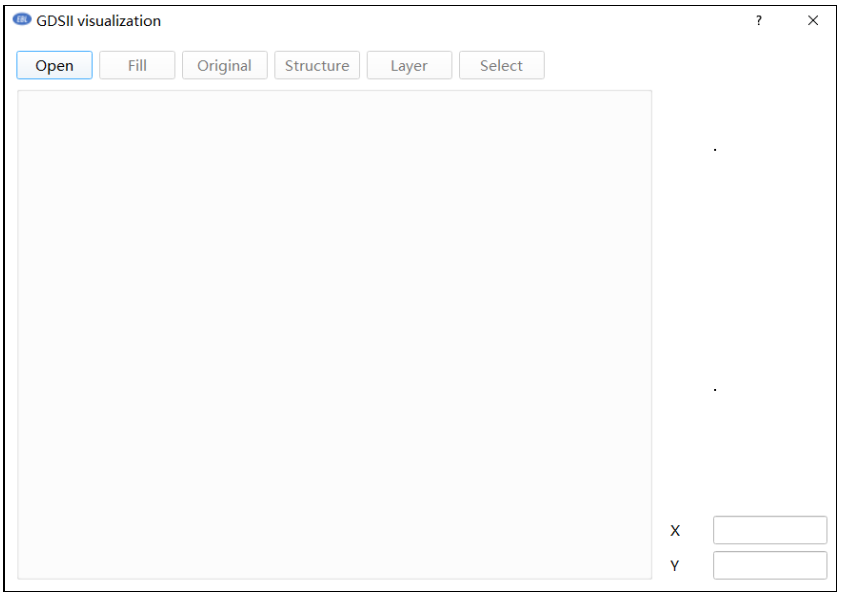


Figure 3.1.1 Main interface

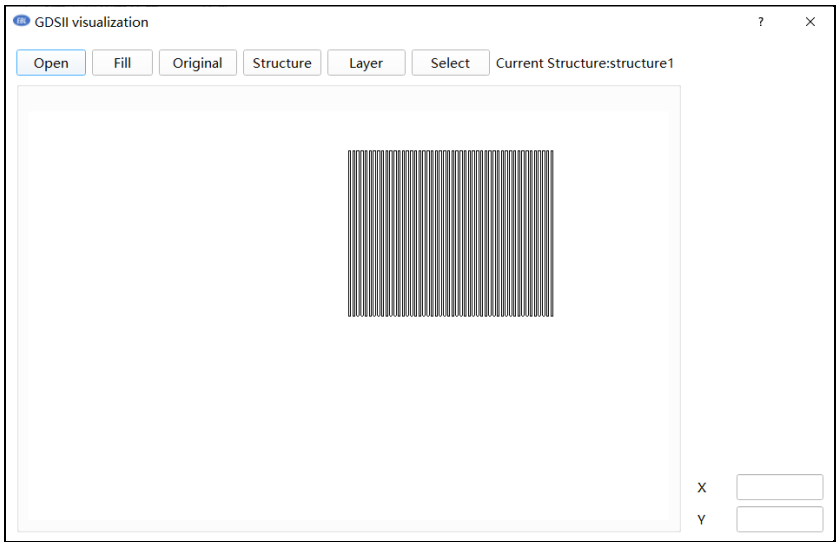


Figure 3.1.2 Open file

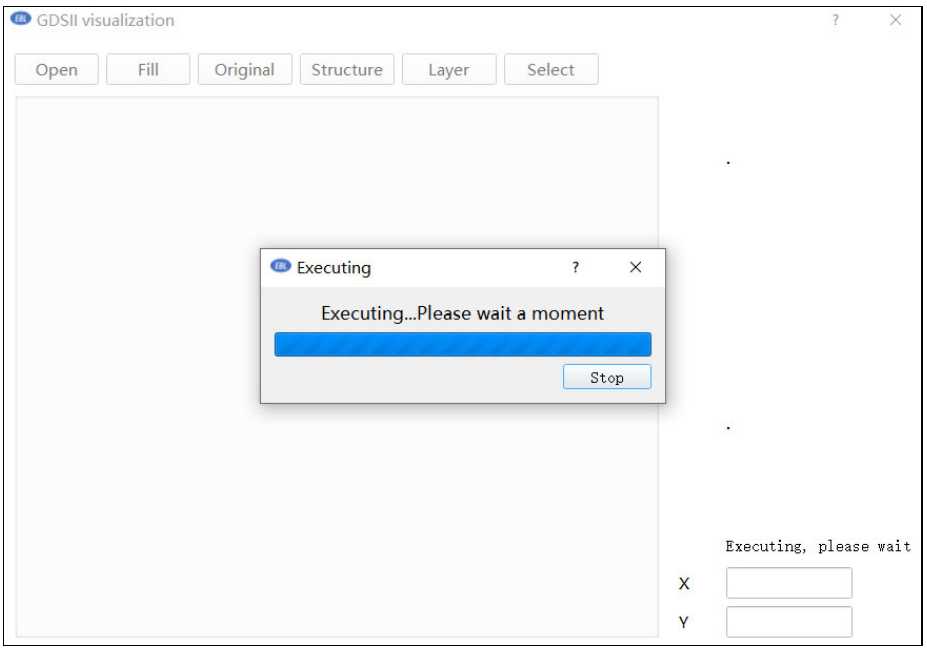


Figure 3.1.3 Loading large file

3.2 Select Structure

1) "Structure" is used to display structure at the top of the interface (Figure 3.2.1). Ticking before a structure you want to view, and click "OK" to view it. (Note: Only one structure can be selected.)

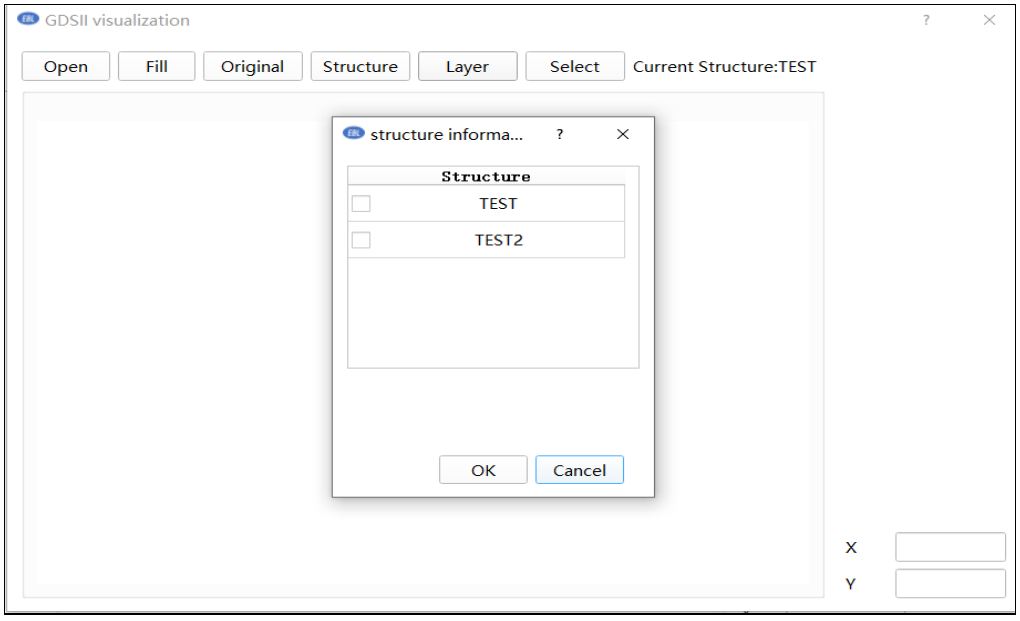


Figure 3.2.1 Select structure

3.3 Select Layer

1) "Layer" is used to display layer at the top of the interface (Figure 3.3.1). Ticking before a layer you want to view, and click "OK" to view it. (Note: Multiple layers can be selected for display)

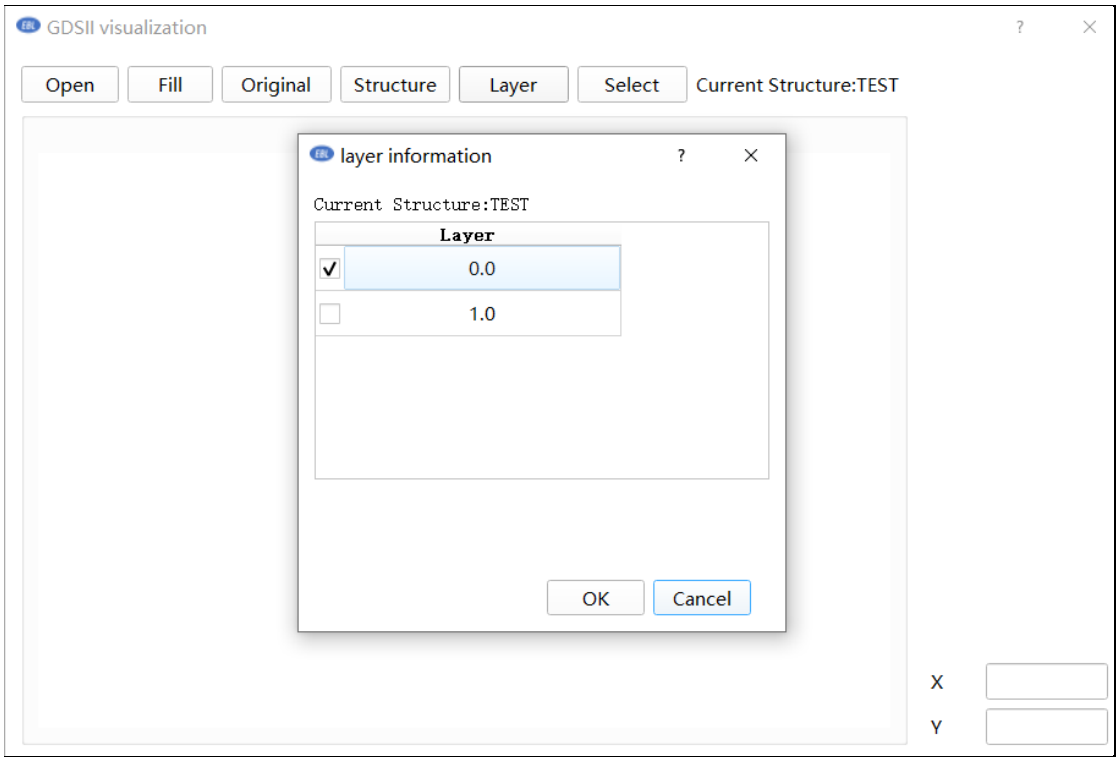


Figure 3.3.1 Select layer

3.4 Display Dose

1) You can click "Fill" at the top of the interface (Figure 3.4.1) to view the layout after displaying the filling dose. The maximum and minimum values of the color bar on the right represent the maximum and minimum doses of the current layout. If the layout has only one dose, the color bar on the right has only one value and the color is red. (Note: Only fill once.)

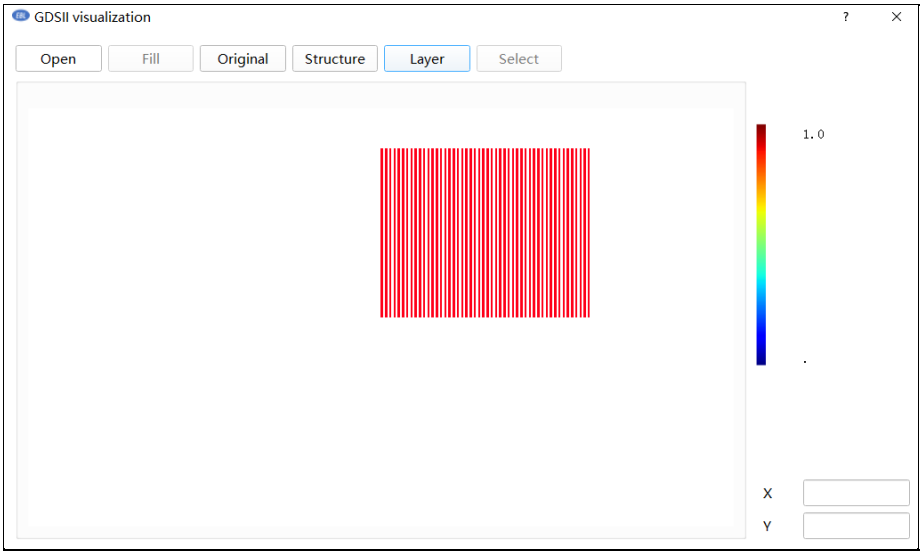


Figure 3.4.1 Display dose

3.5 Position Move

1) You can click the left mouse button and drag to move the layout position (Figure 3.5.1 and Figure 3.1.2).

2) You can click "Original" (Figure 3.5.2) to move the layout to the initial position. The coordinates (0, 0) move to the center of the screen.

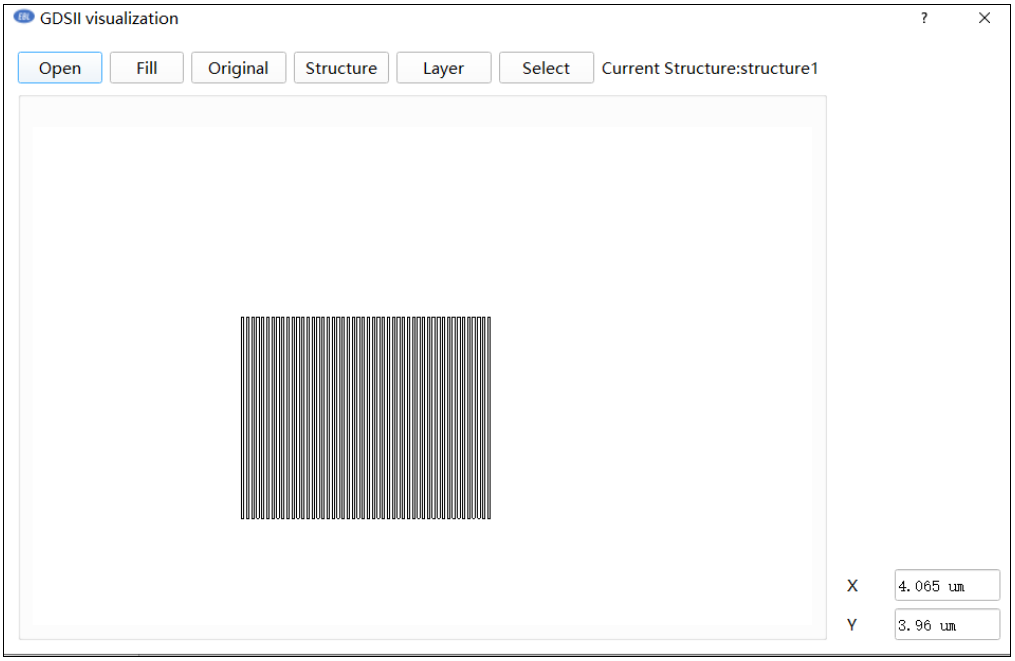


Figure 3.5.1 Layout move

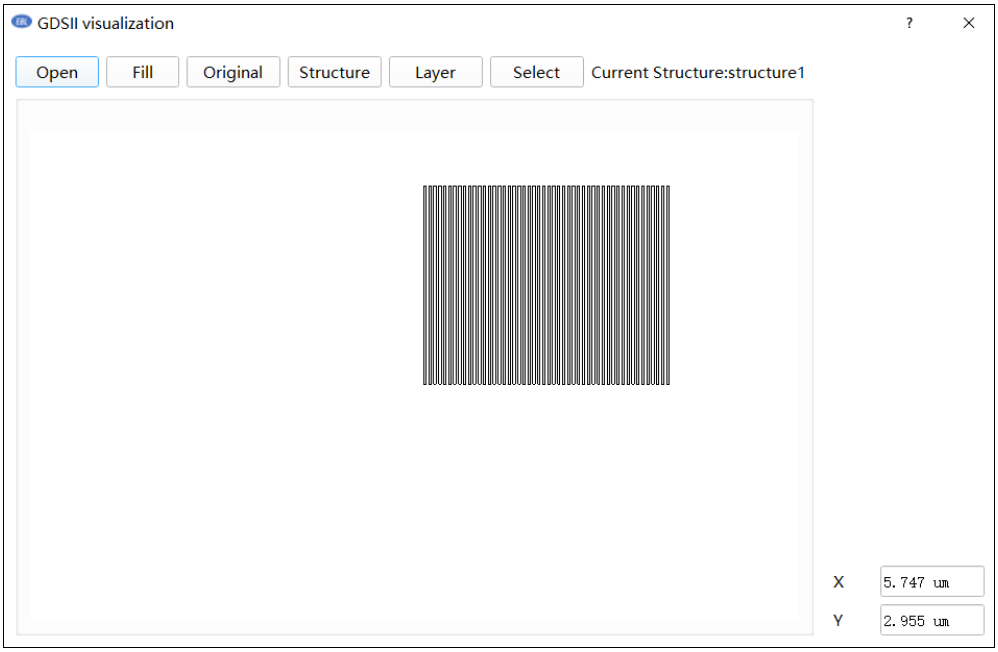


Figure 3.5.2 Click “Original”

3.6 Scaling

1) You can slide the mouse wheel upwards to enlarge the layout, as shown in the comparison between Figure 3.6.1 and Figure 3.1.2, and slide the mouse wheel downward to reduce the layout, as shown in the comparison between Figure 3.6.2 and Figure 3.1.2.

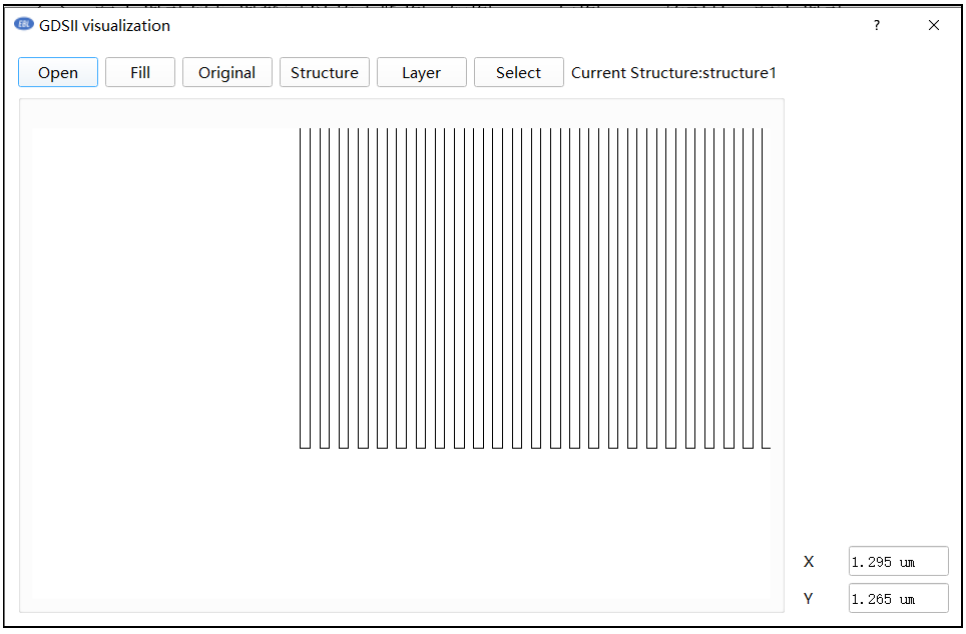


Figure 3.6.1 Enlarge the layout

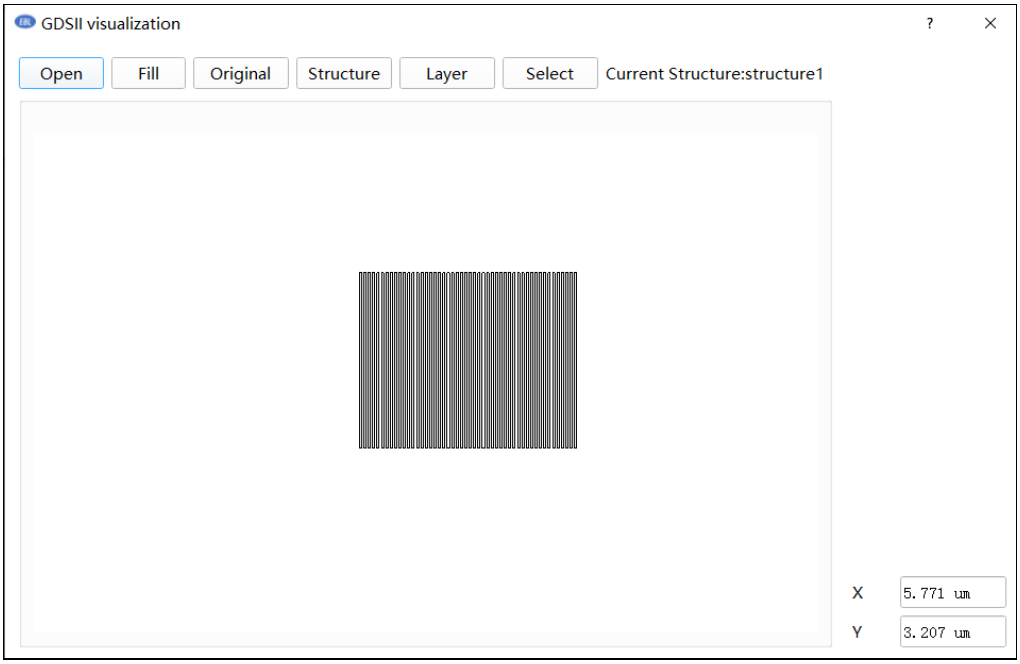


Figure 3.6.2 Reduce the layout

2) You can click the right mouse button to select the partial layout, and the selected part can be enlarged. As shown in Figure 3.6.3 and Figure 3.1.2.

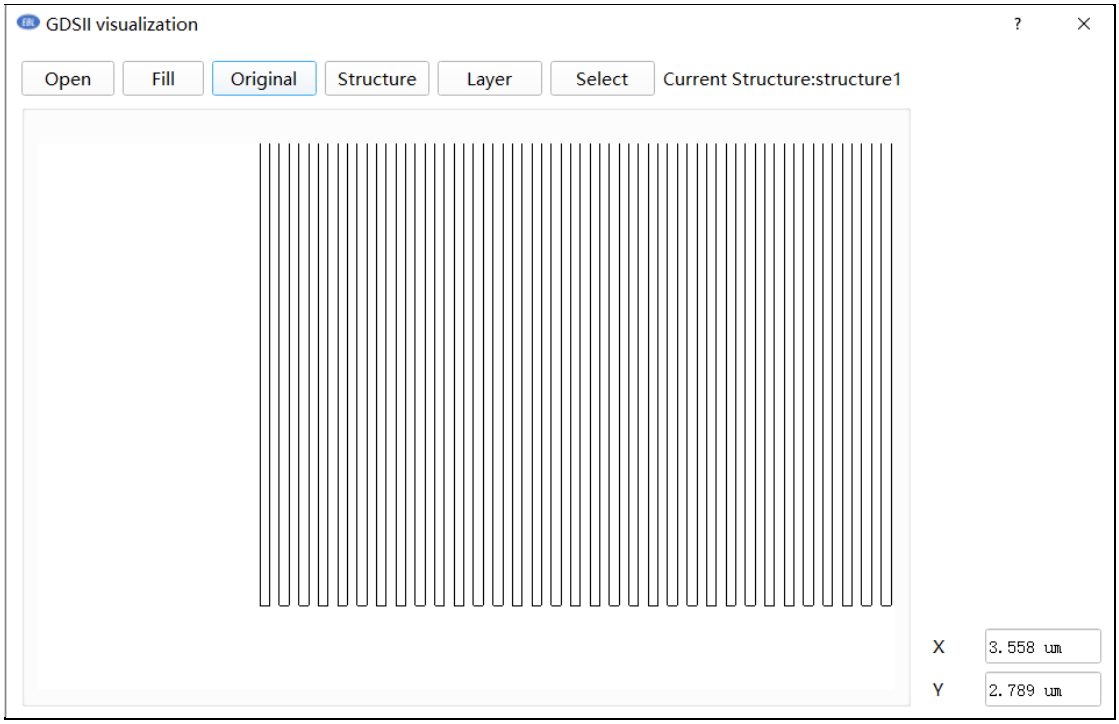


Figure 3.6.3 Partial zoom

3.7 Show Information

1) You can click any point in the layout with the mouse, and the corresponding layout coordinates can be displayed in the lower right corner of the main interface (Figure 3.7.1 bottom right). The main interface can display the structure name currently displayed on the layout (Figure 3.7.1 top).

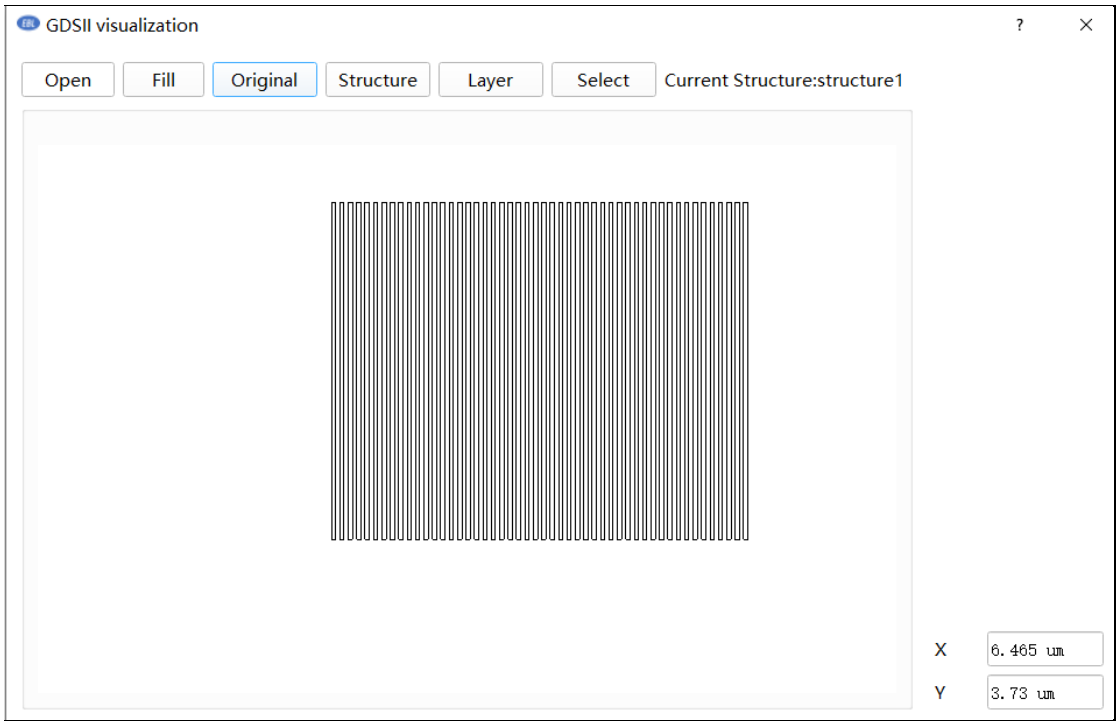


Figure 3.7.1 Display layout coordinates

2) You need to double-click a certain point inside the graph in the layout to display the detailed information of the graph. The information cannot be changed. Figure 3.7.2-7 shows the information display interface of different pixels.

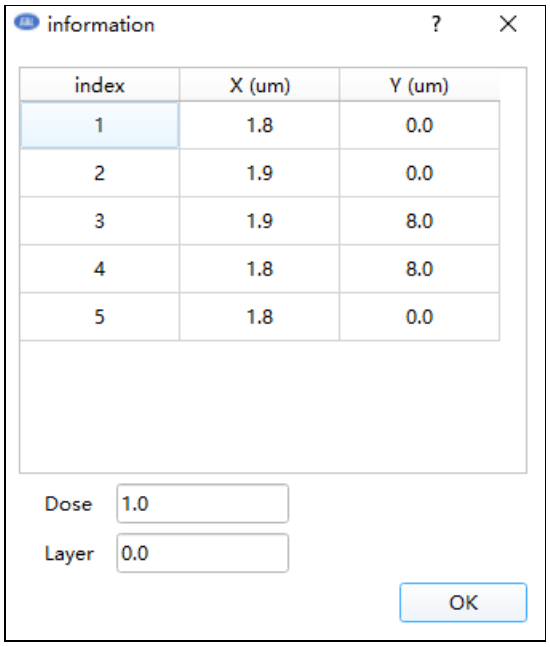


Figure 3.7.2 Boundary pixel details

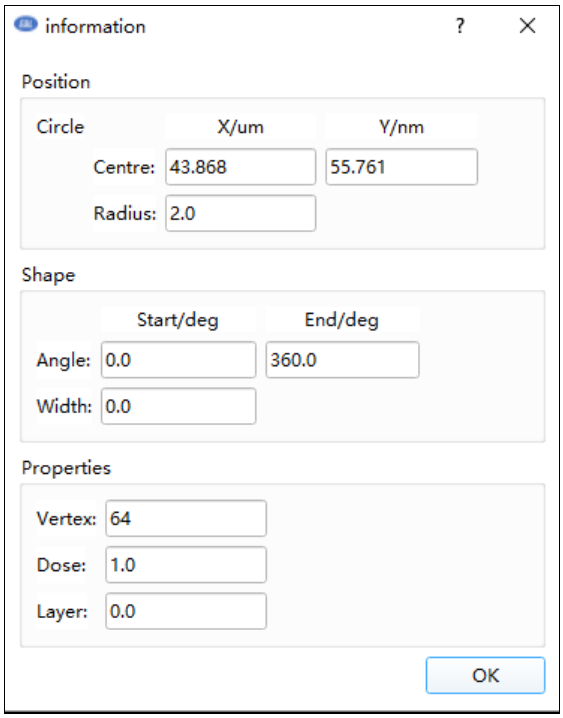


Figure 3.7.3 Circle pixel details

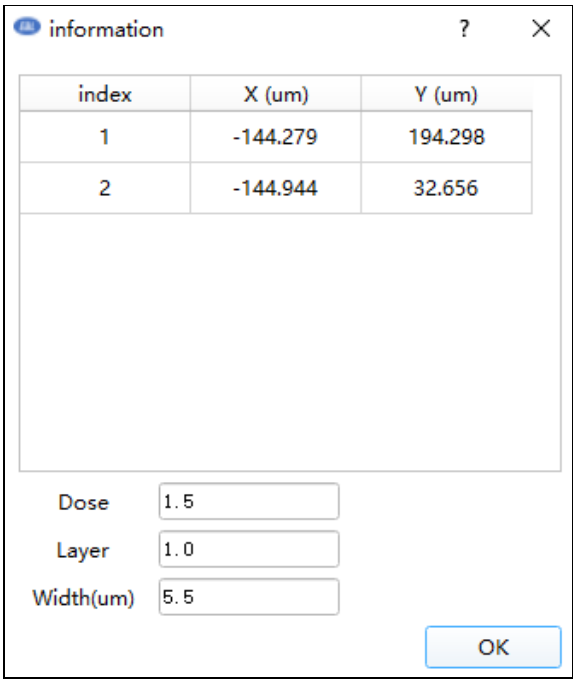


Figure 3.7.4 Path pixel details

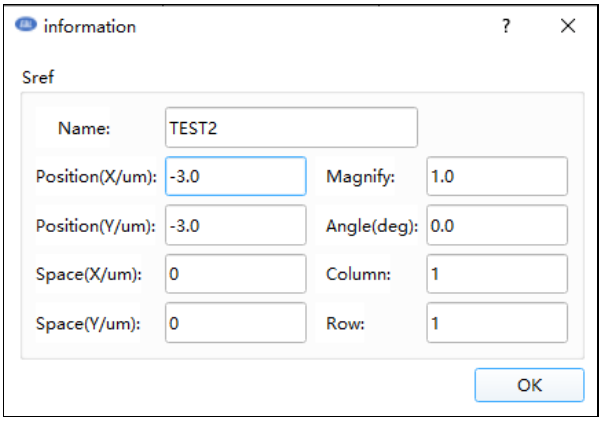


Figure 3.7.5 Sref pixel details

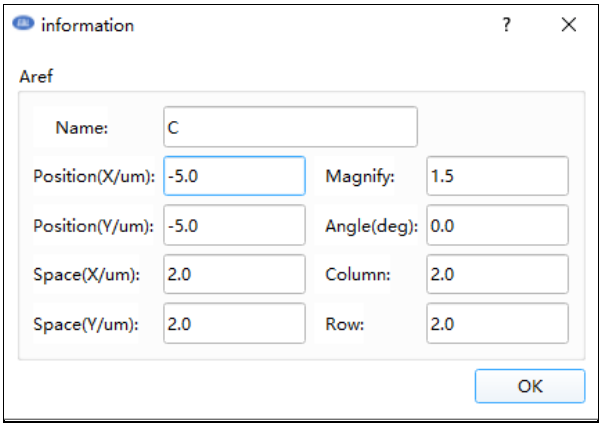


Figure 3.7.6 Aref pixel details

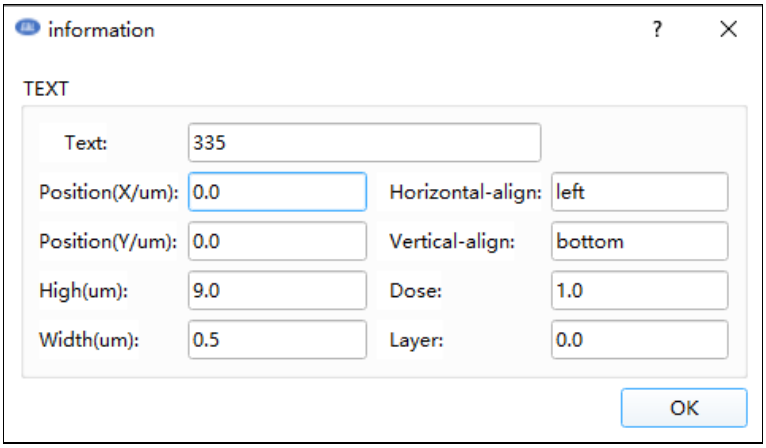


Figure 3.7.7 Text pixel details

3.8 Frame Selection

1) You need to open a gds file and click “Select” (Figure 3.8.1).

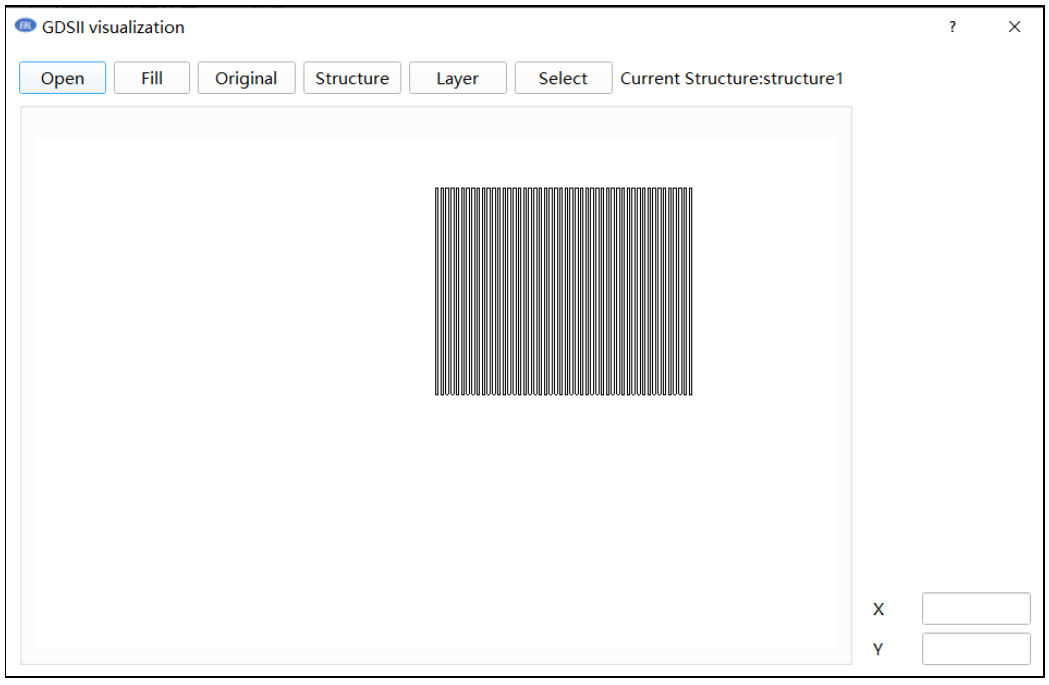


Figure 3.8.1 Open gds file

2) The left mouse button is used to frame selection (You need to press and hold the left button to plan the path and release the mouse to form a frame selection area.). After the selection is completed, a dialog box will pop up to determine whether to frame the area (Figure 3.8.2).

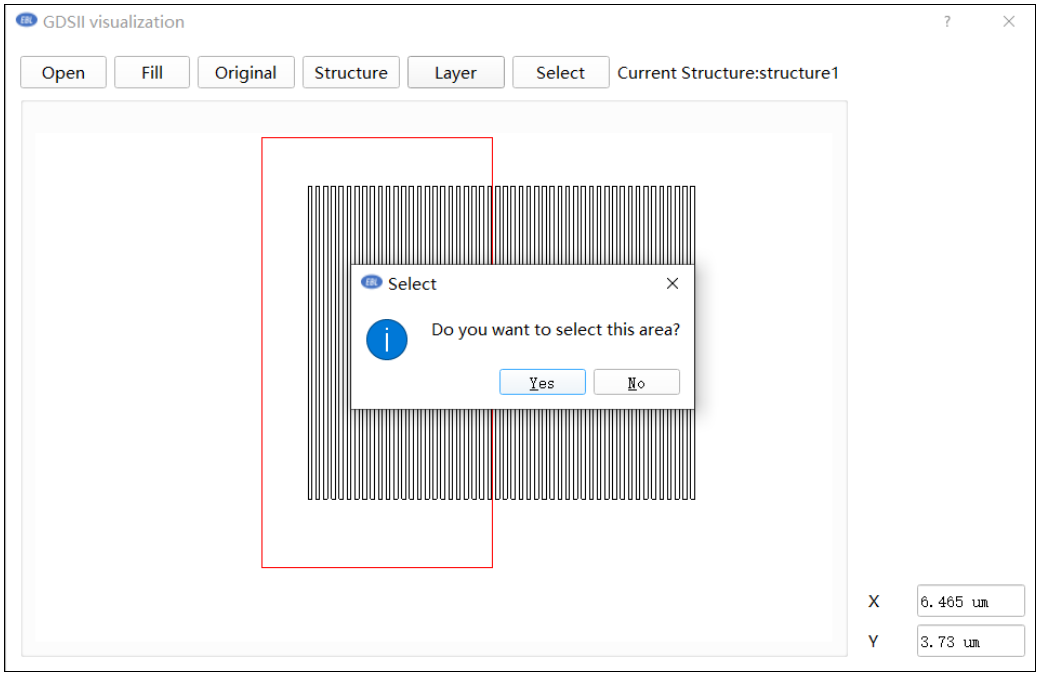


Figure 3.8.2 Show frame selection dialog

3) It will automatically generate the GDSII file you selected in the box if you click yes. And the path is in the folder where your software is located (Figure 3.8.3). At the same time, you can select PEC, EPE, energy and other operations (Figure 3.8.4).

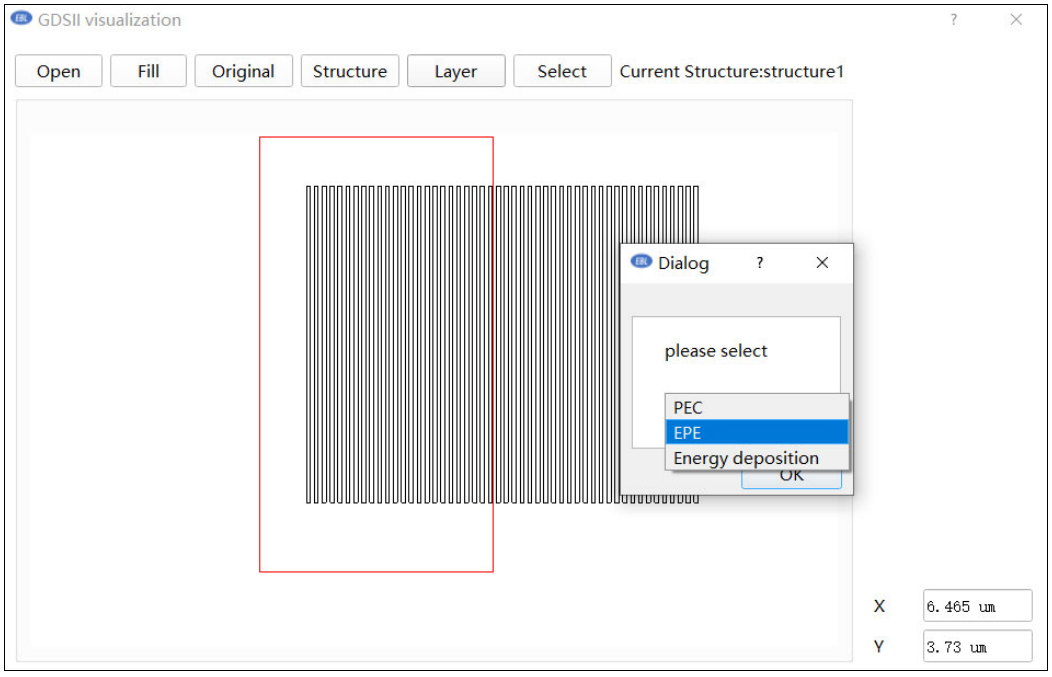


Figure 3.8.3 Operation after box selection

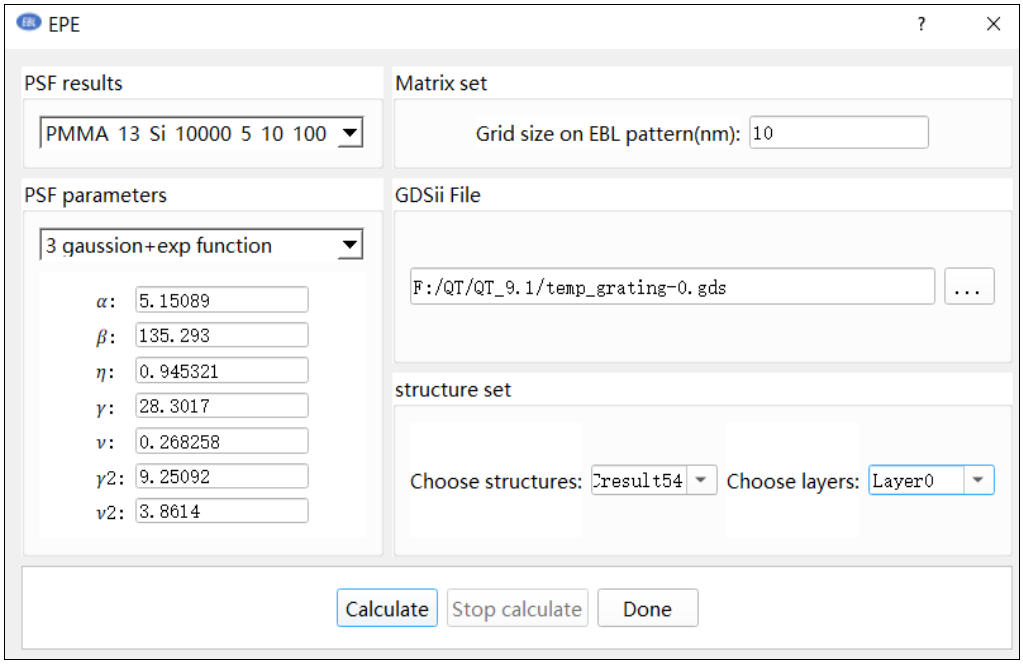


Figure 3.8.4 Select EPE module calculation

4) After the calculation is completed, the calculation result of the part you have selected will be displayed (Figure 3.8.5).

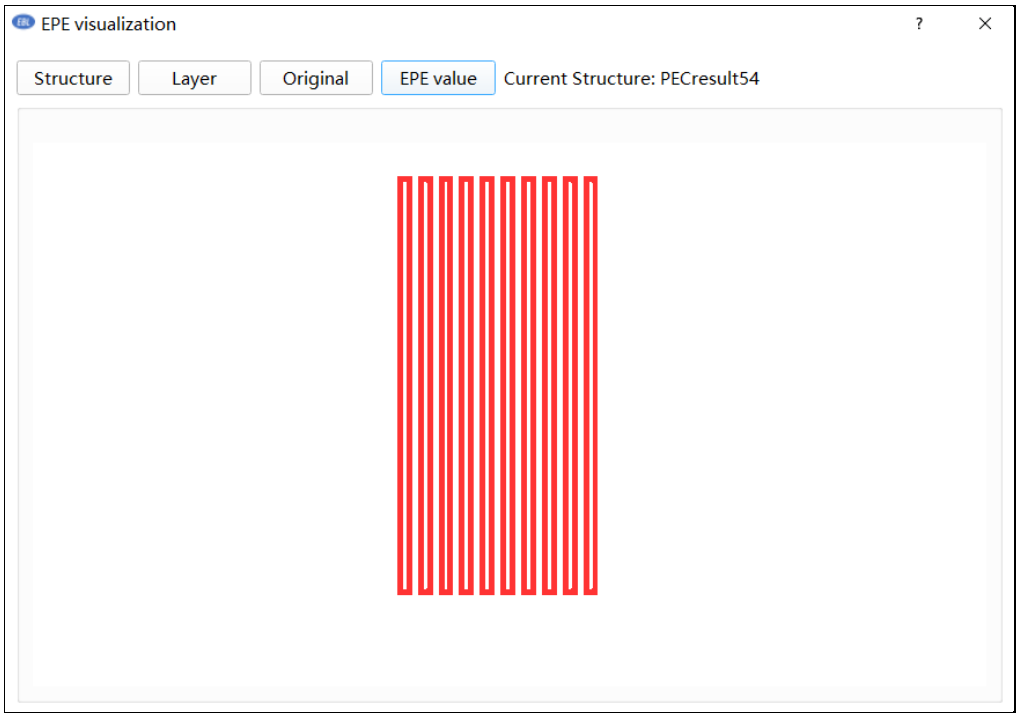


Figure 3.8.5 EPE result of gds file after box selection

Reference

[1] C. Hou, W. Yao, W. Liu, Y. Chen, H. Duan and J. Liu, "Ultrafast and Accurate Proximity Effect Correction of Large-Scale Electron Beam Lithography based on FMM and SaaS," *2020 International Workshop on Advanced Patterning Solutions (IWAPS)*, 2020, pp. 1-3, doi: 10.1109/IWAPS51164.2020.9286816.

[2] Liu Jie; Yao Wenze; Hou Chengyang; Duan HuiGao; Chen Yiqin; Zhou Jian . “ A correction method of electron beam proximity effect based on Neural Network ” . CN. Patent 111538213 B April 27 , 2020

[3] Liu Jie; Yao Wenze; Hou Chengyang; Duan HuiGao; Chen Yiqin; Zhou Jian . “ High precision proximity effect fast correction method for large-scale electron beam exposure layout ”. CN. Patent 111965936 A November 20, 2020

[4] Liu Jie; Yao Wenze; Xu Hongcheng ;Zhao Haojie ; Liu Wei ; Hou Chengyang. ” A proximity effect correction method for electron beam exposure based on two-dimensional fast Fourier transform ” CN. Patent 112987514 A June 18, 2021

[5] Hunan University. “ Simulation and correction software of electron beam exposure based on SaaS mode v1.0 ” June 20, 2020

[6] Hunan University. “ Simulation and correction software of electron beam exposure based on SaaS mode v1.0” January 10, 2020

[7] Hunan University. “ EDA software for simulation calculation of electron beam lithography v1.0 ” May 20,2021

[8] Hunan University. ” GDSII file visualization software v1.0 ” May 21, 2021