HORIBA
Scientific

# DeltaPsi2™ Software

## Reference Manual

**Part Number: 1131087091**

## COPYRIGHT



## NOTICE TO THE USER

This manual should not be construed as any representation or warranty with respect to the unit named herein. Occasionally, changes or variations exist in the unit that are not reflected in the manual. Generally, should such changes or variations exist and affect the product significantly, a release note would accompany the manual. In such a case, be sure to read the release note before using the product.
Trademarks: DeltaPsi is a registered trademark of HORIBA Jobin Yvon, Windows NT/98/XP/7/8/10 are registered trademarks of Microsoft Corporation.


**NP/DeltaPsi2R.fm/264202 - 05/20/2016**

**HORIBA Jobin Yvon S.A.S.**
**Headquarters**
16-18, rue du Canal
91165 LONGJUMEAU CEDEX (France)
Tel: +33 (0)1 69 74 72 00
Fax: +33 (0)1 69 09 07 21
Internet site: http://www.horiba.com/scientific
email: info-sci.fr@horiba.com

**HORIBA Jobin Yvon S.A.S.**
**Palaiseau**
Avenue de la Vauve
Passage Jobin Yvon
CS 45002
91120 PALAISEAU (France)
Tel: +33 (0)1 69 74 72 00
Fax: +33 (0) 69 31 32 20

**HORIBA Instruments Inc.**
3880 Park Avenue, EDISON - New Jersey 08820 (USA)
Tel: +1 (732) 494 86 60 - Fax: +1 (732) 549 51 25

**HORIBA Jobin Yvon GmbH**
Neuhofstrasse 9 - D-64625 BENSHEIM (Germany)
Tel.+49 89 62 51 84 750 - Fax +49 89 62 51 84 7520

**HORIBA UK Ltd.**
2 Dalston Gardens - STANMORE
Middlesex HA7 1BQ (Great Britain)
Tel: +44 (0) 20 8204 8142 - Fax: +44 (0) 20 8204 6142

**HORIBA Italia Srl**
Via Cesare Pavese 19 - 20090 OPERA (Milano) (Italy)
Tel: +39 (2) 57 60 30 50 - Fax +39 (2) 57 60 08 76

**HORIBA Trading Co. Ltd. (China)**
Unit D 1F, Bldg A, Srynnex International Park, No. 1068 West Tianshan Road, SHANGHAI 200335
Tel: +86 (0)21 6289 6060 - Fax: +86 (0)21 6289 5553

**HORIBA Ltd. Japan**
Tokyo Branch Office, 2-6, KandaAwaji-cho, Chiyoda-ku, TOKYO 101-0063, Japan
Tel: +81 (0) 3 6206 4721 - Fax: +81 (0)3 6206 4730

**HORIBA Korea Ltd.**
10, Dogok-ro 6 gil, Gangnam-gu, SEOUL, 135-860, Korea, Tel : +82 (0)2 753 7911, Fax : +82 (0)2 756 4972

05/20/2016

# HORIBA Scientific - Technology

With its long experience in research and manufacture of scientific instruments (since 1819), the HORIBA Scientific equipment uses state-of-the-art technology for optical, mechanical, electronic and computer management.

The result is unequaled comfort and flexibility of use, with less human error, for the kind of precision and reproducibility of measurements that are required in laboratory and production work today.

Last but not least, HORIBA Scientific instruments are designed and manufactured with complete user safety in mind.

**Our Internet Site:** **http://www.horiba.com/scientific**

**QR Code:**

**Service Team mailbox: services.jyfr@horiba.com**
**To increase the processing speed of a request, please insert at the beginning of the email subject line, the following word: [TFD].**

**For example -** Subject: [TFD] How to initialize the instrument

**Phone number: +33 (0)1 69 74 72 02**

05/20/2016

# Warranty and Responsibility

HORIBA Scientific warrants that the Product shall be free from defects in material and workmanship and agrees to repair or replace free of charge, at option of HORIBA Scientific, any malfunctioned or damaged Product attributable to responsibility of HORIBA Scientific for a period of one (1) year from the delivery unless otherwise agreed with a written agreement. In any one of the following cases, none of the warranties set forth herein shall be extended;

- Any malfunction or damage attributable to improper operation
- Any malfunction attributable to repair or modification by any person not authorized by HORIBA Scientific
- Any malfunction or damage attributable to the use in an environment not specified in this manual
- Any malfunction or damage attributable to violation of the instructions in this manual or operations in the manner not specified in this manual
- Any malfunction or damage attributable to any cause or causes beyond the reasonable control of HORIBA Scientific such as natural disasters
- Any deterioration in appearance attributable to corrosion, rust, and so on
- Replacement of consumables

HORIBA SCIENTIFIC SHALL NOT BE LIABLE FOR ANY DAMAGES RESULTING FROM ANY MALFUNCTIONS OF THE PRODUCT, ANY ERASURE OF DATA, OR ANY OTHER USES OF THE PRODUCT.

05/20/2016

# Contents

*HORIBA Scientific - Technology* . . . . . 3
*Warranty and Responsibility* . . . . . 4
*Contents* . . . . . 5

***DeltaPsi2™ Software Reference Manual*** . . . . . ***10***

**Getting Started** . . . . . **11**

Running DeltaPsi2 Software . . . . . 11
Configuring the Working Environment . . . . . 11
TreeView data files environment . . . . . 12
How to create a new user Profile . . . . . 12
How to create a new folder . . . . . 13
Software Options . . . . . 14
Application Library Folders . . . . . 14
File name formats . . . . . 15
File Explorer . . . . . 16
Tool Bar . . . . . 16
Progress Screens . . . . . 17
Observables Setup . . . . . 17
Modeling Output . . . . . 18
Error values for Fit . . . . . 19
Recipe Options . . . . . 19
License keys . . . . . 20
Results deletion . . . . . 21
General Setup . . . . . 21

**Main Working Interface** . . . . . **22**

Accessing . . . . . 22
General Graphics Interface . . . . . 23
TreeView Description . . . . . 28
TreeView Options . . . . . 29
Icon Palette area . . . . . 32

**Graphics Screen** . . . . . **33**

Standard Display Mode . . . . . 33
Data Manipulation feature . . . . . 39
Navigation Display Mode . . . . . 40
Text Display Mode . . . . . 40

**Acquisition** . . . . . **42**

Accessing . . . . . 42

Views for UVISEL Ellipsometer . . . . . 43
Top Bar Buttons . . . . . 44
Real Time Graphics Screen . . . . . 46
Acquisition parameters setup . . . . . 46
Ellipsometric Data . . . . . 50
Motors control panel . . . . . 50
Running real time acquisition . . . . . 51
Spectroscopic Acquisition for UVISEL Ellipsometer . . . . . 51
Accessing . . . . . 52
Spectroscopic acquisition setup . . . . . 52
Graph Setup . . . . . 56
Graph . . . . . 56
Multiwavelength Acquisition for UVISEL Ellipsometer . . . . . 58
Accessing . . . . . 58
MWL Acquisition Setup . . . . . 58
Graph Setup . . . . . 63
Graph . . . . . 63
Views and Setup for UVISEL2 System . . . . . 65
Accessing . . . . . 65
Real Time measurements panel . . . . . 66
Sample Vision panel . . . . . 68
Tilt Vision . . . . . 68
Acquisition with UVISEL2 . . . . . 70
Launching acquisition . . . . . 73
Views for MM-16 Ellipsometer . . . . . 75
Acquisition Parameters Test . . . . . 76
System Calibration . . . . . 78
Acquisition Setup routines . . . . . 80
Views for Reflectometer (UVISEL VIP) . . . . . 86
Acquisition for Reflectometer (UVISEL VIP) . . . . . 87
Accessing . . . . . 87

**Reflectance/Transmittance Measurements on UVISEL . . . . . 88**

Definition . . . . . 88
Measuring the reflectance of your sample . . . . . 88
Measuring the transmittance of your sample . . . . . 92

**Reflectance/Transmittance Measurements on AutoSE . . . . . 95**

Definition . . . . . 95
Measuring the reflectance of your sample . . . . . 95
Measuring the transmittance of your sample . . . . . 98

**Reflectance/Transmittance Measurements on UVISEL2 . . . . . 99**

Definition . . . . . 99
Preliminary . . . . . 99
Measuring the reflectance of your sample . . . . . 100

Measuring the transmittance of your sample . . . . . 106
Principle . . . . . 106
Procedure . . . . . 106

**Simulated Acquisition . . . . . 109**

Definition . . . . . 109
Accessing . . . . . 109
Description . . . . . 110

**Reprocessing . . . . . 113**

Definition . . . . . 113
Accessing . . . . . 113
Description . . . . . 114

**Data Manipulation . . . . . 117**

Accessing . . . . . 117
Spectrum logic comprehension in DP2 . . . . . 119
Data Manipulation Operations . . . . . 120
Edit Feature . . . . . 120

**Acquisition and Data Manipulation Scenarios . . . . . 132**

Acquisition Scenario . . . . . 134
Accessing . . . . . 134
Description . . . . . 135
Data Manipulation Scenario . . . . . 136
Accessing . . . . . 137
Description . . . . . 138

**Getting Started Session with UVISEL VIP System . . . . . 141**

Power on the System . . . . . 141
Starting a session . . . . . 141
Pre-adjustments: general . . . . . 142
Pre-adjustments: Dies Grid creation . . . . . 146
Adjustments: measurement points setup . . . . . 152
Execution: Recipe Creation . . . . . 155

**Modeling . . . . . 157**

References and other materials . . . . . 157
Material Building . . . . . 157
Launching the build material . . . . . 157
Build an alloy . . . . . 158
Set a Dispersion Formula . . . . . 159
User defined formula . . . . . 160

Point by Point Calculated (n, k) . . . . . 161
Model Building . . . . . 166
Launching the building screen . . . . . 166
Screen Layout Setup . . . . . 167
Tools to build a model . . . . . 168
Simulation . . . . . 173
Fitting . . . . . 182
General Fitting Choice properties Setup . . . . . 182
Specific Fitting Choice properties Setup . . . . . 186
General fitting conditions: Modeling Description . . . . . 188
Advanced Model Post-calculation features . . . . . 206
Adding an additional Observable . . . . . 206
Thicknesses calculations . . . . . 207
Advanced Layers Post-calculations . . . . . 208
Adding an additional Observable . . . . . 208
Adding Ge composition and Eg parameters . . . . . 209

**Serial Multimodel . . . . . 210**

Definition . . . . . 210
Accessing . . . . . 210
Description . . . . . 210
How to assign an index number . . . . . 212

**Bound Multimodel . . . . . 215**

Definition . . . . . 215
Accessing . . . . . 215
Description . . . . . 215

**Modeling Scenario . . . . . 219**

Definition . . . . . 219
Accessing . . . . . 219
Description . . . . . 219
Procedure . . . . . 220

**GridEditor . . . . . 225**

What is the GridEditor? . . . . . 225
Starting GridEditor . . . . . 225
Using Simple Grid . . . . . 226
Using Complex Grid . . . . . 244

**Recipe . . . . . 254**

Immediate Analysis Recipe . . . . . 254
Accessing . . . . . 254
Mapping Recipe using a defined simple grid . . . . . 255

Accessing . . . . . 256
Mapping Steps creation . . . . . 256
Running the Recipe . . . . . 260
Output Description feature . . . . . 264
Mapping Recipe using a defined complex grid . . . . . 267
Accessing . . . . . 267
Mapping Steps creation . . . . . 268
Running the Recipe . . . . . 272
Output Description feature . . . . . 276

**Reading Recipe Results files (.RCR) . . . . . 277**

**Reporting . . . . . 280**

Creating Template . . . . . 280
Template Setup . . . . . 280
Generating a report . . . . . 285
How to use the created report template ? . . . . . 286

**Kinetic Measurements . . . . . 289**

General . . . . . 289
Acquisition Routine . . . . . 290
Model Creation . . . . . 297
Simulation . . . . . 303
Kinetic Recipe . . . . . 303
Fitting Process . . . . . 304
Saving Kinetic results . . . . . 308

**Checking files . . . . . 310**

Accessing . . . . . 310

**Cooling/Heating Stage . . . . . 315**

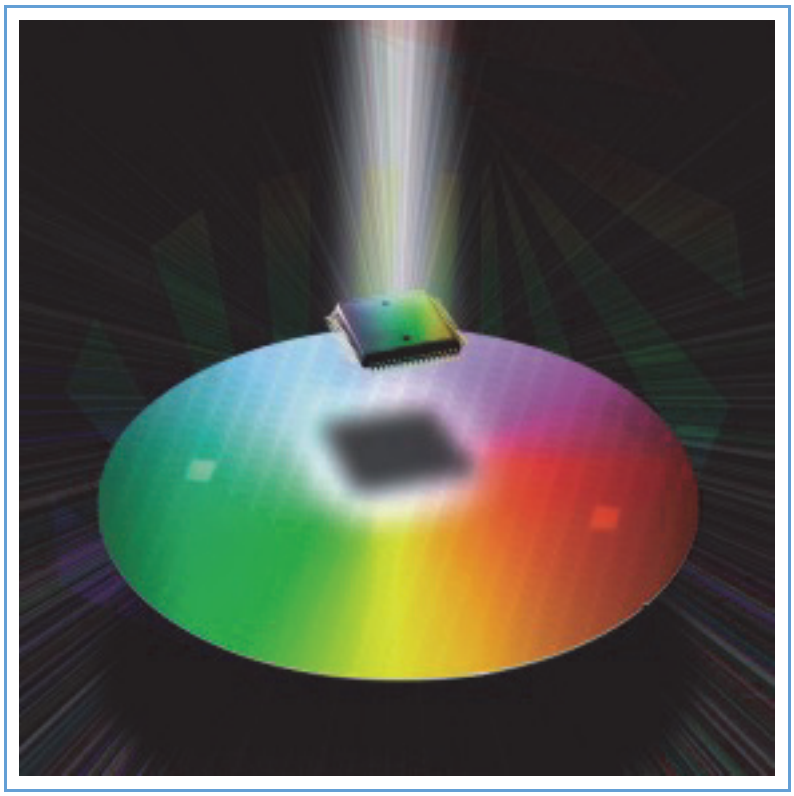
Definition . . . . . 315
Driver Installation . . . . . 315
Accessing . . . . . 317
Creating a Temperature Routine . . . . . 319
How to set the Routine . . . . . 319
How to use the Routine . . . . . 320
Creating an Immediate Recipe with Temperature Control . . . . . 320

**Troubleshooting . . . . . 327**

**Index . . . . . 328**

05/20/2016

# DeltaPsi2™ Software
## Reference Manual



*The DeltaPsi2 Software is a completely integrated program for acquisition and treatment of ellipsometric Data.*
*Ellipsometry is an optical method for the characterization of an interface or a film between two media. The Ellipsometry method is based on the resulting processes from the polarization variation. This variation occurs every time a controlled polarized light is reflected from or transmitted through the interface or film.*

05/20/2016

# 1 Getting Started

## 1.1 Running DeltaPsi2 Software

To run the DeltaPsi2 Software, follow these instructions:

- Using the mouse, double-click on the DeltaPsi2 icon 

located on the «Windows™ desktop».
- DeltaPsi Software will start up.

## 1.2 Configuring the Working Environment

To get the most out of DeltaPsi2, you should be familiar with some of its general operating concepts. The following information should help you to understand the DeltaPsi2 working model in a few key areas.

Since DeltaPsi2 is a data-based processing software which uses files, it is important to understand the files structure and organization before working with DeltaPsi2.

Once running, the following screen will be displayed:

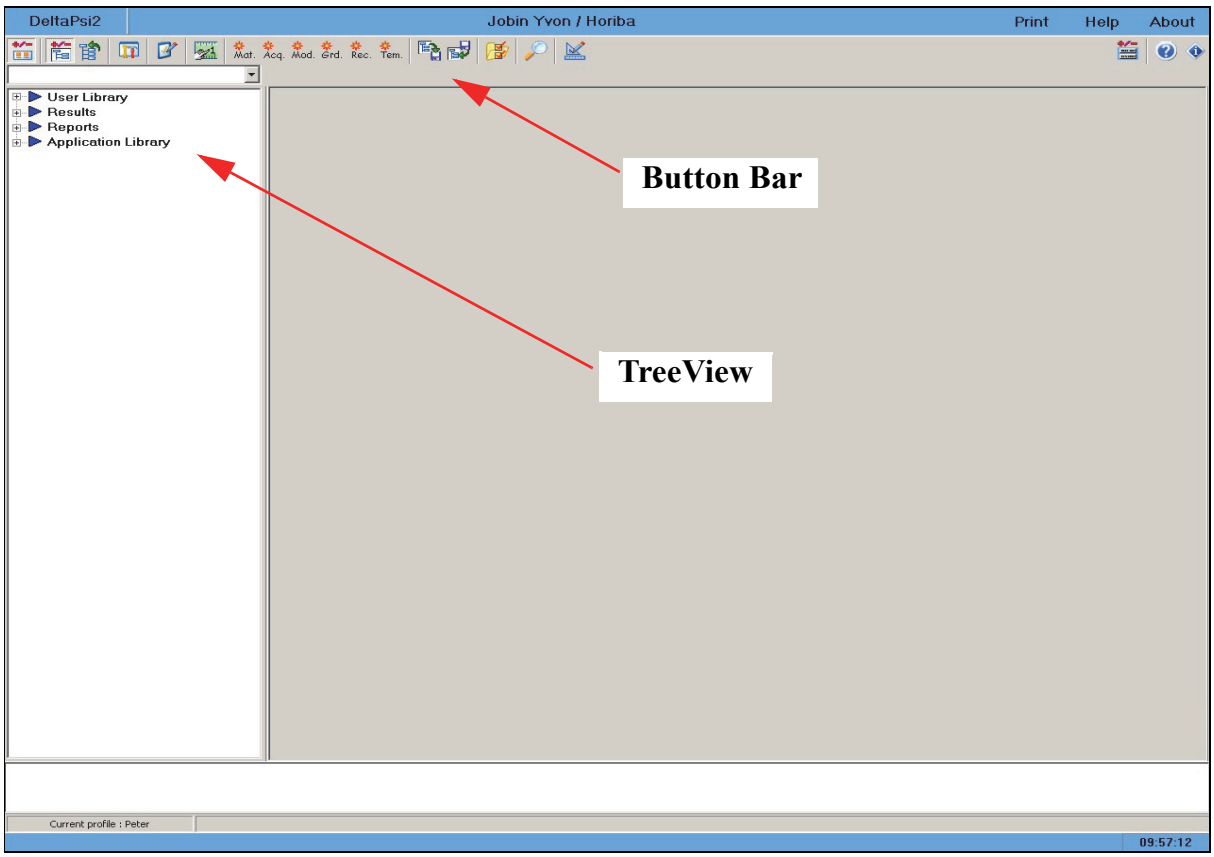



**Figure 1-1 DeltaPsi2 Start screen**

The DeltaPsi2 Software is organized around the Button Bar and the TreeView window. The Button Bar is located on the top of the screen and the TreeView, on the left hand side of the screen.

### 1.2.1 TreeView data files environment

Figure 1-2 shows the general TreeView data files environment. Two main Libraries are listed. The user data files are rearranged inside a unique node: «User Library». The «Application Library» includes the reference and licensed data delivered with the DeltaPsi2 package. For safety purposes, these data cannot be modified or deleted from the DeltaPsi2 interface.

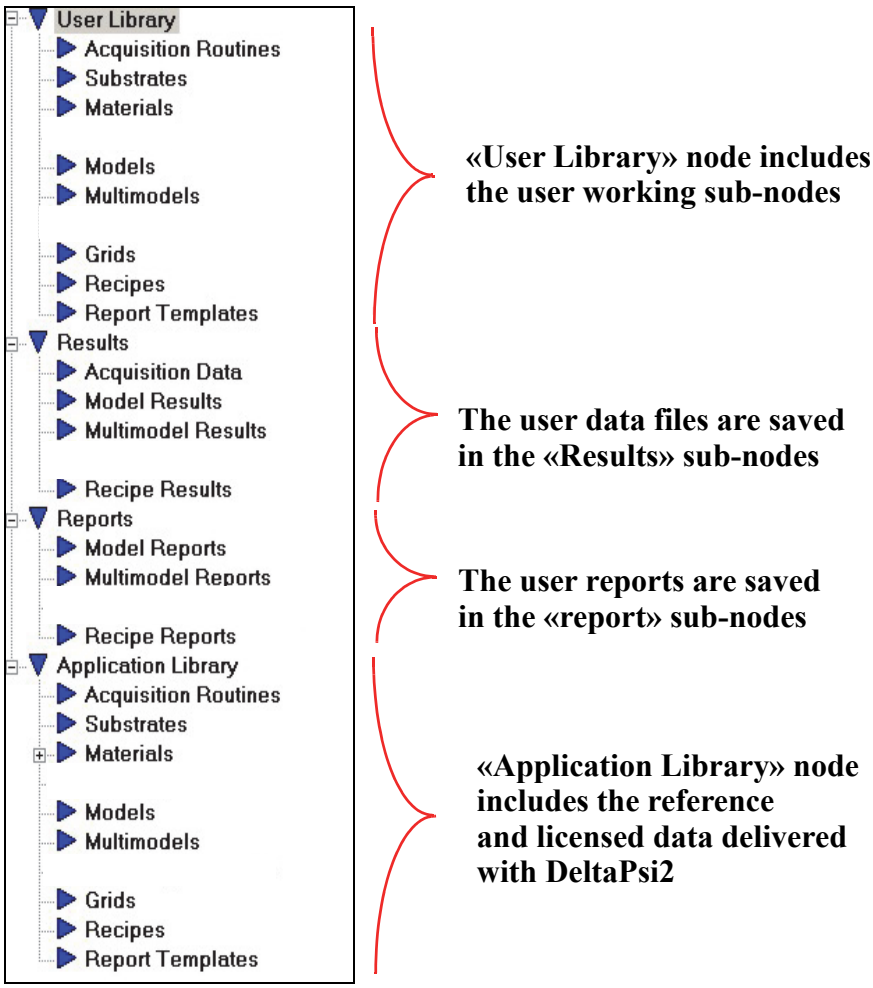



**Figure 1-2 TreeView data files environment**

### 1.2.2 How to create a new user Profile

DeltaPsi2 includes a powerful tool which allows the user to create one or several user Profiles. This Profile feature works as a preset which assigns every type of user file to a specific TreeView node and sub-node.

If several projects are planned or if several users should work on the same computer, a different working user Profile should be built.

The following procedure will allow you to setup DeltaPsi2 on your new working environment. This step is highly recommended and will help you to understand the philosophy of the Software.

**1** Click on the Software Options icon  to open the ***Software Options*** screen. The following screen will be displayed:

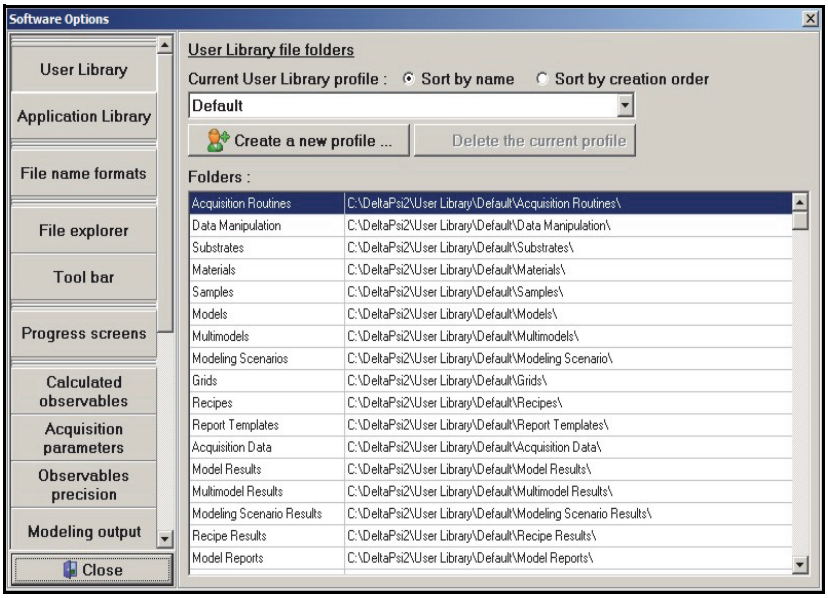



**Figure 1-3 Software options**

**2** Select the ***User Library Folders*** tab as shown on the figure above.

- The first parameter that should be set concerns the user profile. To create a new user profile, click on the ***Create a New Profile*** button  . The following pop-up window will be displayed:


- Enter the new ***Profile*** name then click ***OK*** to validate.
- The package folders for the new ***Profile*** are automatically created.

The created package folders are important for a convenient use of the packages. Each time you have to receive from or send to somebody ellipsometric results or materials you can unpack or pack them using DeltaPsi2. These folders will help you to manage these type of files. The ***Package*** feature is detailed on page 30.

### 1.2.3 How to create a new folder

A new folder can be created everywhere from the TreeView window: right click on a folder then select the ***New folder*** choice. Then, a pop-up screen will ask you to enter the new folder name. Click ***OK*** to validate.

**NOTICE**: Do not create folders outside the DeltaPsi2 Software: It could generate unpredictable errors. Indeed, the data files created during the DeltaPsi2 session are linked together. Moreover, the ***Export/Import Package*** feature automatically performs a perfect data files transfer from one computer to another (see TreeView Options page 29).

## 1.3 Software Options

The previous chapters detail how to create new working folders environment. However, many other options can be set from DeltaPsi2 Software according to the analysis requirements. This chapter will detail these working preferences. The following screen shows how to open the ***Software Options*** screen.

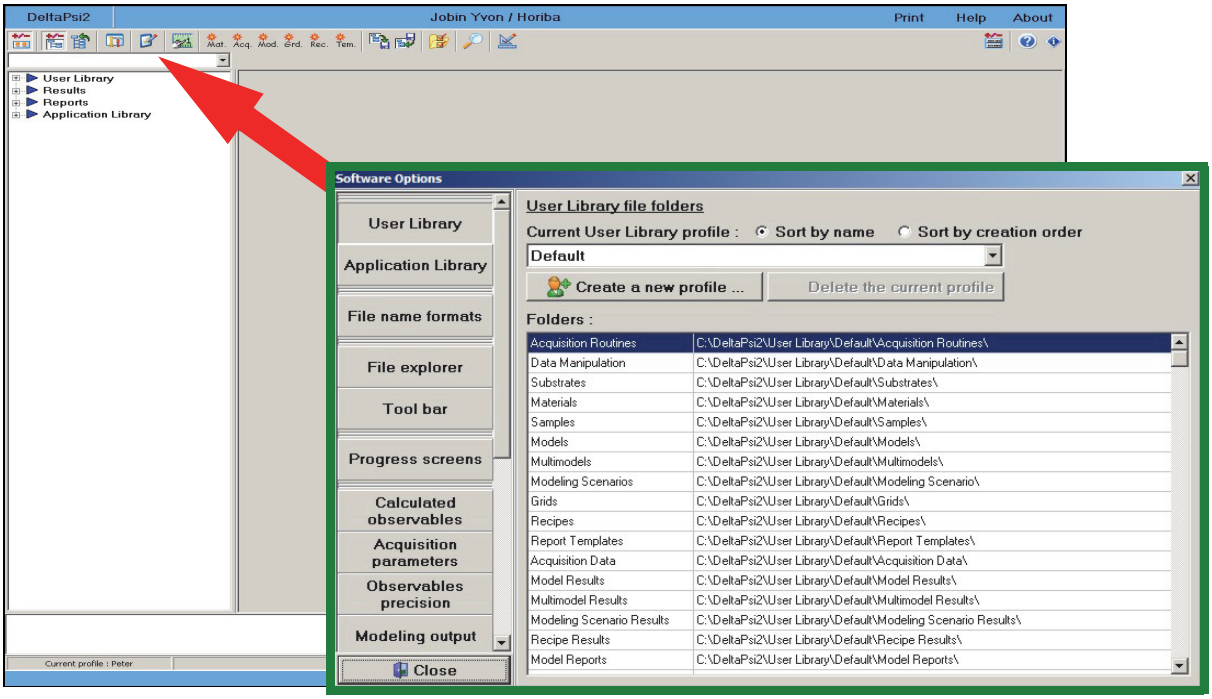



**Figure 1-4 Software Options screen**

### 1.3.1 Application Library Folders

Clicking on the ***Application Library Folders*** tab will list the folders related to the licensed or not licensed data applications delivered with the DeltaPsi Software. We do not recommend to change or move these folders.

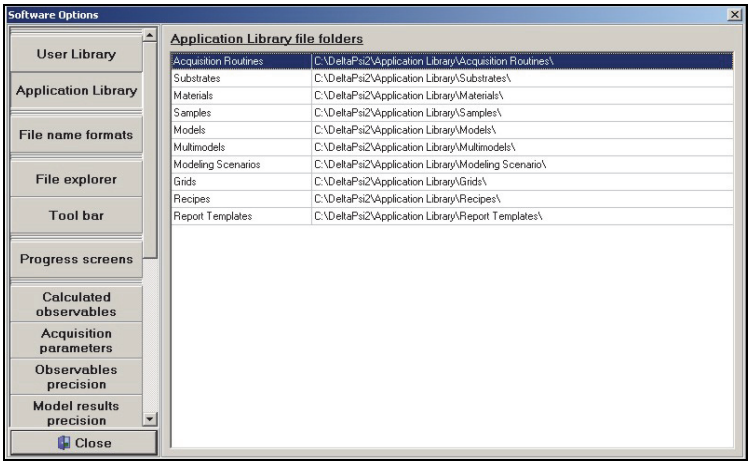



**Figure 1-5 Application Library folders screen**

### 1.3.2 File name formats

- **Acquisition result file naming format**

The Acquisition result can be sorted using a specific format order. Select the desired format from the drop-down menu.

- **Acquisition scenario result file naming format**

The acquisition scenario can be also sorted with a specific format order. Select the desired format from the drop-down menu.

- **Model result file naming format**

The same as for the Acquisition result, each Model result can be sorted using a specific format order. Select the desired format from the drop-down menu.

- **Recipe result file naming format**

The same as for the Acquisition result, each Recipe result can be sorted using a specific format order. Select the desired format from the drop-down menu.

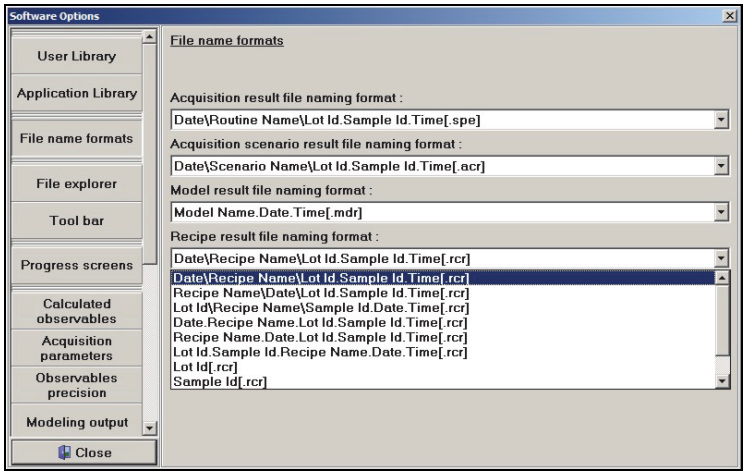



**Figure 1-6 Application Library folders screen**

### 1.3.3 File Explorer

The ***File Explorer*** screen displays the TreeView nodes. If a specific node is not used, it can be removed from the TreeView list. Just, check out the node to remove.

The upper field allows the user to choose the sorting order of the listed files.

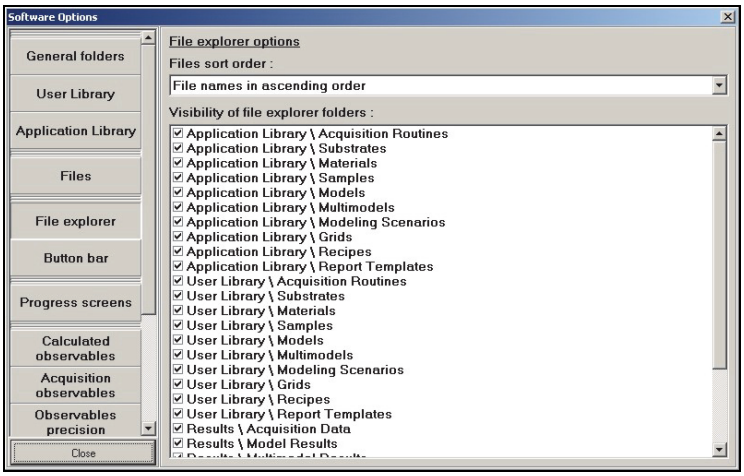



**Figure 1-7 File Explorer**

### 1.3.4 Tool Bar

The ***Tool Bar*** screen shows the action icons located on the upper part of the ***Main*** screen. By default, all action icons are selected. The user can deselect the actions which are not necessary.

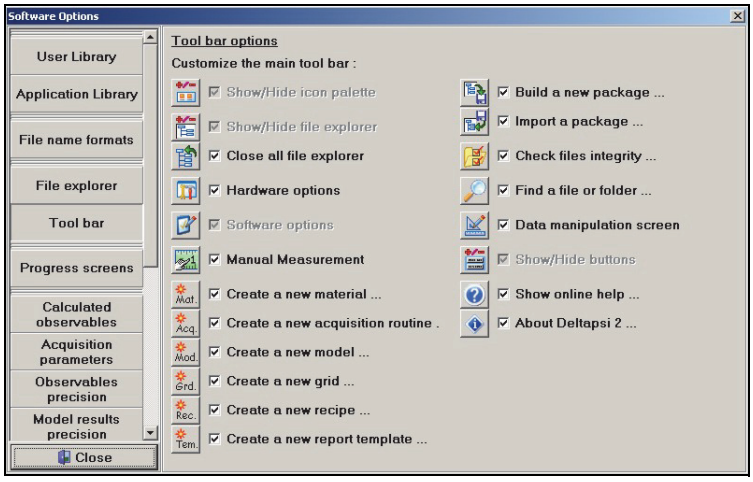



**Figure 1-8 Tool bar setup**

### 1.3.5 Progress Screens

- **Show progress screens for Manual Mode, Manually started recipe and automatically started recipe.**

If checked, the Acquisition and/or Fitting and/or Recipe progress screen will be displayed during the process.

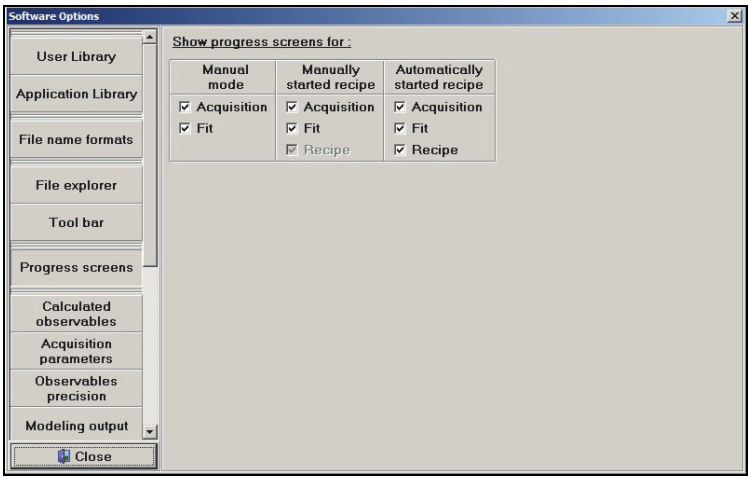



**Figure 1-9 Progress Screen Setup**

### 1.3.6 Observables Setup

The ***Observables Setup*** are set via three parameters groups which can be activated for various analysis purposes. By default, the main parameters are activated. Figure 1-10 shows the displayed screens according to the selected buttons.

***Calculated Observables*** are the parameters available to define the axis of any standard Graphics Manipulation Screen (See "Graphics Screen" on page 33).

***Acquisition Parameters*** are parameters that are stored in the raw data file or *.spe file after an acquisition is completed. These parameters are the permanent record of the acquisition. If the Software controls several applications e.g. a UVISEL with a Reflectometer, the parameters will be listed separately. Select the parameters from the list that you wish to be stored with the *.spe file. These values are unrelated to those displayed graphically during the acquisition. See "Graph Setup" on page 56 to select the parameters you wish to see during an acquisition.

***Observables Precision*** will determine the number of significant digits displayed for either Calculated or Acquisition Observables.

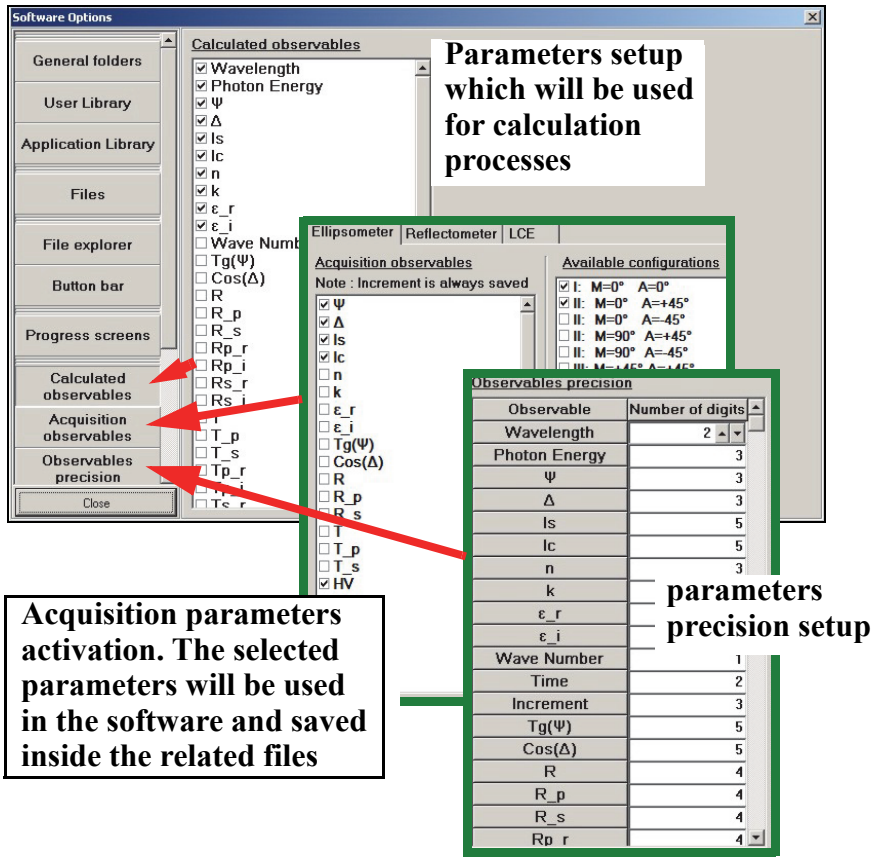



**Figure 1-10 Observables setup**

### 1.3.7 Modeling Output

When a Modeling Output is generated, it becomes possible to set the precision level of the exported modeling values.

Check in the ***Translate alias parameter names*** to translate the greek to latin fonts. This is only required for some specific uses which do not accept greeks fonts. By default this choice is not activated.

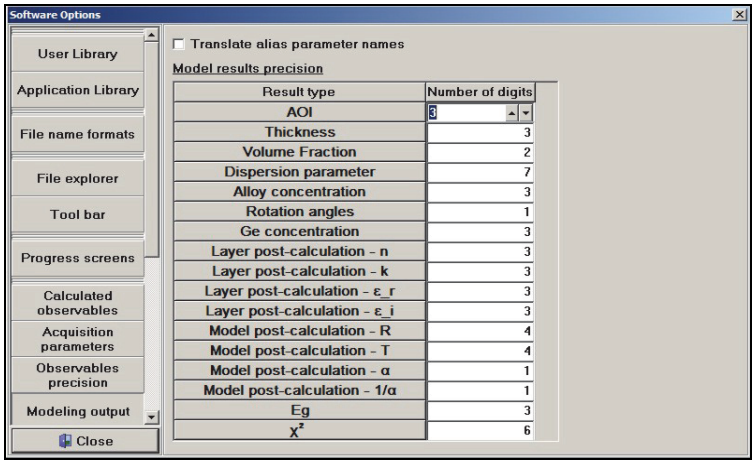



**Figure 1-11 Modeling output**

### 1.3.8 Error values for Fit

Each measured value used in the calculated algorithm can be assigned to a specific error value. The default values is highly recommended. If all values are changed by an identical value, the fitting result will not be transformed but the $\chi^2$ value will be modified.

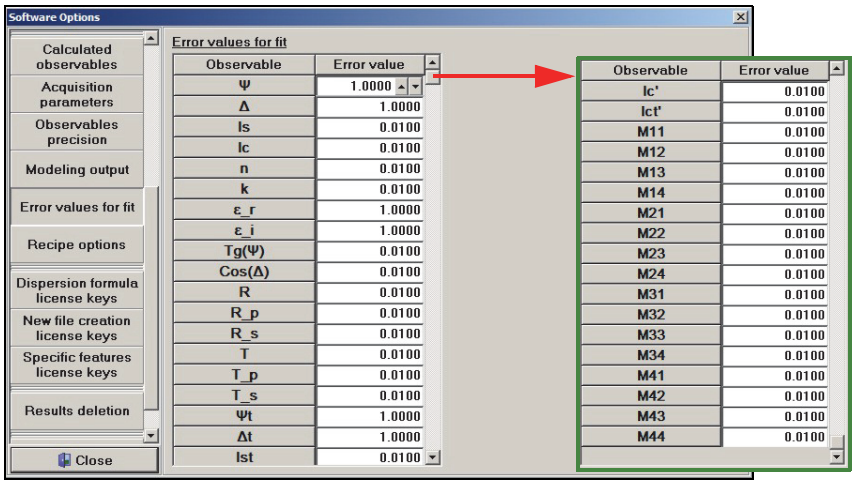



**Figure 1-12 Error values for Fit**

### 1.3.9 Recipe Options

The Recipe Options screen displays the options related to the recipe. These options are clearly described on the screen shown below:

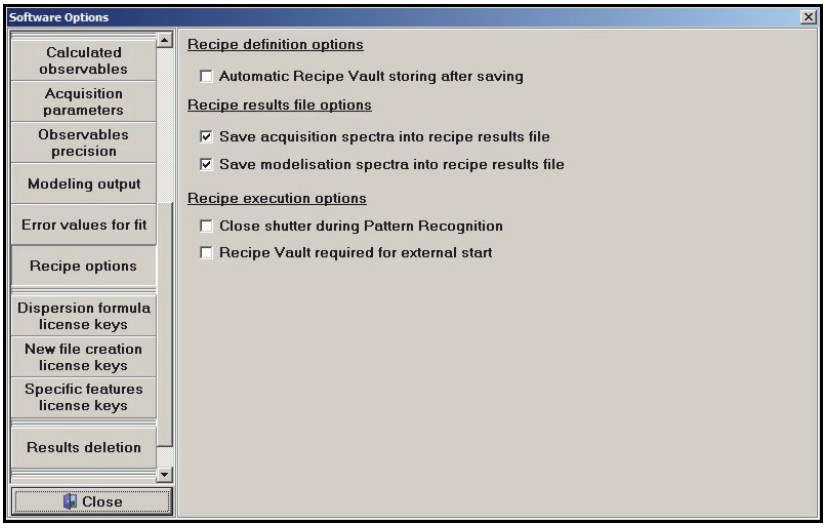



**Figure 1-13 Recipe Options**

### 1.3.10 License keys

DeltaPsi2 Software intensively uses powerful set of data processing modules. These processing modules are licensed and delivered on request.

The three ***License keys*** buttons display the list of the installed processing license keys. Please contact HORIBA Scientific or your local representative for additional licensed modules.

.

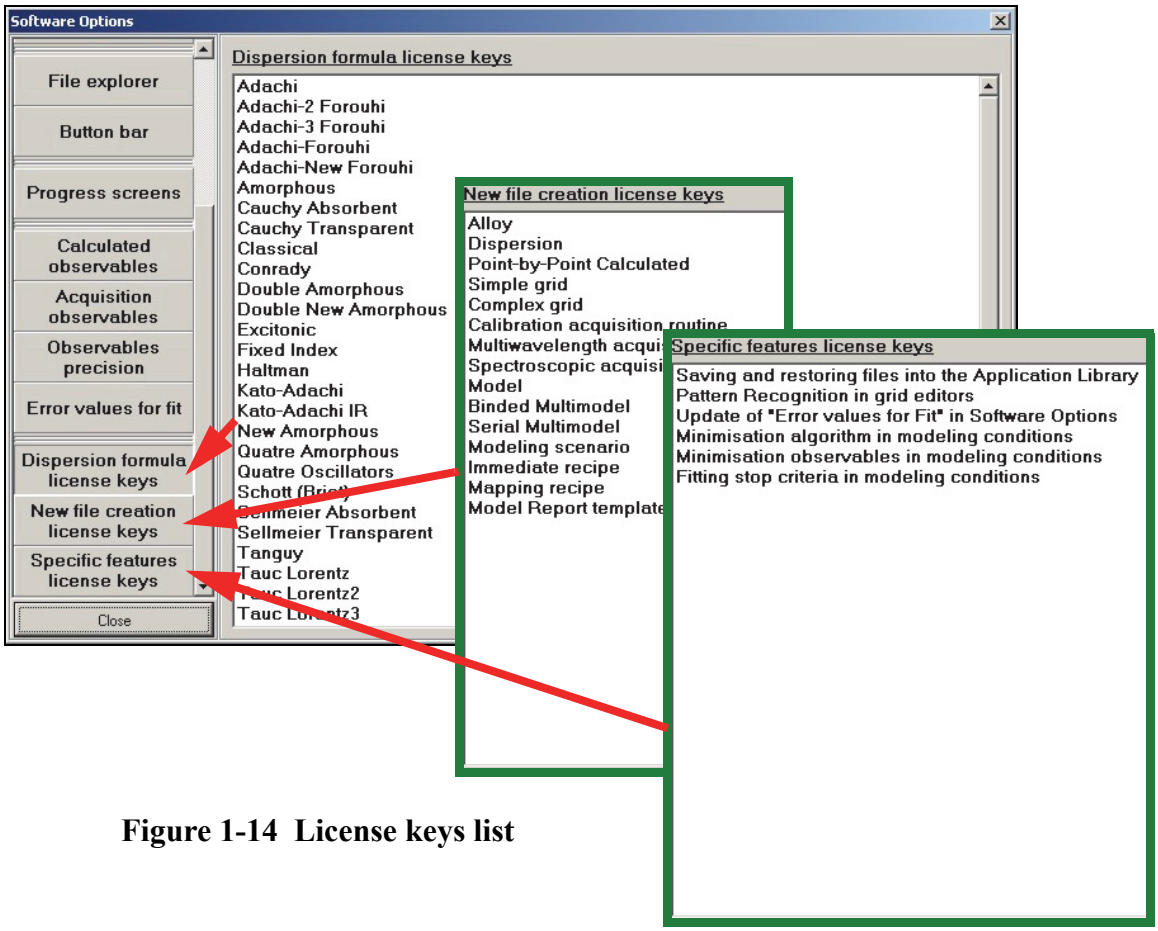



**Figure 1-14 License keys list**

### 1.3.11 Results deletion

With extensive use of the system, the number of files recorded on the hard disk drive could be large enough to affect the software's performance. The Results Deletion screen allows the user to delete old files from the DeltaPsi2 folders to the Garbage folder or completely delete these files. Some other options are available: deletion with specific *.spe, *.mdr, *.mmr, *.scr or *.rcr files and automatic activation with logs or not.

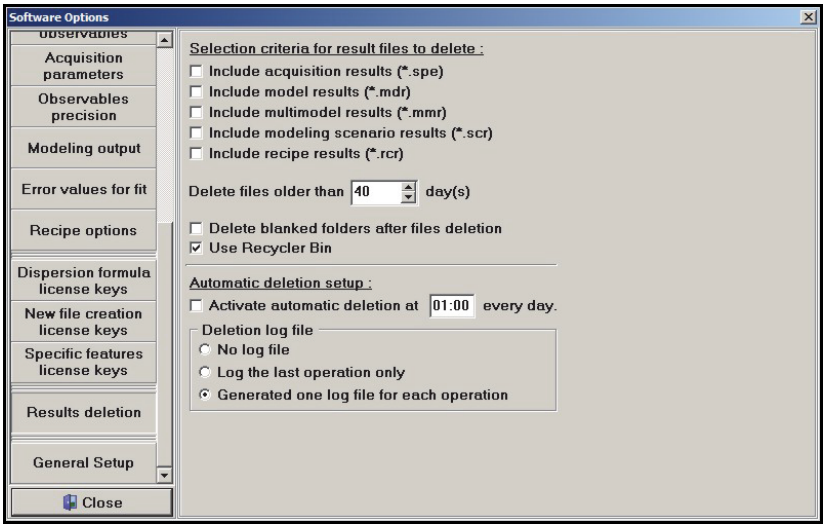



**Figure 1-15 Results deletion**

### 1.3.12 General Setup

The **General Setup** screen displays the main installation folders. These are only information fields useful for the Pattern Recognition (P.R.) program. The Pattern Recognition program is an option useful for locating specific measurement sites on patterned wafer and samples and requires a separate installation.

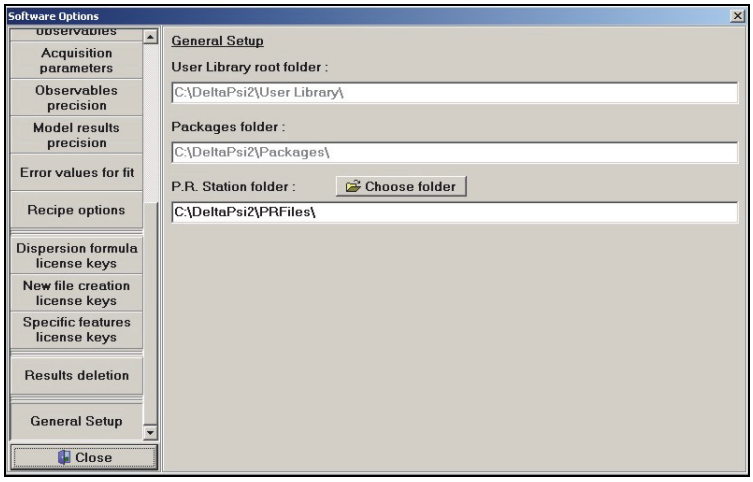
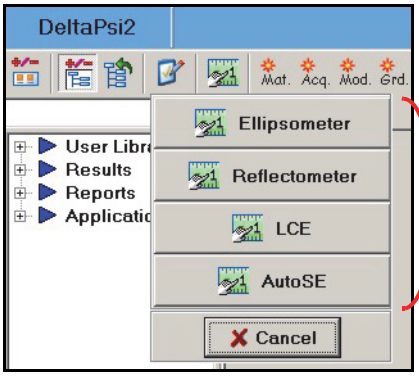



**Figure 1-16 General folders screen**

# 2 Main Working Interface

## 2.1 Accessing

Click the ***Manual Measurement*** icon  located on the ***Button Bar.*** Depending on the ordered System and the hardware configuration, a popup screen could list several applications. Select the appropriate application to launch the View screen (Real Time Acquisition). According to the selected application, the Views screen can be very different. This chapter will detail the working philosophy.




Select one of the available applications.
Only valid applications are displayed.

## 2.2 General Graphics Interface

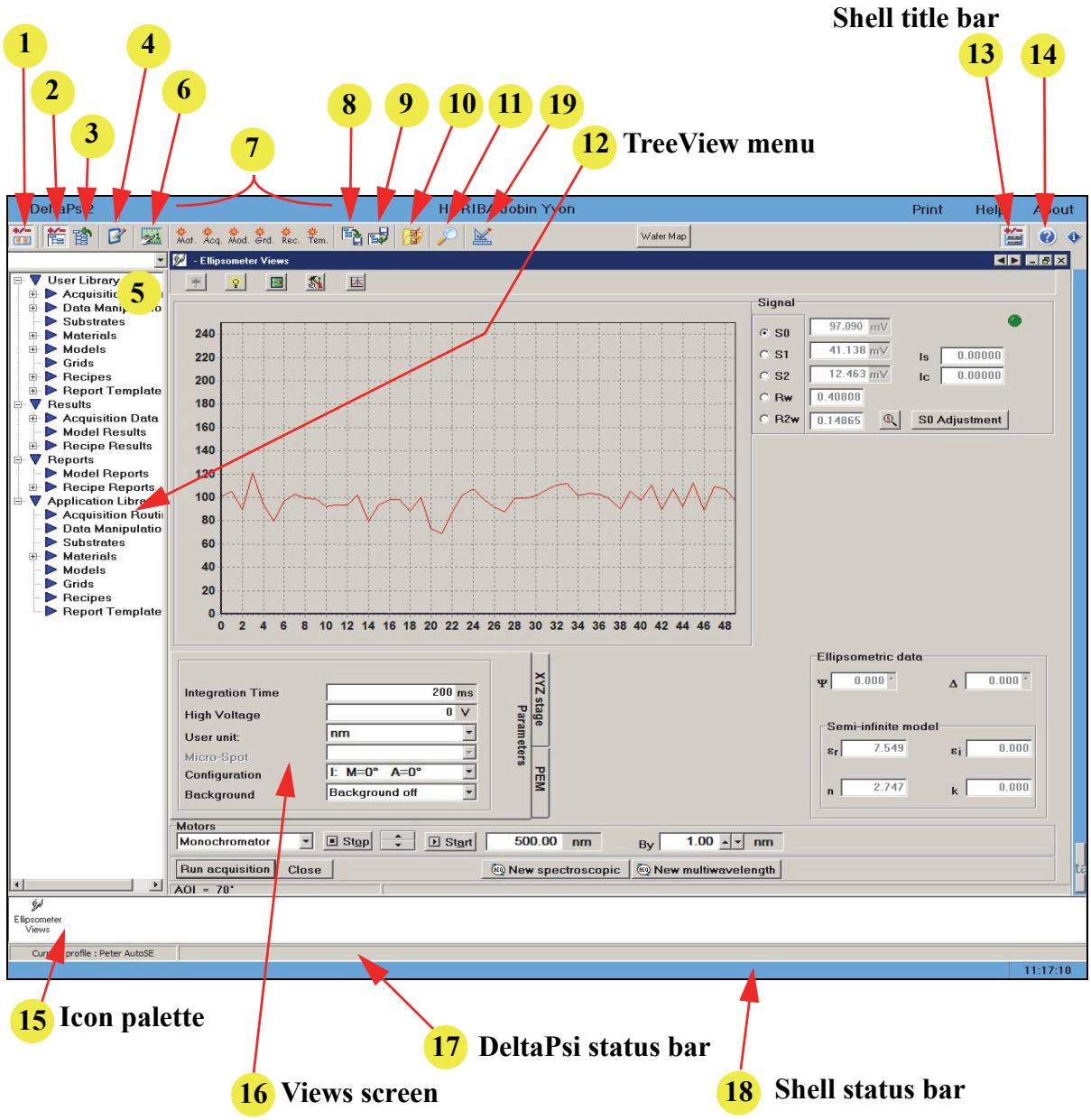



**Figure 2-1 Managing Views screen (UVISEL System example is shown)**

05/20/2016

**1** **15** The Show/Hide icon palette is a toggle command which hides the area located below the TreeView and the Main Screen. This area displays an icon for every file which has been loaded. Thus, it becomes easy to take a look and switch between the currently loaded files.

**2** The Show/Hide ***TreeView*** icon is a toggle command which hides the TreeView menu. This is useful as it magnifies the working View screen.

**3** This command is a short-cut which closes the opened ***TreeView Explorer*** folders.

**4**  Open the ***Options Software*** screen (see *"Software Options" on page 14*).

**5** The ***TreeView Explorer*** simplifies the access to the TreeView. Click on the field to activate the scroll bar, then select the node to open it.

**6**  The ***Manual Measurement*** icon launches the ***Views*** screen which is shown on the figure 2-1, page 23. The Views screen is in fact the Real Time Acquisition panel which is detailed on the chapter "Views for UVISEL Ellipsometer" on page 43. Depending on your System(s), an intermediate screen could ask for the acquisition to launch.

**7** 

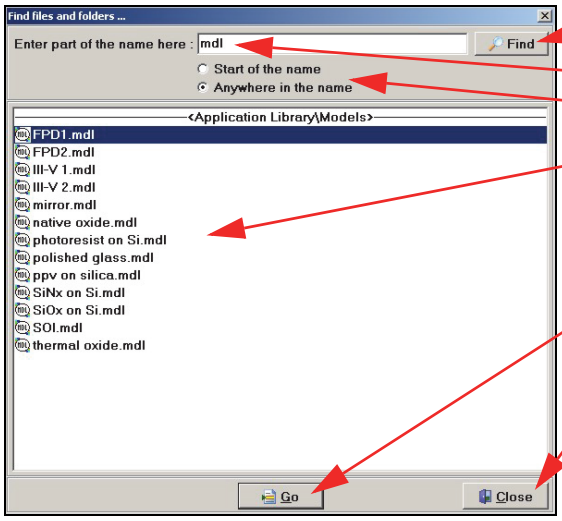
***Creation*** icons: from these «short-cut» icons, the user can create a new material, acquisition routine, model, grid, recipe or report template.

**8**  Launch the ***Export*** files package feature. See the detailed description chapter "Import/Export feature" on page 29.

**9**  Launch the ***Import*** files package feature. See the detailed description chapter "Import/Export feature" on page 29.

**10**  The Check Files Integrity is a very powerful feature which is recommended for experimented users only. This feature analyzes the files integrity and verify the links between built models, recipes etc...

**11**  Click on the ***Search*** feature to find a DeltaPsi2 file. The following screen will be displayed:




Activates the search

Enter the query string to find

Select specific criteria

Select the searched file in the list, then click on the Go icon to select this file in the TreeView.
Double-click goes directly to the file.

Close the **Search** screen and select the searched file in the TreeView

Close the **Search** screen

**NOTICE**: the * sign must not be used in the search query string

05/20/2016

**12** ***TreeView*** Main Menu allows the user to access files. It contains nodes which are detailed below:

Functional node which is used to browse through the files of the software. There are three possibilities to open/close the functional node:

- Using the left button of the mouse, double-click the node name,
- Left-click on the +/- indicator in front of the icon. If this indicator does not exist, the node is empty,

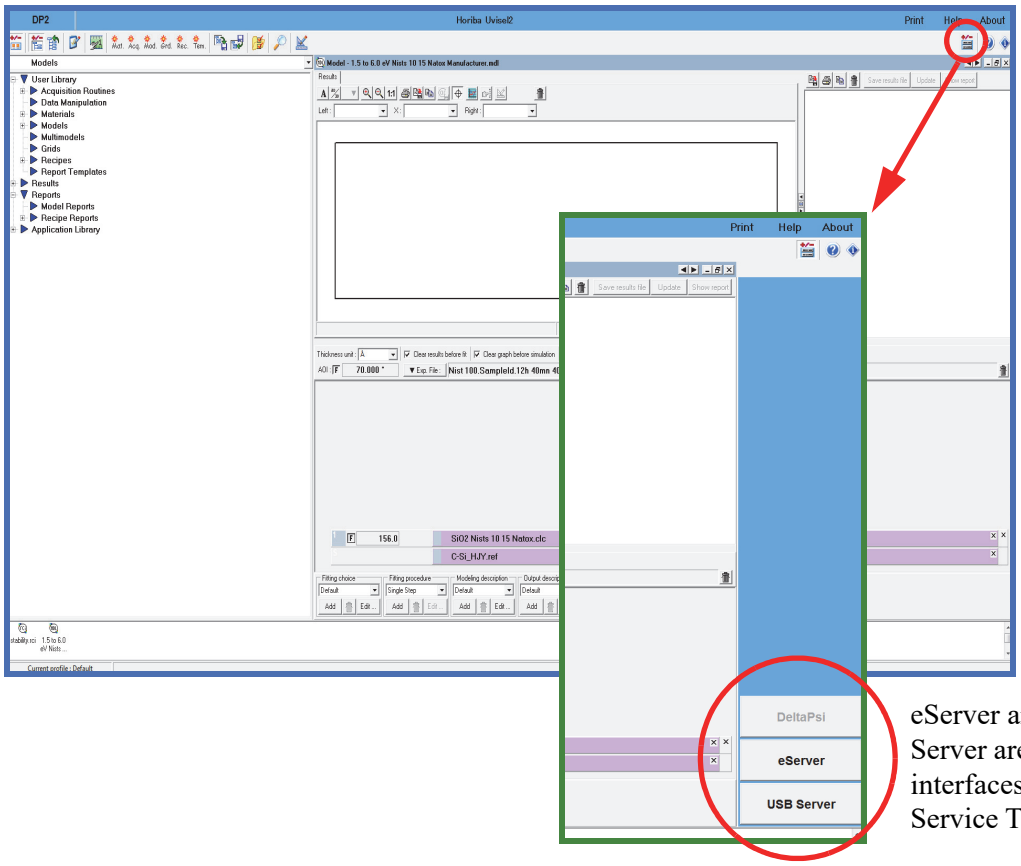
- DeltaPsi files: each file node represents a DeltaPsi file located on the hard disk. To open the file, double-click with the left button of the mouse.

  Each type of file is marked on the icon: i.e. SPE for spectroscopy, REF for reference...

- Unknown file: this icon signifies that an unknown file has been found.

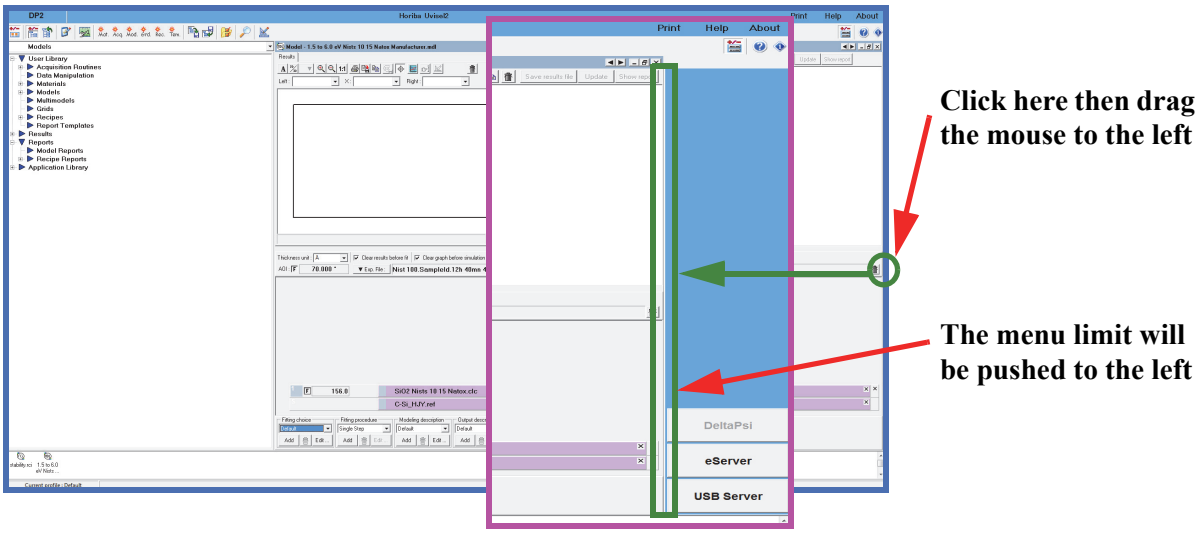
**NOTICE**: Right-clicking on an element name will display a popup menu. This menu contains specific options related to the node. Do not hesitate to try features included in this menu.

**13** ***Show/Hide button:*** once activated, an additional menu is displayed on the right hand side of the current desktop screen. This menu is mainly useful to switch between the internal DeltaPsi2 interfaces.

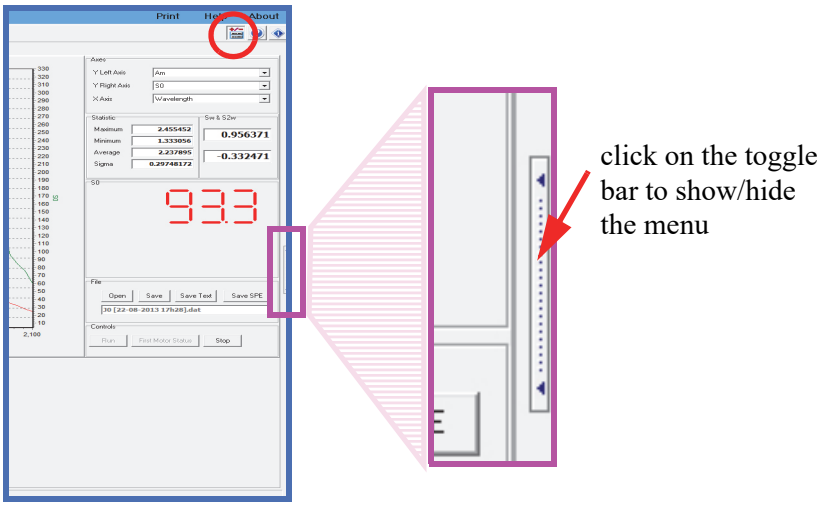



eServer and USB Server are internal interfaces used by Service Team

In some circumstances, this menu is not displayed. This is only possible if the menu has been manually hidden by moving the menu limit. In such situation and using the mouse, move this limit to left to enlarge the menu area (see figure below).

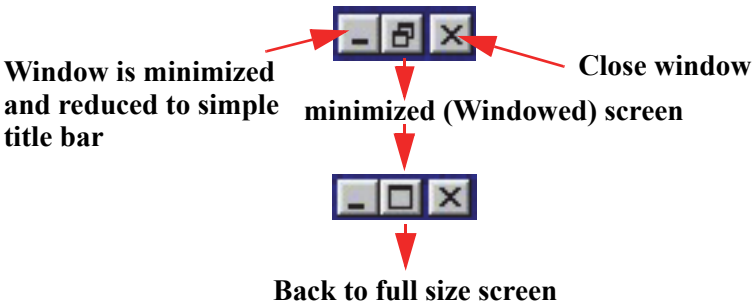


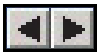

From the eServer application, if the ***Show/Hide button*** is activated, the menu could be hidden too if the menu has been previously manually closed. In this case, click on the toggle bar (see the figure).




**14**  Help online is available by clicking the icon located on the ***Button Bar.***

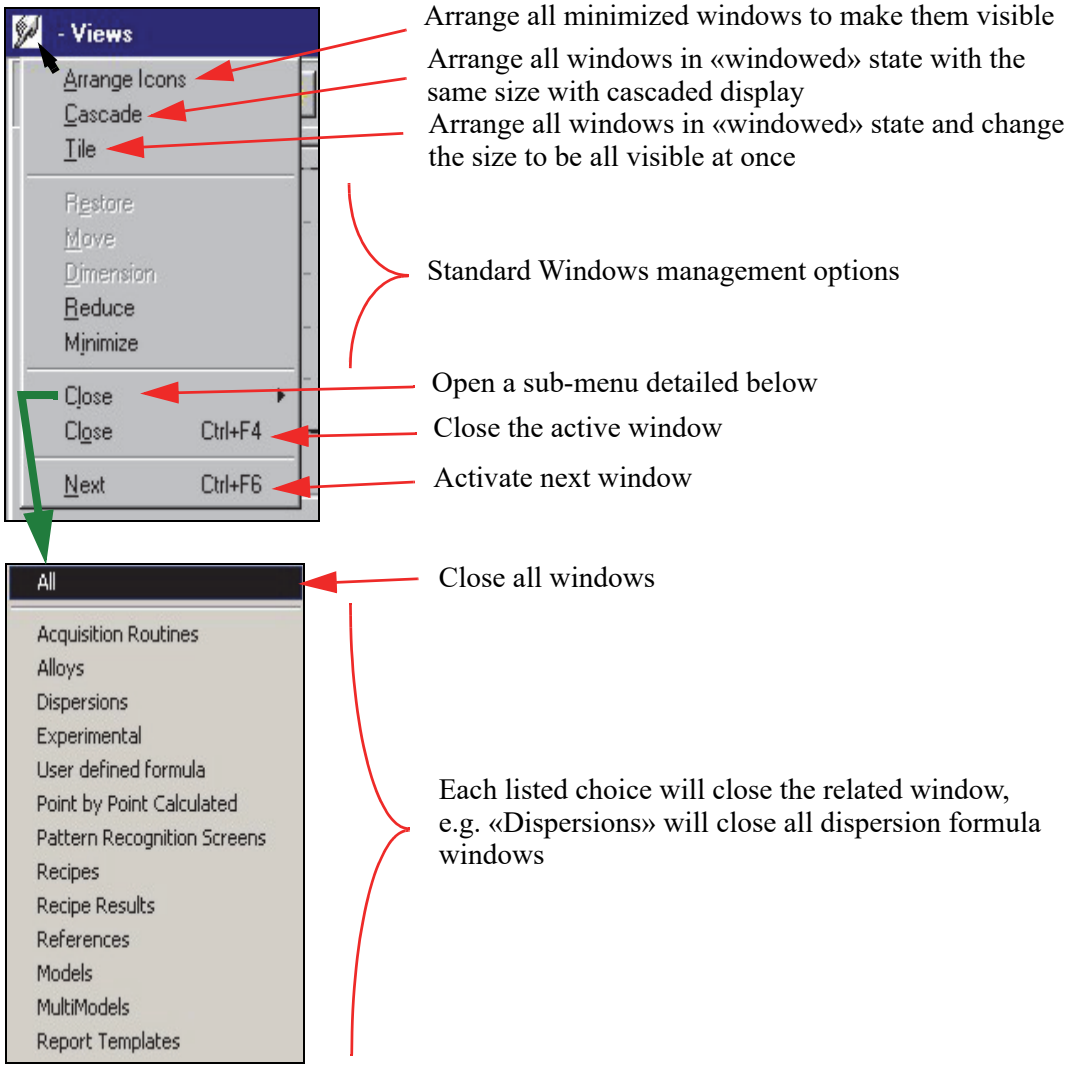
**16** Views screen: this is the main working area. This screen is in fact an independent window which can be resized using the following icon located on the upper right hand side of the screen:




 If several windows have been opened, these two buttons switch between the windows.

05/20/2016

Additional options can be accessed by clicking on the icon located on the upper left hand side of the window. Depending on your size status, some options cannot be activated (greyed). Here below is an example of such of screen:




**17** DeltaPsi2 status bar information.

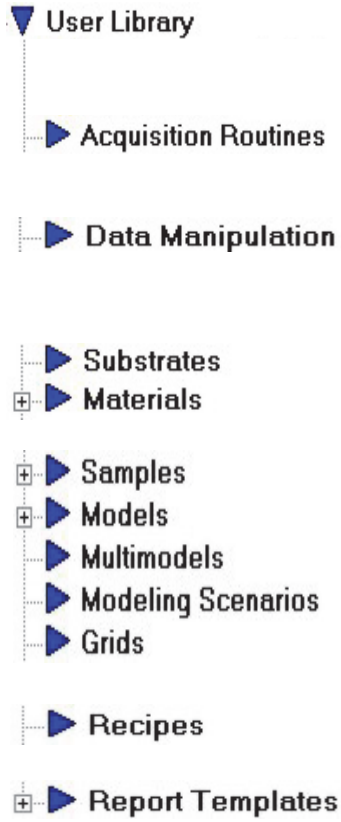
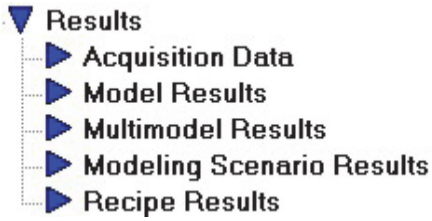
**18** Shell status bar information.

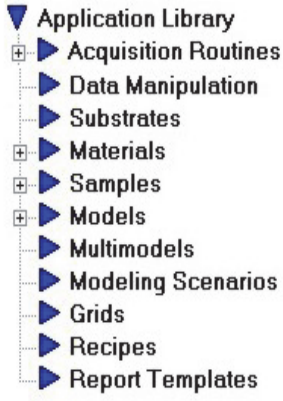
**19** Data manipulation feature

*05/20/2016*

### 2.2.1 TreeView Description

This chapter describes the TreeView list.

| Node | Description |
|---|---|
|  | User Library main node. User defined files created from DeltaPsi2 are always saved in one of these folders. |
| | Contains the acquisition routine files: *.ACQ and acquisition scenario files: *.ACN |
| | Contains the graphics screens backup of the Data Manipulations screens: *.DMS. For Scenario Data Manipulation: *.DMN |
| | Contains the substrate material files: *.REF |
| | Contains material files: reference (*.REF), dispersion formula (*.DSP), alloy (*.ALY) and point by point (*.PPC), user defined formula: *.UDF |
| | Contains Sample (*.SPL) files |
| | Contains Model (*.MDL) files, for Kinetic model: *.MDK |
| | Contains Bound Multimodel (*.BMM) and Serial Multimodel (*.SMM) files |
| | Contains Modeling Scenarios files (*.SCE) |
| | Contains Grid (*.GRD) files |
| | Contains Recipes *.RCI (immediate), *.RCM (mapping) and analytical package *.APK |
| | Contains templates: *.MDT (Model), *.MMT (Multimodel), *.SCT (Modeling scenario), *.RCT (Recipe), *.ACT (Acquisition), *.DMT (Data Manipulation), *.RSN (Report Scenario) |
|  | Contains the acquisition data files: spectroscopic (*.SPE) kinetic (*.KIN), calculated (*.CLC) and Result Acquisition Scenario (*.ACR) |
| | Contains Modeling results files (*.MDR) |
| | Contains Multimodel results files (*.MMR) |
| | Contains Modeling Scenario results files (*.SCR) |
| | Contains Recipe Results files (*.RCR) |

| Node | Description |
|---|---|
|  | These folders will receive the graphs and texts saved from various working screens but also the generated reports (MS Word, HTML, Formatted or Flow). The files formats located in these folders could be very different: *.HTML, *.WMF, *.JPG, *.DOC, *.EMF, *.TXT, *.RTF, *.RAF etc... |
|  | Application library has the same structure as the User Library except:<br>• Folders are linked to only one set of folders containing files delivered with DeltaPsi2 Software,<br>• This Library is not supposed to be altered by the user,<br>• As the Acquisition Data and Modeling Results are user data, the nodes of these files are only present in the User Library. |

### 2.2.2 TreeView Options

To access specific options of the TreeView, right-click on each node or file. The options availability depends on the node or file type. These options are arranged as follows:

- **On the upper part of the options are listed actions related to «ellipsometric» operations: running new spectroscopic acquisition routine screen, new alloy screen, new point by point calculated screen....etc**
- **The options concerning the nodes, files organization and tools are listed on the central part. Then, it is easy to create a folder, open a data file as a text format.**
- **The lower part of the popup menu concerns the *Import/Export* and *Update Model* features.**

---

**Import/Export feature**

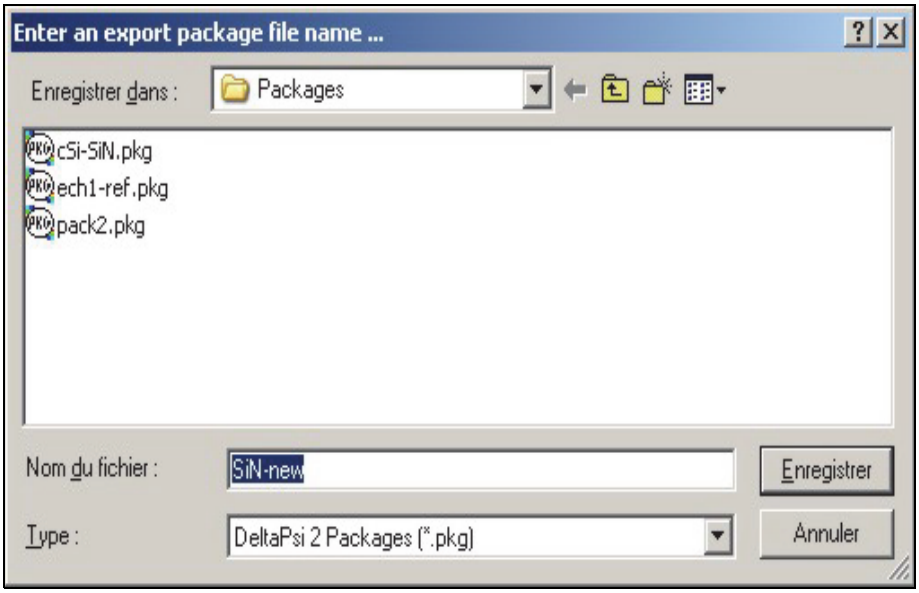
Three options are available:

***Simple Export***: only available from a file (right-click), this command creates a file ready to be exported to another system running DeltaPsi2. This export feature is not a simple file transfer, in fact DeltaPsi2 collects the data which are linked to the selected file. Then, the retrieved data on the other computer are dispatched inside the right folders.

05/20/2016

***Build Package:*** this feature allows the user to export nodes, sub-nodes or data files. The procedure is explained below:

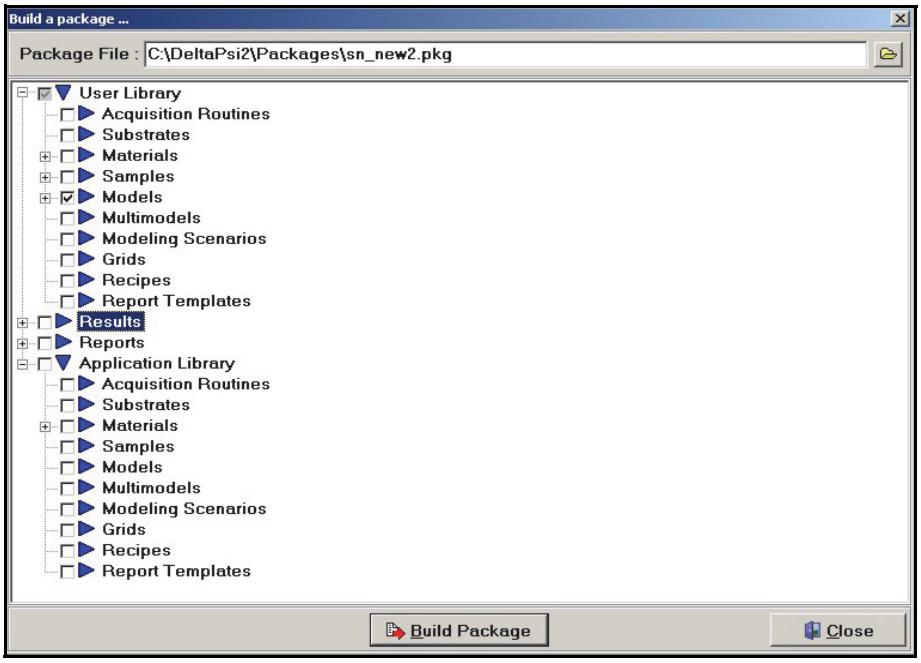
**1** Using the mouse, click on the ***Build Package*** icon located on the Button Bar menu. The following screen will be displayed:




Enter the name and choose the destination folder of the export package. Click on ***Save*** to validate the choices.

The screen shown below will be displayed: open the nodes and select those which must be exported. Every combination is possible.

Once the selection has been made, click on the ***build package*** button. Then, a popup screen informs the user for packaging creation.

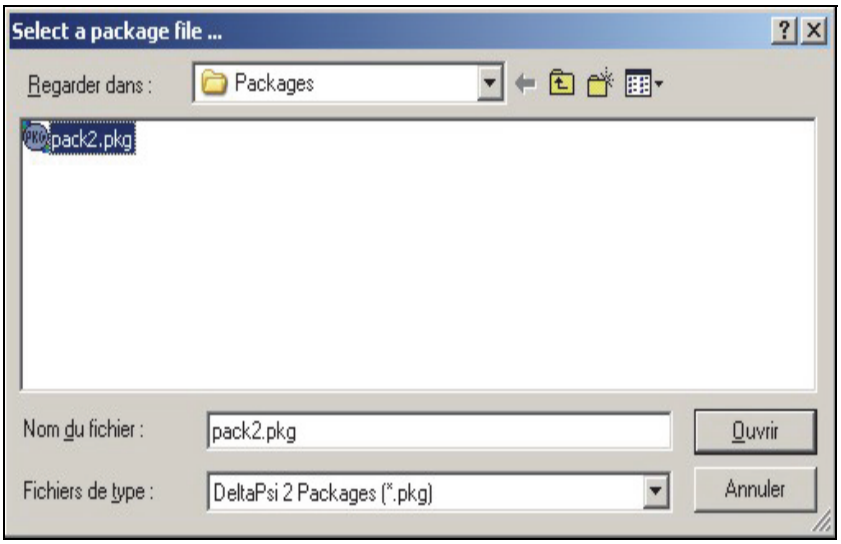



***Import Package***: this feature allows the user to retrieve a package previously created with the Export Package DeltaPsi2 feature.

05/20/2016

The procedure is explained below:

**1** Click on the Import Package icon located on the Button Bar menu. The following screen is displayed:




Select the appropriate drive and folder where the package is located to retrieve. Once found, click on the ***Open*** button. The screen shown in figure 2-2 will be displayed.

This screen lists the packaged files and the user can choose to deactivate a file or select the target library (Application or User). Take into account that in some cases, Application Library could not be available due to copyright laws. The screen also informs if a file with the same name is already present in the assigned location.

Once the files import choice has been done, click on ***Import files*** button to unpack the files.

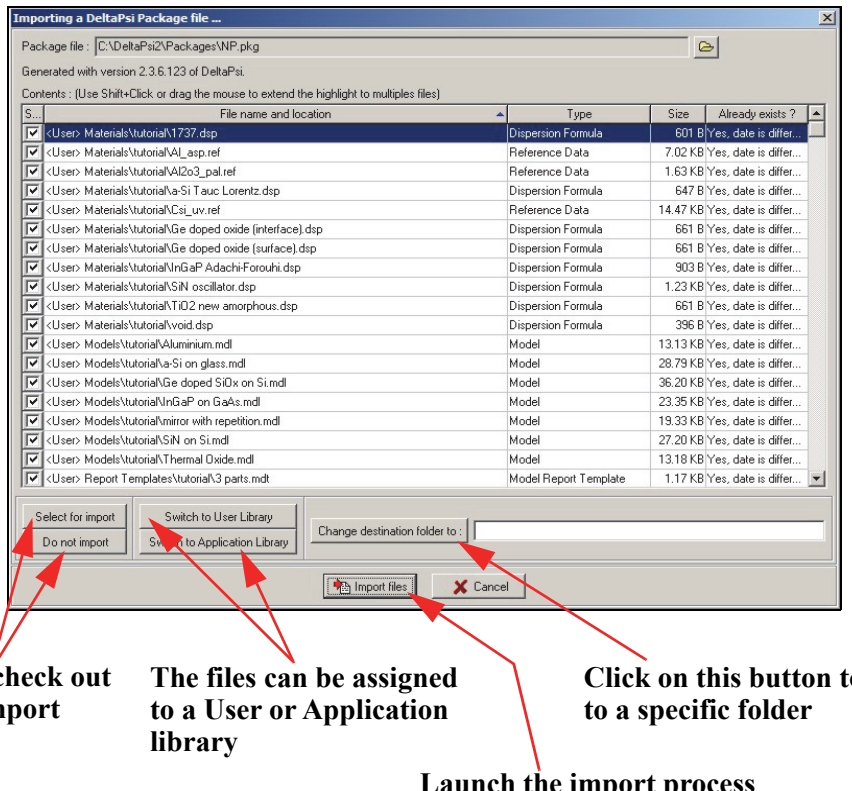



**Figure 2-2**

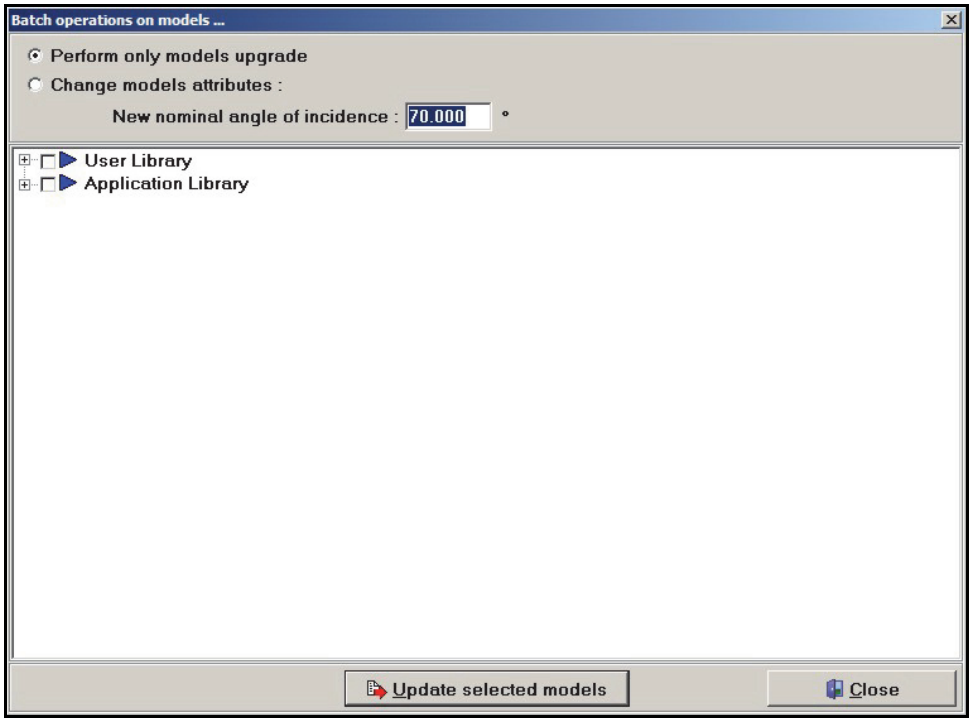
**NOTICE**: Managing seriously the profiles, nodes, files will avoid dead links when running DeltaPsi2.

**Update Models**

To access to this feature, right-click everywhere in the TreeView and select the Update Models choice. Two different processes can be performed from this screen:

Update old models formats to new models format. For example, old models had associated *.spl file. Now every model includes this *.spl information. If the update process is applied on such old format, the spl information will be included to the new model format. The *.spl file will not be deleted, it just will be not used.

Change immediately the angle of incidence (Φ1=Φ2) on one or several models, which will appear in the active Fitting Choice.




From the screen, first select the feature to use. Then select the models on which the update must be performed. If required, change the angle of incidence. Click on the ***Update selected models*** button to validate the choice.

## 2.3 Icon Palette area

Each time a file is opened, an icon shortcut of this file is inserted in the «**Icon Palette**» area.

The **«Icon Palette area»** is located on the bottom of the screen, if the software is running for the first time or if the user has chosen the default layout screen.

# 3 Graphics Screen

Many DeltaPsi2 screens contain a graphical data representation. This graphic interface has many features detailed in the chapter. From this interface, three display modes can be activated according to current needs.

Every simple file (materials, spectroscopic results...) containing curves can be dragged and dropped from the TreeView list to the opened graphics screen area. Right-click on the graphics area to select the useful curves to display.

## 3.1 Standard Display Mode

The Standard Display Mode is the default graphics interface. Figure 3-1, “Standard display mode”, page 35 shows the parameters location.

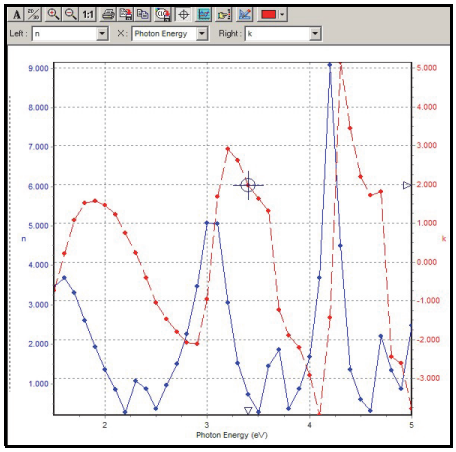
### Text Mode Switch

Click once this icon will switch the *Standard Display Mode* to *Text Display Mode*. This mode is detailed in this chapter.

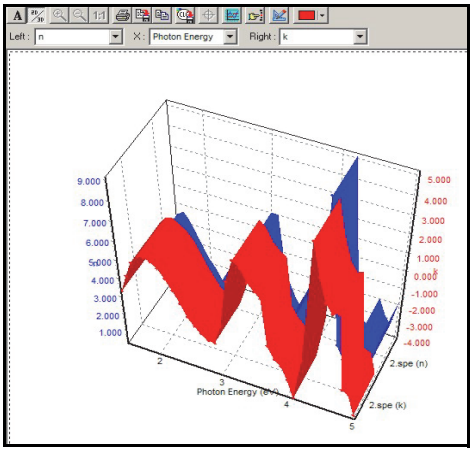
### 2D/3D

This toggle button switches between two graphics modes: 2-dimension and 3-dimension representation.

2D representation




3D representation




### Zoom in

If a portion of a graph is not enough detailed, this portion can be zoomed in. Using the mouse, go to the upper left point of the area to zoom, then press on the mouse left button then drag the mouse to the bottom right point of the area. Release the mouse button: the selected area is immediately zoomed full screen.

To zoom out, the **Zoom out** or the **Full Screen** button can be used.

**Zoom out**

Each click on this button will zoom out the graph. The zoom out progression is performed from the screen center.

**Full Screen**

Click once on this button to display the graph in full screen display.

**Print**

Click on this icon to print the current graph. The printer setup must be previously set from the Windows® operating system. DeltaPsi2 uses the selected ***default*** printer in the operating system.

**Graphics export**

The **Graphics export** button allows the user to save the current screen in a file using universal graphic formats (.WMF or .EMF). Then it becomes possible to import graphics into a Word processing system or Graphics software.

**Copy graph to clipboard**

This command copies the current graph to the active Windows® clipboard. Then, the graph can be retrieved from the clipboard and pasted to any Windows® software like Microsoft Word, Adobe Photoshop etc...

**Save data to calculated file**

This command copies all or part of the spectra of one displayed data, with a selection of observables (even those which are hidden) to a file with the extension **.CLC**. This feature is used for the following purposes:

Create a new material (calculated),

Correct a partially disturbed spectrum (using text editor),

Create a file without any configuration parameters but which can be visualized in DeltaPsi2. This is useful if you want to send graphs to a third party.

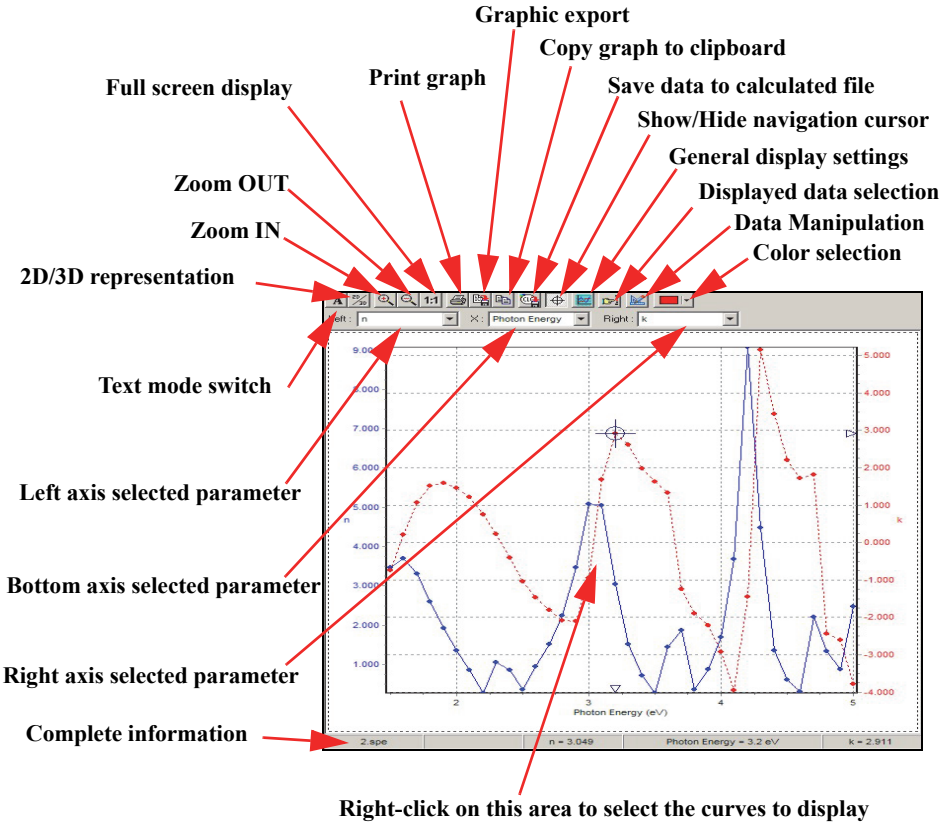



**Figure 3-1 Standard display mode**

### Show/Hide navigation cursor

Click once on this button to activate the cursor on the selected curve. Use the Right/Left keys to move the cursor on the curve. The parameters values of the cursor position are displayed on the bottom part of the graphics screen.

### Graph Setup

The **Graph Setup** feature allows the user to custom the «Graph» presentation. The options are detailed below:

---

**The *General Attributes* tab screen shows the general parameters which can be set: the Graph Title, the font size, the graphic background and cursor of the vertical or/and horizontal grid activation and the data and/or label unit displaying.**

The legend can be also activated and positioned around the graph.

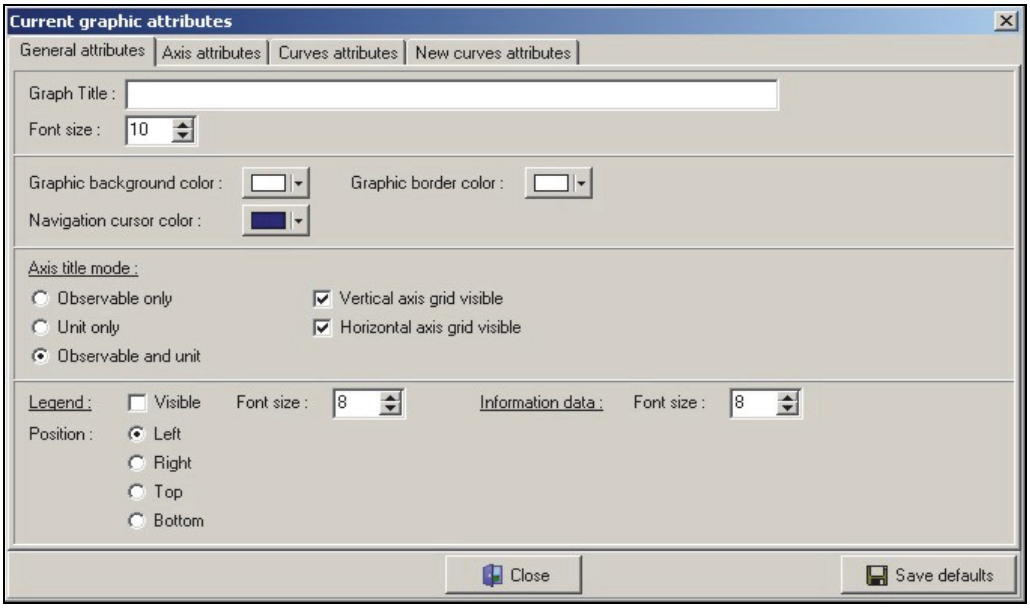



**Figure 3-2 General Attributes**

**The Axis attributes tab contains parameters which could improve the presentation. An example of setup screen is shown in figure 3-3.**

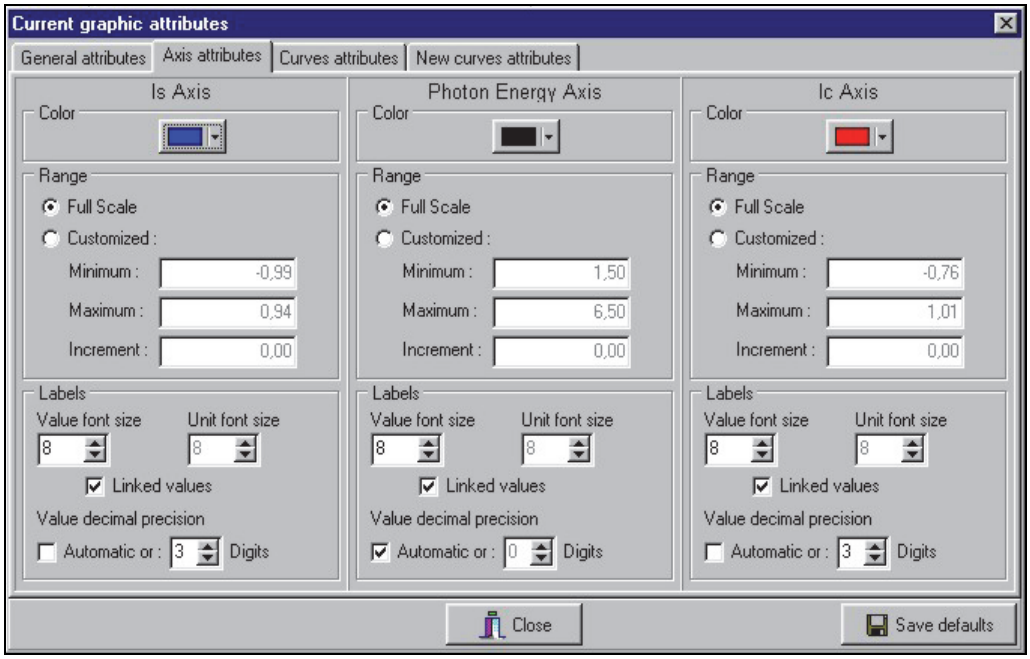



**Figure 3-3 Axis Attributes**

- ***Color***: the default *Color* option can be changed. The selected color is only limited by the graphics card and the Windows® screen parameters.

- ***Full Scale/Customized***: The user has the choice to display all of the axis values («*Full Scale*») or only those which present some interest («*Customized*»). In this case, specific minimum, maximum and interval values can be set to improve the presentation.

- ***Label Font Size/Title Font Size***: The font sizes of each axis «Label» and each axis «Value» can be setup. An additional option can link both values.

- ***Label decimal precision***: The decimal precision can be set on «Automatic» or user defined. The «Automatic» choice will add three decimal values if they exist. To test this feature, try to zoom in a part of the curve, the decimals will appear according to the expected precision.

---

**The Curve Attributes tab includes the displayed setup parameters for each graph. The left hand side of the screen lists the loaded spectra. Select one of them and apply changes described below.**

**NOTICE**: The changes applied in this screen are only valid to the currently displayed graphs. If you wish to permanently apply a set of colors, lines etc.. read the next chapter carefully: New Curve Attributes.

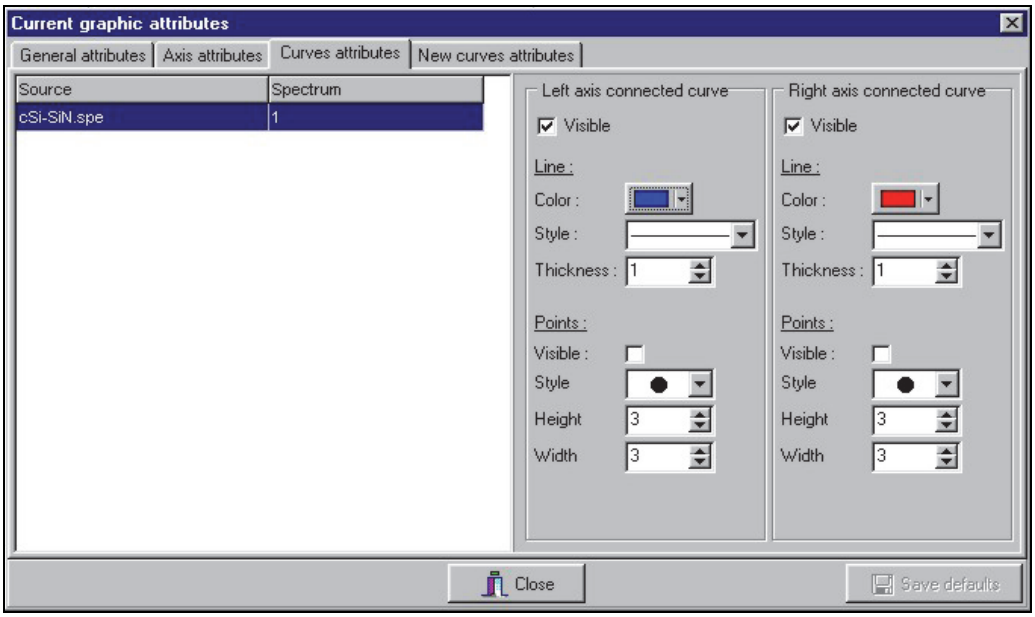



**Figure 3-4 Curve attributes**

- ***Visible***: deselecting this option will remove the graph from the display screen.The ***Line*** parameters allow the user to choose the visual characteristics for each graph: color, style, thickness.

- The ***Points*** parameters can activate the data points display as well as the visual characteristics for these data points.

---

**The New Curves Attributes tab is an important screen as it includes powerful features related to the graphs attributes of the entire DeltaPsi2 Software.**

Unlike the tabs described in this section, the ***New Curves attributes*** screen is not «real time» and does not automatically refreshed the changes.

This screen will be used for the following purposes:

Change the graphs and data points attributed to each files group (***Material***, ***Experimental curve*** and ***Simulation curve***). These changes can be momentarily applied if new graphs will be added on the currently opened graphs or permanently if you choose to save the changes using the ***Save to profile..*** or ***Save defaults*** buttons.

The ***Color*** attributes follow the same philosophy: new selected color will be applied only if new graphs are retrieved to the current graphs screen. The selected colors will be permanently used if you choose ***Save defaults*** option or saved as a profile if the ***Save to profile..*** option is selected.

IMPORTANT: the changes saved through the option «***Save to profile***» are valid only for the files belonging to the same group.

Groups reminder:

- **Material: Alloy (*.ALY), Dispersion (*.DSP), Reference (*.REF), User Defined (*.UDF),**
- **Simulation: Models (*.MDL), Modeling scenario (*.MLS), Bound Multimodel (*.MMD), Serial Multimodel (*.MMD),**
- **Experimental: Spectroscopic acquisition (*.SPE), Kinetic acquisition (*.KIN), Calculated file (*.CLC).**

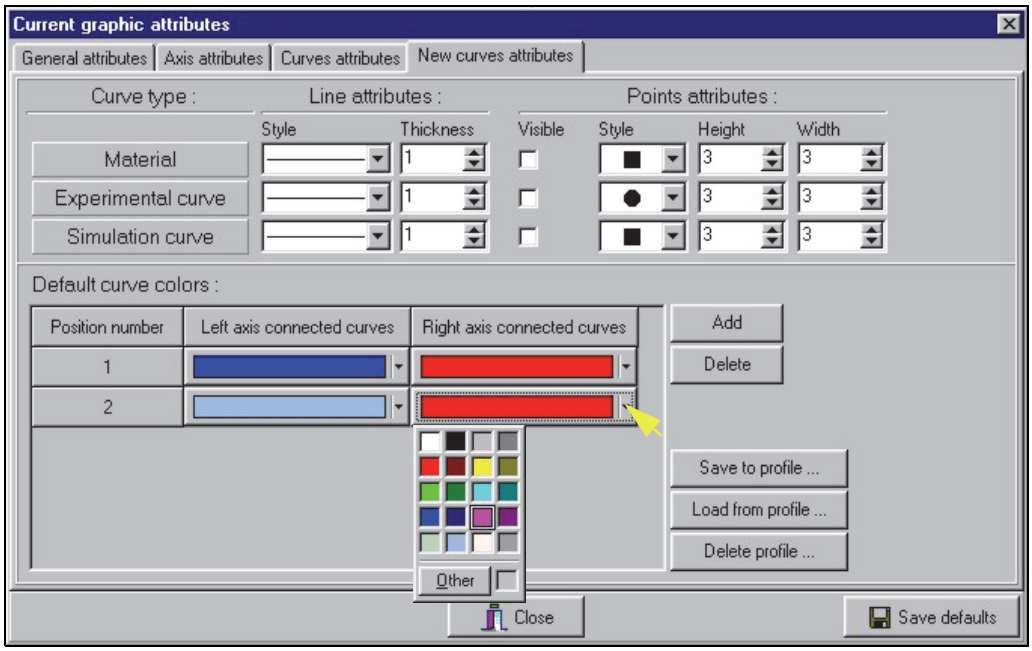
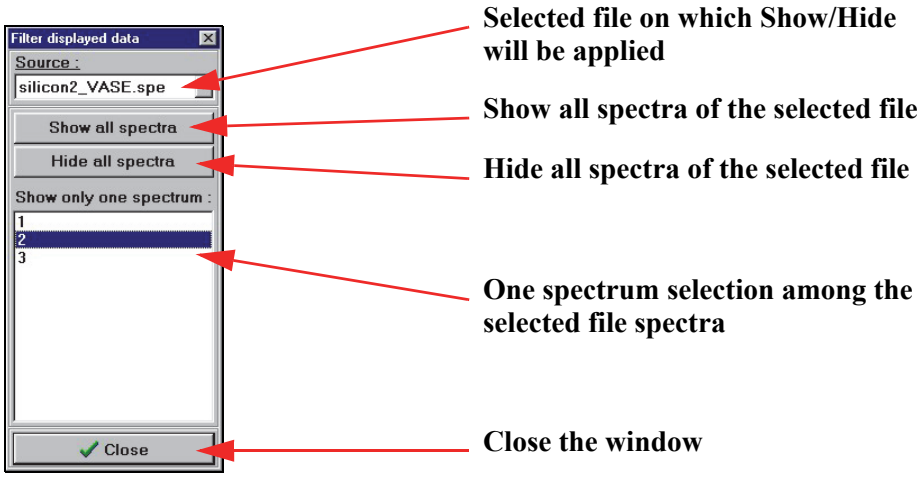



**Figure 3-5 New Curves Attributes**

### Displayed data selection

The **Displayed data selection** feature allows the user to deselect graphs or spectrum from the display screen. Click once on the button to display the following section screen:

05/20/2016




On the above **Displayed data selection,** at least one file must be displayed. Then, it is impossible to remove all the spectra from the display screen.

### Color selection

The ***Color selection*** button is used to change color on one graph. The procedure is described in the figure 3-6 below.

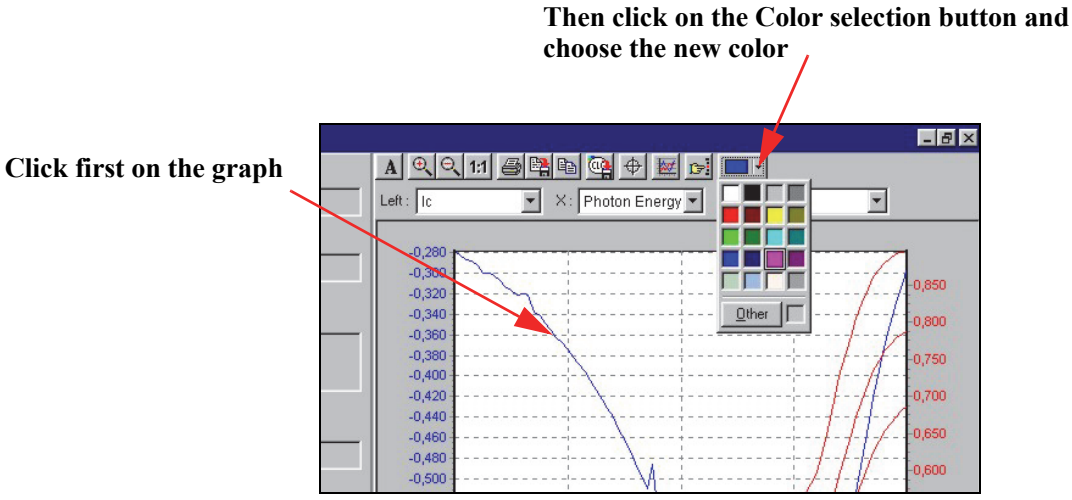



**Figure 3-6 Color selection**

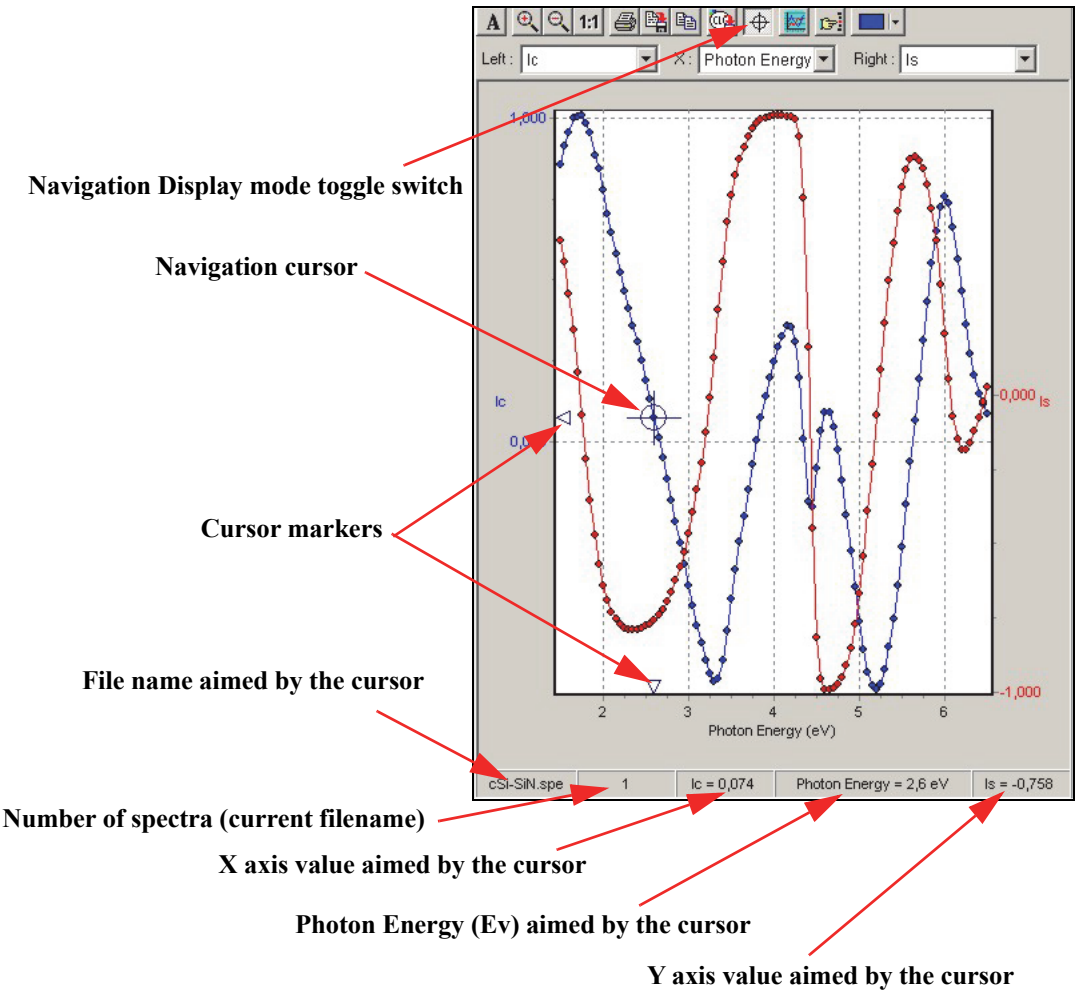
## 3.1.1 Data Manipulation feature

The Data Manipulation is a powerful feature which allows a user to modify spectra. The chapter "Data Manipulation" on page 117 describes all of the parameters and resulted effects.

## 3.2 Navigation Display Mode

This screen is the same as the ***Standard Display Mode*** except for the navigation cursor. To activate/deactivate this mode, click on the  icon. The navigation screen will be then displayed.

Click on a curve then move the cursor using the left/right keys or left-click the mouse button directly on the curve.




## 3.3 Text Display Mode

The ***Text Display Mode*** allows the user to visualize the data in text mode. To activate this mode, click on the  icon.

To activate back the ***Standard Display Mode***, click on the  icon.

Figure 3-7 shows the ***Text Display Mode*** screen.

05/20/2016

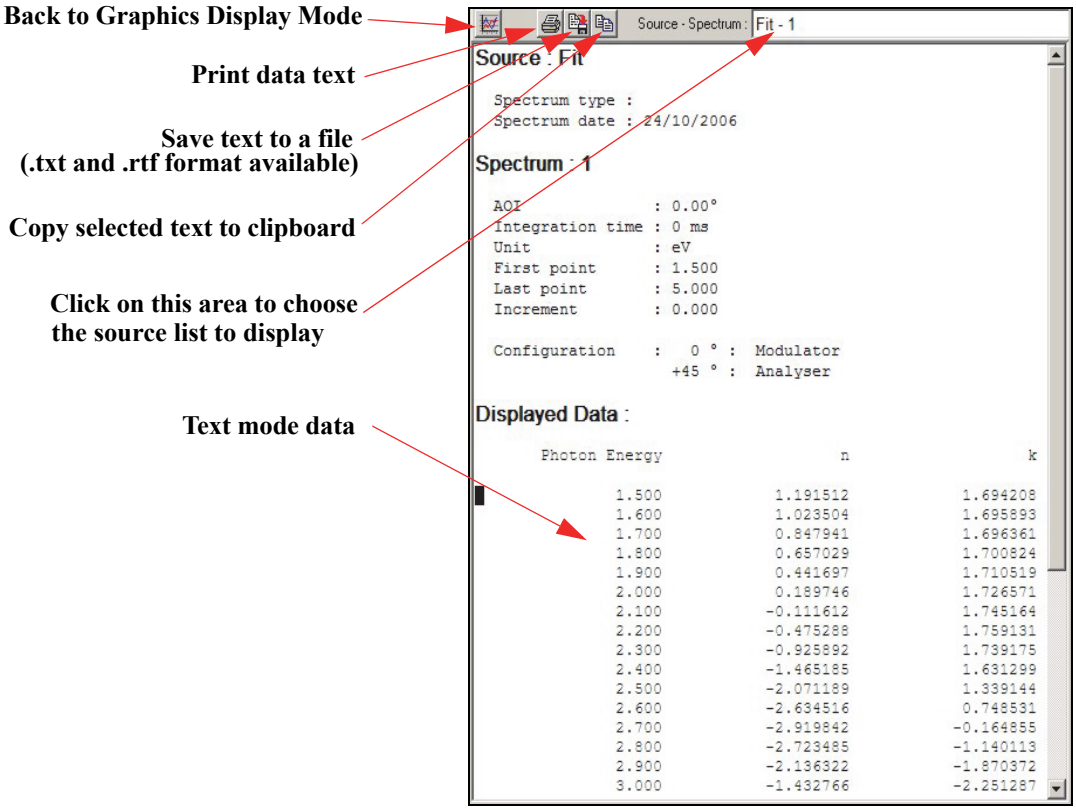



**Figure 3-7 Text Display Mode**

The numerical values are displayed in text with however the following restriction:

- **One curve, at a time, can be taken into account to display the numerical values,**
- **Only the data related to the selected displayed parameters (see Figure 3-1, "Standard display mode", page 35) from the *Standard Display Mode* screen are shown.**

As shown on figure 3-7, the buttons located on the top of the screen perform the printing, saving to file and copying to clipboard features.

The displayed data can be immediately switched between the loaded sources using the ***Source*** field.

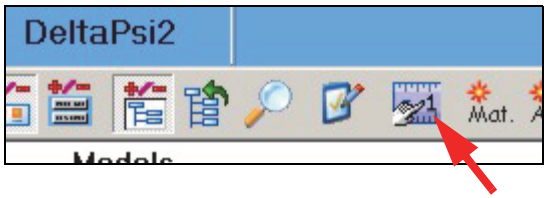
**TIP**: If you switch from the ***Navigation Display Mode*** to the ***Text Display Mode***, the selected data (cursor position) report the related numerical value (highlighted line).

# 4 Acquisition

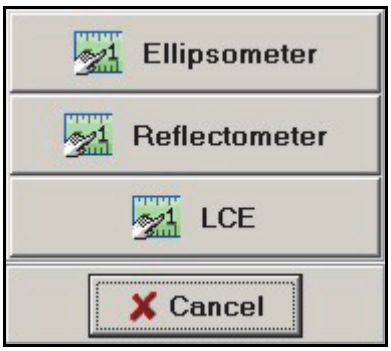
Acquisition is an important part of the DeltaPsi Software. This chapter concerns the data acquired during one of the following acquisition modes: the ***Views*** real time acquisition panel and the ***Spectroscopic*** and ***Multiwavelength*** acquisition processing.

## 4.1 Accessing

Double-click on the ***Manual Measurement***  icon from the ***Button Bar*** menu.




If several applications are available, the following screen will be displayed:




**Select the application to load**

**According to the chosen application, the use, the measured parameters and the results should be very different.**

## 4.2 Views for UVISEL Ellipsometer

**For UVISEL2 System, please see the chapter "Views and Setup for UVISEL2 System" on page 65.**

The following real time screen ***Views*** is displayed when the UVISEL Ellipsometer System is chosen:

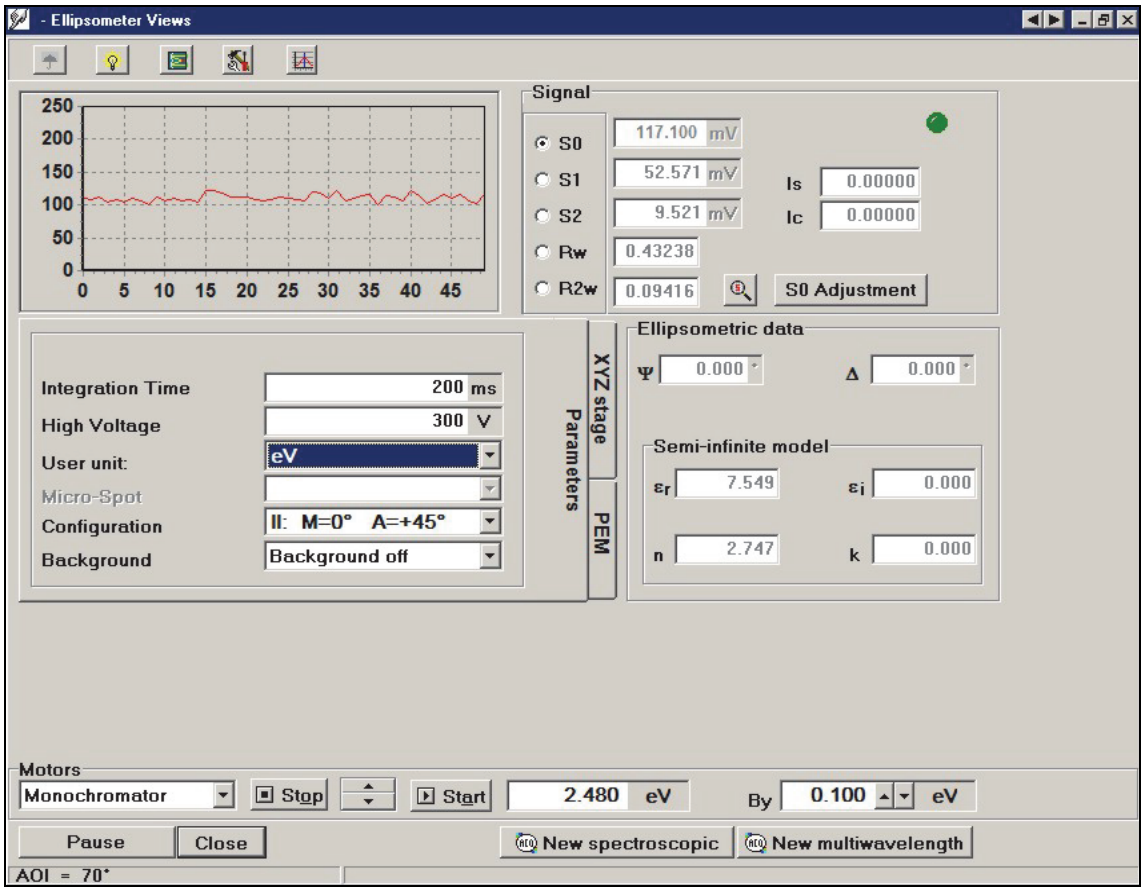



**Figure 4-1 Views screen**

The ***Views*** screen allows the user to setup acquisition parameters and control the real time measured parameters.

The screen is organized by operating sections described below:

- **Top bar buttons,**
- **Real Time Graphics Screen,**
- **Measured signal selection panel,**
- **Acquisition parameters setup,**
- **Ellipsometric data setup,**
- **Motors Control panel.**

### 4.2.1 Top Bar Buttons

The **Top Bar Buttons** is located on the upper side of the ***Views*** screen.

(*UV Filter icon*) Activates/de-activates UV filter which protects the sample from damaging UV light. The filter is a heavy side filter blocking wavelengths below 400nm.

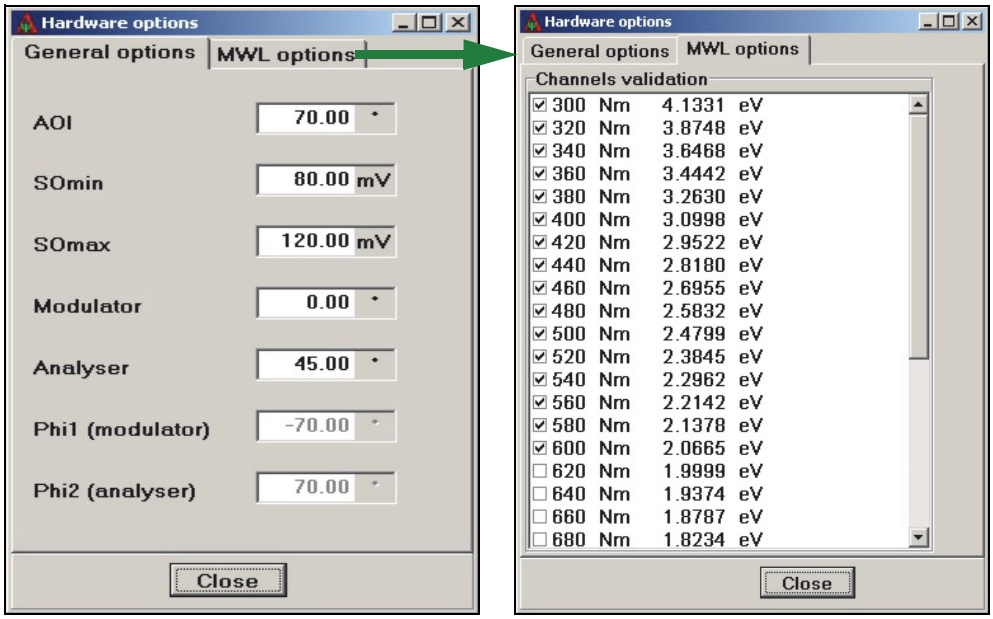
(*Shutter icon*) This toggle button controls the **shutter** located on the exit path of the Light Source sub-unit.

**(***Monoch/MWL icon***) Monochromator/Multiwavelength** toggle button will select the monochromator or polychromator (MWL)

OR

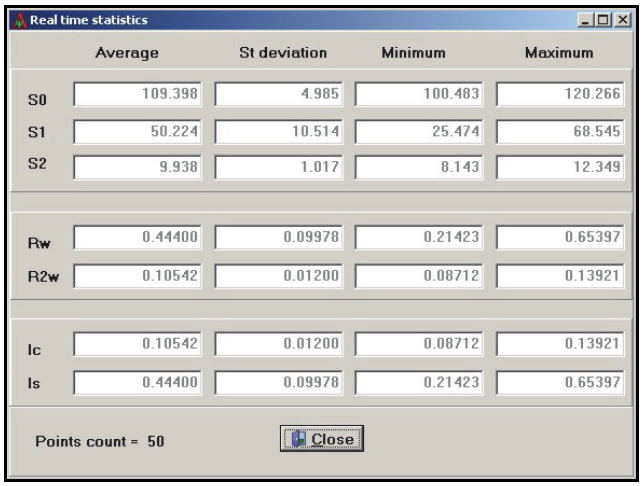
*FUV/iHR320 (or HR460)* will select the UV monochromator (FUV200) or the IR monochromator. See your owners documentation for applicable spectral ranges.

Click on this button to activate the following ***Hardware Options*** screen:



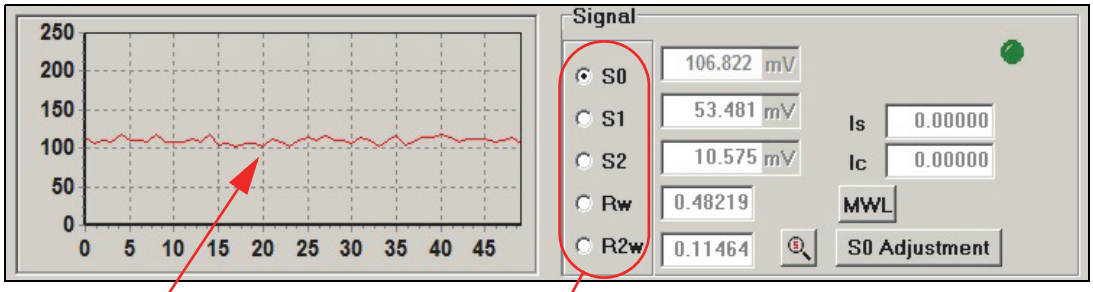
- The ***AOI*** (Angle of incidence) parameters informs the user about the real incidence angle of the ellipsometer.
- The user can choose a finite S0 range (minimum and maximum) in which the High Voltage value is adjusted to achieve minimum<S0<maximum. Function is performed when the «S0 Adjustment» button is pressed in the Views screen (See "Real Time Graphics Screen" on page 46). High Voltage is applied to the Photomultiplier tube (PMT).

Click on this button to visualize the measurement statistics. The following screen shows an example of such a screen.




The DeltaPsi2 Software can control, as shown above with the MWL (Multi-WaveLength) sub-unit, many different peripherals which are options to the system. If one or several options are not included to the system, the command button will be deactivated and will be shown graphically in greyed. If the MWL sub-unit is not delivered with the System, the MWL tab will not be displayed.

### 4.2.2 Real Time Graphics Screen




**Selected signal display (here S0 is displayed)**

**Selected signal to display**

**Numerical real time parameters display**

**This icon will display the S0, Rω, R2ω using large digits**

**S0 Adjustment** **Click on this button to calibrate S0 offset**

**MWL** **This button is displayed only if MWL is installed. Click on this button to calibrate all the channels**

**Figure 4-2 Real Time Graphics Display**

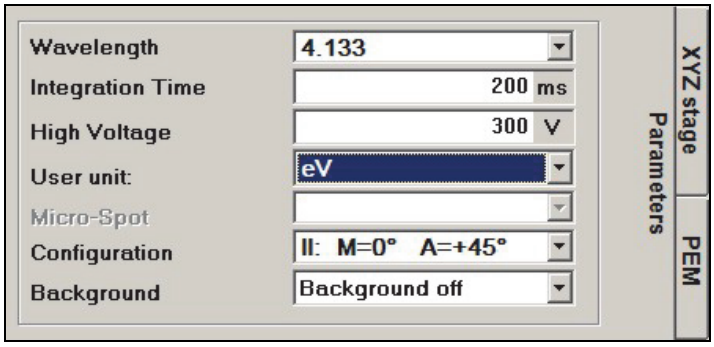
| | |
|---|---|
| **S0** | DC signal measurement |
| **S1** | First harmonic level measurement (50 kHz) |
| **S2** | Second harmonic level measurement (100 kHz) |
| **Rω**<br>**R2ω** | Normalized measurements<br>J1 = first harmonic of the Bessel function<br>J2 = second harmonic of the Bessel function |
| **Is and Ic** | Variables which are used in the ellipsometric general equations. |

### 4.2.3 Acquisition parameters setup

This part is composed of two-tab screen. The first tab concerns the acquisition parameters, the second, the XYZ table and hardware extensions setup.

*05/20/2016*

- Acquisition Parameters..




Enter the appropriate parameters related to the acquisition. The parameters are detailed below:

| | |
|---|---|
| **Wavelength (if MWL available)** | This parameter is only available if the MWL is installed and selected with the icon button  . Each wavelength of the MWL can be selected and parameters below set. |
| **Integration Time** | Enter the Integration Time value. The integration time is the amount of time allowed for the signal to be acquired for each measurement point. In general, select the shortest integration time that will give sufficient S/N for determining essential spectral features |
| **High Voltage** | Enter the High Voltage value to be applied to the Photomultiplier Tube(s) (PMT). |
| **User Unit** | Enter the user working unit. The choices are **eV, cm-1, nm, A, µm** |
| **MicroSpot (If MicroSpot avail.)** | The beam diameter can be set on the **Micro-Spot** sub-unit. The choices are 1200, 250, 120 and 60 µm. |

| | |
|---|---|
| **Configuration** | Select one of the preprogrammed configurations. These configurations are chosen from the Acquisition Parameters screen located in the Software Option section (see "Observables Setup" on page 17)<br>**Test position:**<br>**M=0° A=0°**<br>**Configuration II:**<br>**M=0° A=+45°**<br>**M=0° A=-45°**<br>**M=90° A=+45°**<br>**M=90° A= -45°**<br>**Configuration III:**<br>**M=+45° A=+45°**<br>**M=+45° A=-45°**<br>**M=-45° A=+45°**<br>**M=-45° A=-45°**<br>**Other:**<br>**M=-45° A=0°**<br>**M=+45° A=90°** |
| **Background** | **Background off:** performs a noise background measurement before an acquisition. This noise background will be subtracted from each data point acquisition.<br>**Background on:**<br>If the Background box is checked, for each data point, two acquisitions will occur. The system will close the shutter located on the light source path and perform an acquisition. A signal plus dark acquisition will be performed with the shutter open. The dark acquisition will be subtracted from the signal plus the dark acquisition. This technique minimizes the effect of any drift conditions that may occur.<br>**Automatic Background**<br>If the Offsets calibration has been performed, a results file (background.ini) has been saved on the hard disk. If an acquisition is launched using this mode, the [$S0_{\text{background noise}}$ (HV)] will be subtracted from the acquisition data. |

- **XYZ Stage and hardware extensions Setup**

Click on the ***XYZ Stage*** tab to display the control panel of the motorized XYZ stage. Of course, this screen will be active only if the System includes a motorized Stage

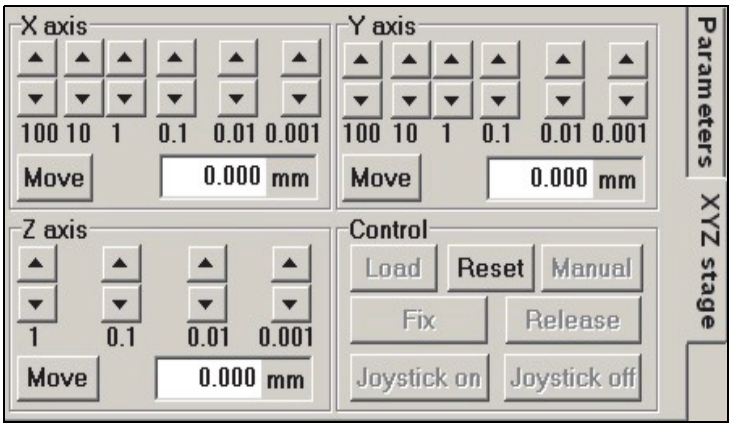



**Figure 4-3 Stage control panel**

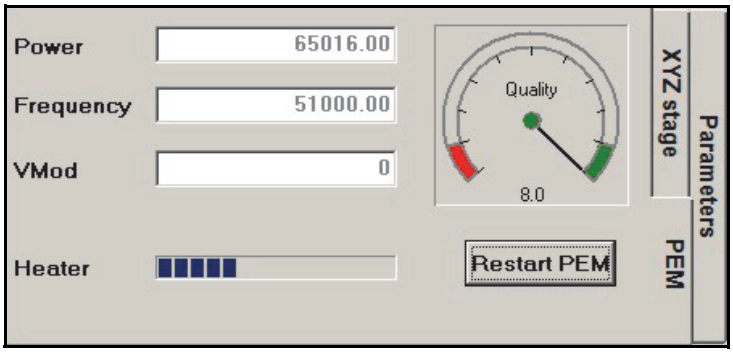
Each axis can be precisely set using decimal buttons or by entering the value in millimeter with the keyboard. The ***Move*** button validates the value.

The ***Control*** section includes additional hardware features:

- ***Load/Reset/Manual***: these buttons are active only with specific motorized stages,
- ***Fix/Release***: activate/deactivate the sample vacuum circulation process which fix or release the sample on the stage (if available on the System),
- ***Joystick ON/OFF***: activate/deactivate the joystick control instead of the Software control, described above (if available on the System).
- **PEM setup**

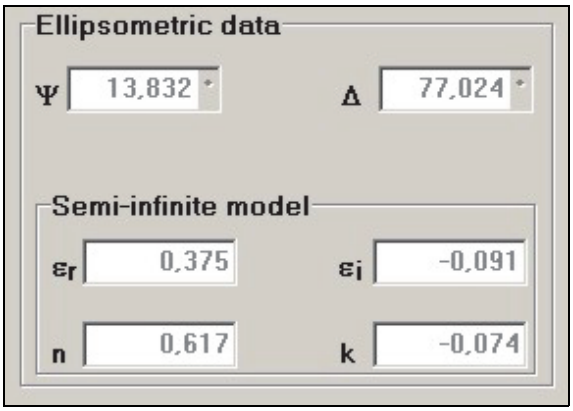
The PEM control panel shows the Modulator part status. This panel will be mainly used by the Service Staff. The ***Restart PEM*** button performs the Modulator reactivation if the Modulator electronic board has lost control on it.

The analog indicator shows the PEM status. If the PEM works fine, the indicator stays on the green range, with some very short variations to red.



### 4.2.4 Ellipsometric Data

The ***Ellipsometric data*** section of the **Views** screen is only for informational purpose.




| | |
|---|---|
| **Ψ and Δ** | Calculated values based on the configuration |
| **$\varepsilon_r$**<br>**$\varepsilon_i$**<br>**n**<br>**k** | Calculation for a semi-infinite model. |

### 4.2.5 Motors control panel

The **Motors control panel** allows the user to control each of the System Motors.

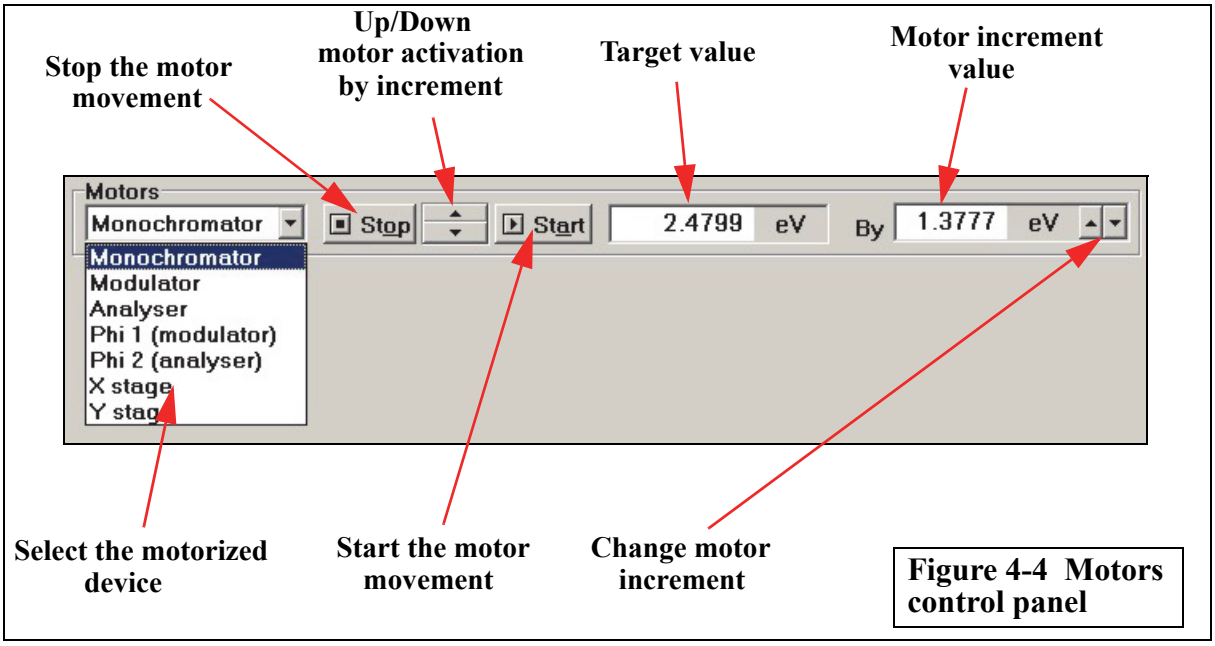
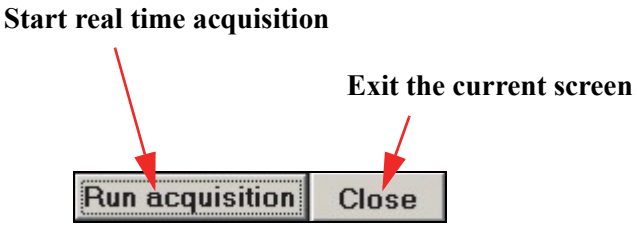



**Figure 4-4 Motors control panel**

Figure 4-4 and the table below show how to use the Motors control panel.

| | |
|---|---|
| **Motors** | Select the motorized device to move. Only the current devices of the System are listed. |
| **Start** | Start the motor to the entered target value. In this case, the motor will use the entered increment value. The target and increment values fields are located on the right hand. |
| **Up/Down buttons** | The Up/Down buttons will move the selected motor by one increment. The increment value is the value entered in the field located on the right hand side. |
| **Stop** | Stop the motor movement. |
| **Target value** | Enter in this field the target value for the selected motor. |
| **Motor increment value** | Enter the motor increment value. |
| **Motor increment buttons** | The Motor Increment buttons increase or decrease the increment value by one elementary programmed increment. |

### 4.2.6 Running real time acquisition

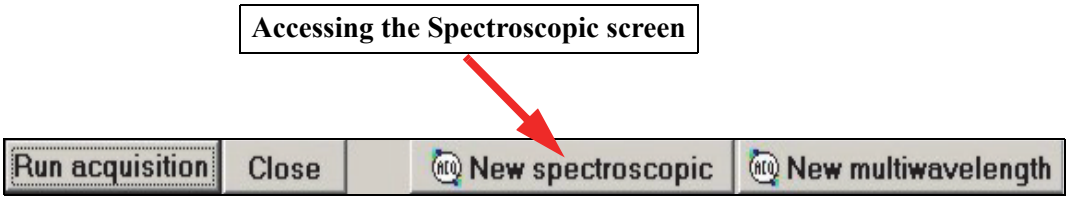



## 4.3 Spectroscopic Acquisition for UVISEL Ellipsometer

The Spectroscopic acquisition is the most standard mode. It usually concerns acquisitions which are performed on a specific wavelength range with a monochromator.

### 4.3.1 Accessing

Click on the ***New Spectroscopic*** button from the **Views** screen.




### 4.3.2 Spectroscopic acquisition setup

The following ***Spectroscopic Acquisition Setup*** screen will be then displayed:

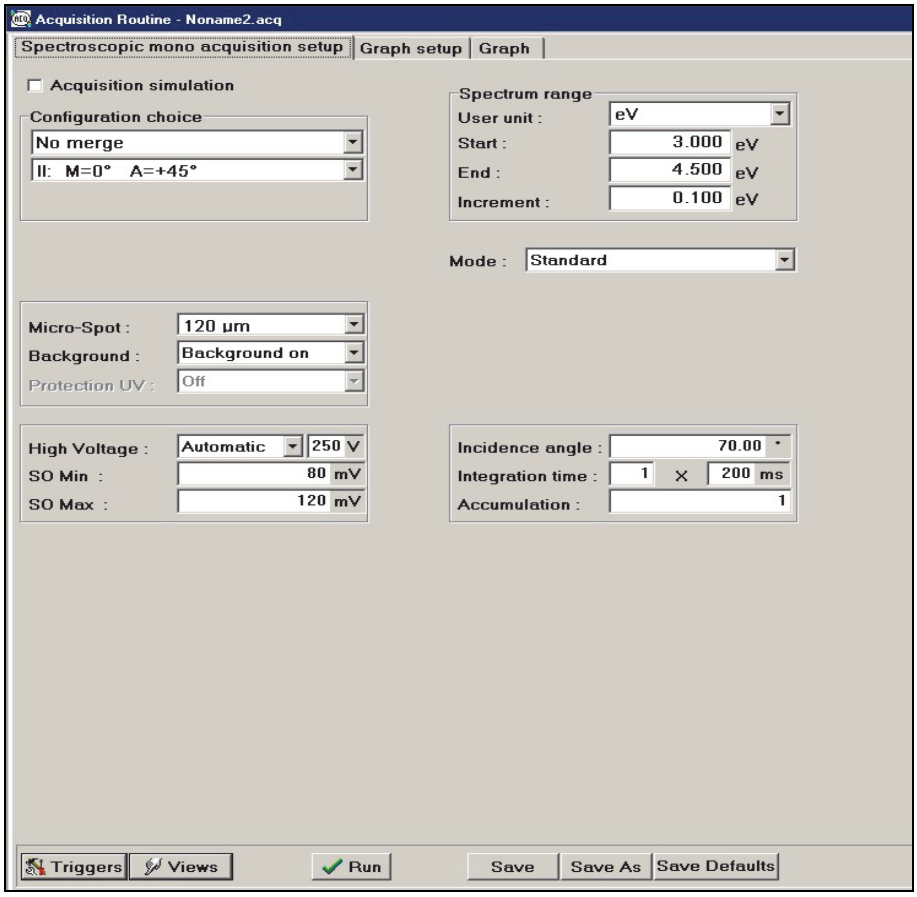



**Figure 4-5 Spectroscopic acquisition setup**

Before performing an acquisition process, a routine must be created. Each of the parameters is detailed below:

| | |
|---|---|
| **Routine Name** | The Routine name is displayed on the top blue header. By default, the «NonameX» is displayed. To rename the routine, click on the ***Save As*** button and enter the new name. |
| **Acquisition Simulation** | Checking this option will inhibit access to external hardware. |
| **Configuration choice** | Choosing a Merge method is useful to increase the precision on the Psi and Delta measurements according to the sample characteristics.<br>Additional choices are added to chosen method.<br>**No merge:** Psi and Delta are calculated from one acquired spectrum,<br>**Standard merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained in the following manner: «Delta» curve is taken from the Configuration II scan. «Psi» curve is taken from the Configuration III,<br>**High Accuracy merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained in the following manner: «Delta» curve is taken from the Configuration II scan. «Psi» curve is calculated by taking the Is and Ic from configuration III and «Delta» from configuration II with the following formula: Tg(2 * Psi) = IsII / [IcIII * Sin(DeltaII)],<br>**Smart merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained in the following manner: «Delta» curve is taken from the Configuration II scan. «Psi» curve is composed from several segments. The limits of segments are determined by some specified «Psi» threshold value crossed by the «Psi» curve (for example 40°). The segments with «Psi» less than this threshold value are taken from the Configuration II, if the values are over, they are taken from Configuration III. |
| **MicroSpot** | The beam diameter can be set on the **Micro-Spot** sub-unit. The choices are 1200, 250, 120 and 60 µm. |

| | |
|---|---|
| **Background** | A high background level can mask weak features and reduce the dynamic range of the Real Time Control.<br><br>**Before applying the «background» feature, be sure that the offsets calibration procedure has been performed. This calibration should be performed at least once a year or in case of important changes of the system, e.g. PMT replacement.**<br>**Background off:** performs a noise background measurement before an acquisition. This noise background will be subtracted from each data point acquisition.<br>**Background on:**<br>If the Background box is checked, for each data point, two acquisitions will occur. The system will close the shutter located on the light source path and perform an acquisition. A signal plus a dark acquisition will be performed with the shutter open. The dark acquisition will be subtracted from the signal plus the dark acquisition. This technique minimizes the effect of any drift conditions that may occur.<br>**Automatic Background:**<br>If the Offsets calibration has been performed, a results file (background.ini) has been saved on the hard disk. If an acquisition is launched using this mode, the [$S0_{background\ noise}$ (HV)] will be subtracted from the acquisition data. |
| **Protection UV** | Choice is ON or OFF<br>**ON:** activates a filter to protect the UV sensitive samples. |
| **High Voltage** | The user has the choice between two High Voltage modes:<br>• **Fixed**<br>Enter the fixed high voltage value in the related field (located on the right). During the acquisition, the high voltage value will not be changed.<br>• **Automatic**<br>The software performs an automatic feedback control on this relation: HV = f(S0) using a calibrated HV curve and the S0 measured value. |
| **S0 Min / Max** | Choose a S0 range (minimum and maximum) in which the High Voltage does not change. |
| **User unit** | Enter the user working unit. The choices are **eV, cm-1, nm, A, µm.** |

*05/20/2016*

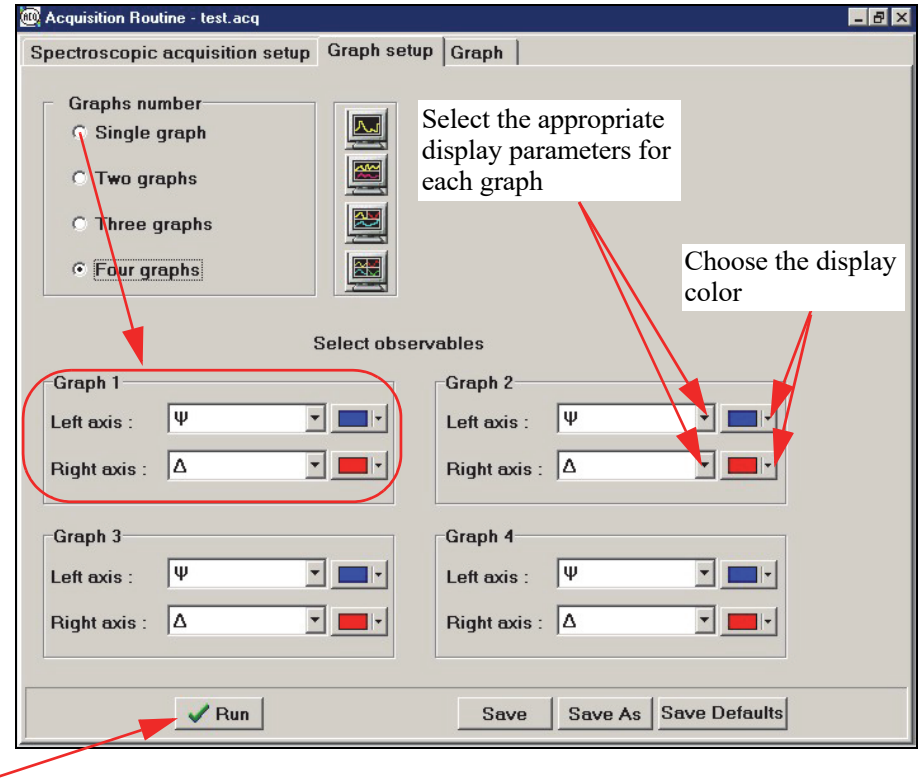
| | |
|---|---|
| **Spectrum Range** | Enter the wavelength limits and increment step of the monochromator acquisition. |
| **Mode** | **Standard:** The standard spectroscopic acquisition mode will be performed.<br>**Variable angle:** one spectroscopic acquisition will be performed for each PHI angle. Obviously, the «Modulator» and «Analyzer» angles change at the same time by the same angle increment.<br>The user can set the Start, End and Step values. If the Phi (Analyzer) adjustment box is checked, the Analyzer head will automatically move up and down to find the maximum strength of the reflected beam. This optimization process is performed for each new angle. |
| **Incidence angle** | Enter the incidence angle: angle determined between the Modulator, the sample and the Analyzer.<br>For information:<br>- Measurements for most semiconductors are usually made around 70°-75°,<br>- Measurements on transparent materials are made at around 60°. |
| **Integration Time** | Enter the Integration Time value. The integration time is the amount of time allowed for the signal to be acquired for each measurement point. In general, select the shortest integration time that will give sufficient S/N for determining essential spectral features. |
| **Accumulation** | The signal/noise ratio may be improved by performing more accumulations per spectrum, where each accumulation is an individual integration time. Again, for *n* number of accumulations, the signal/noise ratio will improve by the *square root of n*. The trade-off is that the experiment takes *n* times longer. |

Click **RUN** to perform the acquisition. The results are saved in a file. This file is located in the **Acquisition Data** folder. Select this folder then double-click on the acquisition file. The results will be then displayed.

NOTICE: The Triggers feature will be available soon but is not yet active.

### 4.3.3 Graph Setup

***Graph Setup*** tab allows the user to setup the acquisition graphics presentation.




### 4.3.4 Graph

If the **RUN** button is activated from the ***Spectroscopic acquisition setup*** or the ***Graph setup*** screen, an intermediate screen is then displayed which allows the user to fill in various information fields which generate the file name (see figure 4-6). Click on the OK button to start the acquisition. The ***Graph*** tab is automatically switched to display acquisition results.

Here below is an example:

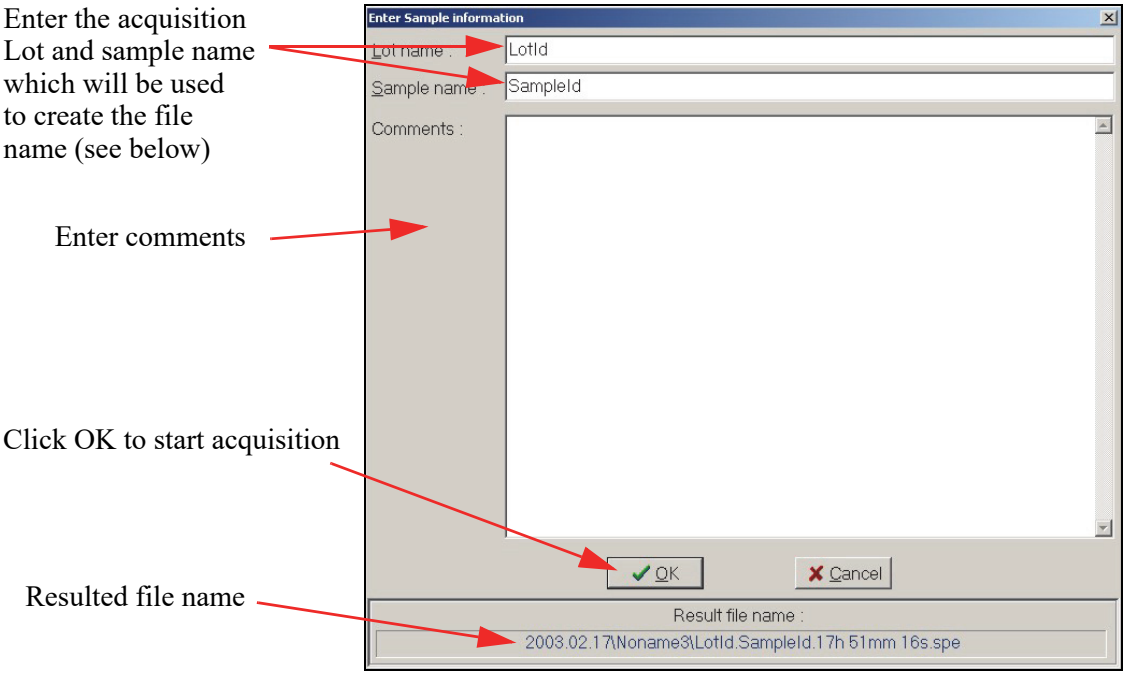



**Figure 4-6 Information screen**

Here below is an example of acquired spectra if four graphs are chosen:

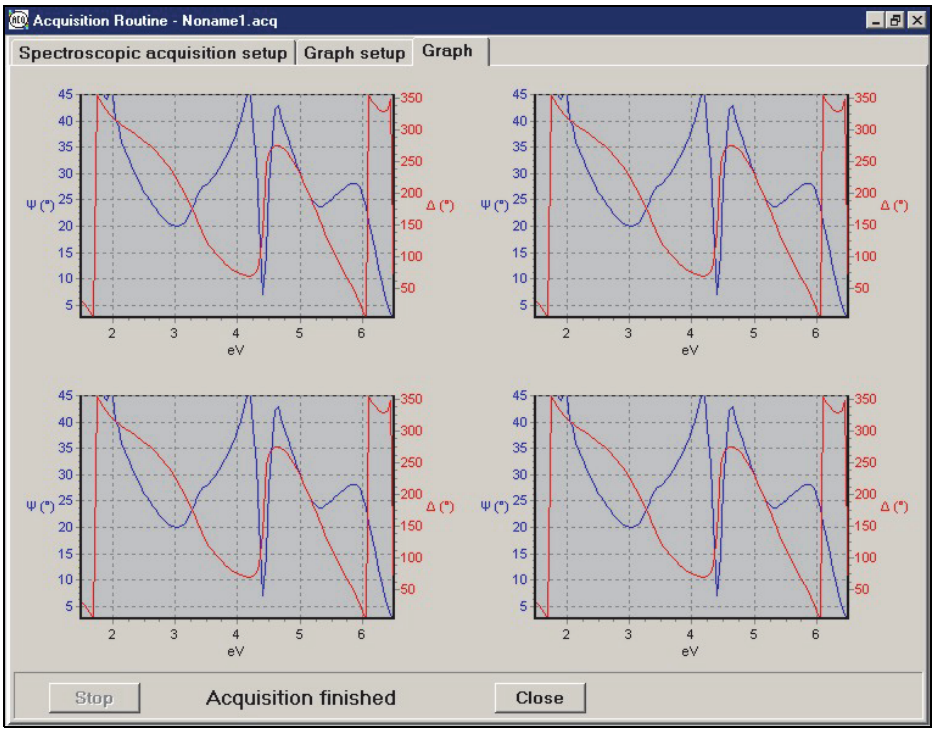


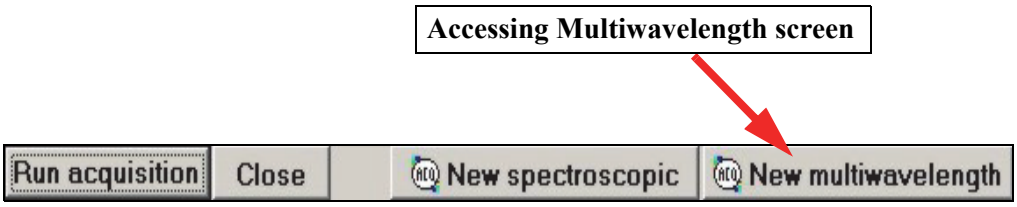

**Figure 4-7 Graph screen**

# 4.4 Multiwavelength Acquisition for UVISEL Ellipsometer

The Multichannel Acquisition mode is also called Multiwavelength. This acquisition mode uses a spectrograph which includes 32 photomultiplier tubes. If the Spectrograph includes a motorized wavelength scanning device, a multiple (up to 5x) of the elementary amount of points could be analyzed. The full range on a System could be able to measure up to 160 wavelengths points. The result is a very fast spectrum acquisition.

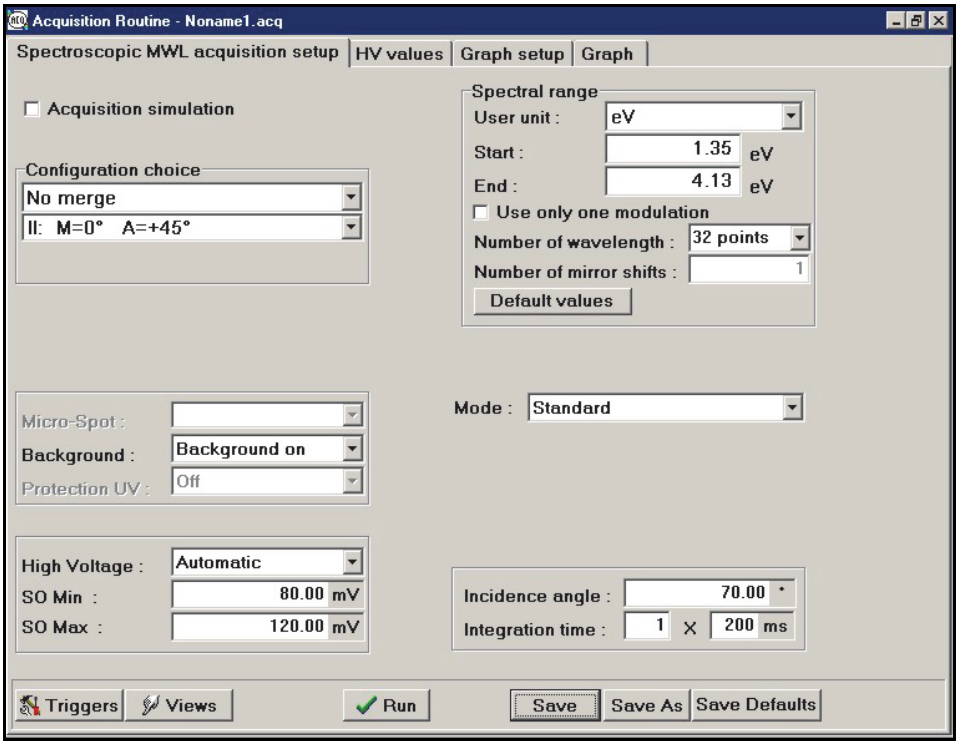
## 4.4.1 Accessing

Click on the ***New Multiwavelength*** button from the **Views** screen.




## 4.4.2 MWL Acquisition Setup

The following ***MWL Acquisition Setup*** screen will be displayed:




Most of the parameters are the same as those found in the Spectroscopic Acquisition Setup.

Before performing an acquisition process, a routine must be created. Each of the parameters are detailed below:

| | |
|---|---|
| **Routine Name** | By default, the «NonameX» is displayed. To rename the routine, click on the ***Save As*** button and enter the new name. |
| **Incidence angle** | Enter the incidence angle: angle determined between the Modulator, the sample and the Analyzer.<br>For information:<br>- Measurements for most semiconductors are usually made around 70°-75°,<br>- Measurements on transparent materials are made around 60°. |
| **User unit** | Enter the user working unit. The choices are **eV, cm-1, nm, A, µm.** |
| **MicroSpot** | The beam diameter can be set on the **Micro-Spot** sub-unit. The choices are 1200, 250, 120 and 60 µm. |
| **Background** | A high background level can mask weak features and reduce the dynamic range of the Real Time Control.<br>**Before applying the «background» feature, be sure that the offsets calibration procedure has been performed. This calibration should be performed at least once a year or in case of important changes of the system, e.g. PMT replacement.**<br>**Background off:** performs a noise background measurement before an acquisition. This noise background will be subtracted from each data point acquisition.<br>**Background on:**<br>If the Background box is checked, for each data point, two acquisitions will occur. The system will close the shutter located on the light source path and will perform an acquisition. A signal plus dark acquisition will be performed with the shutter open. The dark acquisition will be subtracted from the signal plus the dark acquisition. This technique minimizes the effect of any drift conditions that may occur.<br>**Automatic Background:**<br>If the Offsets calibration has been performed, a results file (background.ini) has been saved on the hard disk. If an acquisition is launched using this mode, the [$S0_{background\ noise}$ (HV)] will be subtracted from the acquisition data. |

| | |
|---|---|
| **Protection UV** | Choice is ON or OFF<br>**ON:** activates a filter to protect the UV sensitive samples. |
| **Accumulation** | The signal/noise ratio may be improved by performing more accumulations per spectrum, where each accumulation is an individual integration time. Again, for *n* number of accumulations, the signal/noise ratio will improve by the *square root of n*. The trade-off is that the experiment takes *n* times longer. |
| **Integration Time** | Enter the Integration Time value. The integration time is the amount of time allowed for the signal to be acquired for each measurement point. In general, select the shortest integration time that will give sufficient S/N for determining essential spectral features. |
| **High Voltage** | The user has the choice between two High Voltage modes:<br>• **Fixed**<br>Enter the fixed high voltage value in related field (located on the right). During the acquisition, the high voltage value will not be changed.<br>• **Automatic**<br>The software performs an automatic feedback control on this relation: HV = f(S0) using a calibrated HV curve and the S0 measured value. |
| **S0 Min / Max** | Choose a S0 range (minimum and maximum) in which the High Voltage does not change. |

| | |
|---|---|
| **Configuration choice** | Choosing a Merge method is useful to increase the precision on the Psi and Delta measurements according to the sample characteristics.<br>Additional choices are added to the chosen method.<br>**No merge:** Psi and Delta are calculated from one acquired spectrum,<br>**Standard merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second is done with Configuration III. The final spectrum is obtained in the following manner: Delta curve is taken from the Configuration II scan. Psi curve is taken from the Configuration III.<br>**High Accuracy merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained in the following manner: «Delta» curve is taken from the Configuration II scan. «Psi» curve is calculated by taking the Is and Ic from configuration III and Delta from configuration II with the following formula: Tg(2 * Psi) = IsII / [IcIII * Sin(DeltaII)],<br>**Smart merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained like following: «Delta» curve is taken from the Configuration II scan. «Psi» curve is composed from several segments. The limits of segments are determined by some specified «Psi» threshold value crossed by the «Psi» curve (for example 40°). The segments with «Psi» less than this threshold value are taken from the Configuration II, if the values are over, they are taken from the Configuration III. |
| **Mode** | **Standard:** The standard spectroscopic acquisition mode will be performed.<br>**Variable angle:** one spectroscopic acquisition will be performed for each PHI angle. Obviously, the «Modulator» and «Analyzer» angles change at the same time by the same angle increment.<br>The user can set the Start, End and Step values. If the Phi (Analyzer) adjustment box is checked, the Analyzer head will automatically move up and down to find the maximum strength of the reflected beam. This optimization process is performed for each new angle. |

| | |
|---|---|
| **Spectral Range** | **Start/End:** enter the working wavelength range.<br>It is important to know that limits are defined according to your System configuration.<br>It is recommended to really enter your minimal working coverage to increase scanning speed.<br>**Use only one modulation:** checking this option will use a fixed modulation amplitude. In this case, acquisition time is faster but the measurements precision is slightly reduced in the IR range.<br>**Number of wavelength:** Depending on your System, up to 5 wavelength points ranges can be selected.<br>**How is it determined ?**<br>The Multiwavelength Spectrograph includes 32 photomultiplier tubes. Each of these PMTs analyzes one wavelength on the Spectrograph exit. If you choose a limited working range, a limited amount of PMTs could perform measurements. These limited measurement points are the lowest value displayed in the ***Number of wavelength*** field.<br>Now, if your Spectrograph includes a motorized wavelength scanning device, a multiple (up to 5x) of the elementary amount of points could be chosen. For example, if you select the full range of your System, it should be able to measure up to 160 wavelengths points.<br>**Number of shifts:** this is only an information field which indicates the number of mirror shifts are required to cover the amount of wavelengths on the specified wavelength coverage.<br>**Default values:** click on this button to enter the default parameters. |

Click **RUN** to perform the acquisition. The results are saved in a file. This file is located in the **Acquisition Data** folder. Select this folder then double-click on the acquisition file. The results will be then displayed.

**NOTICE**: The Triggers feature will be available soon but is not yet active.

*05/20/2016*

### 4.4.3 Graph Setup

***Graph Setup*** tab allows the user to setup the acquisition graphics presentation.

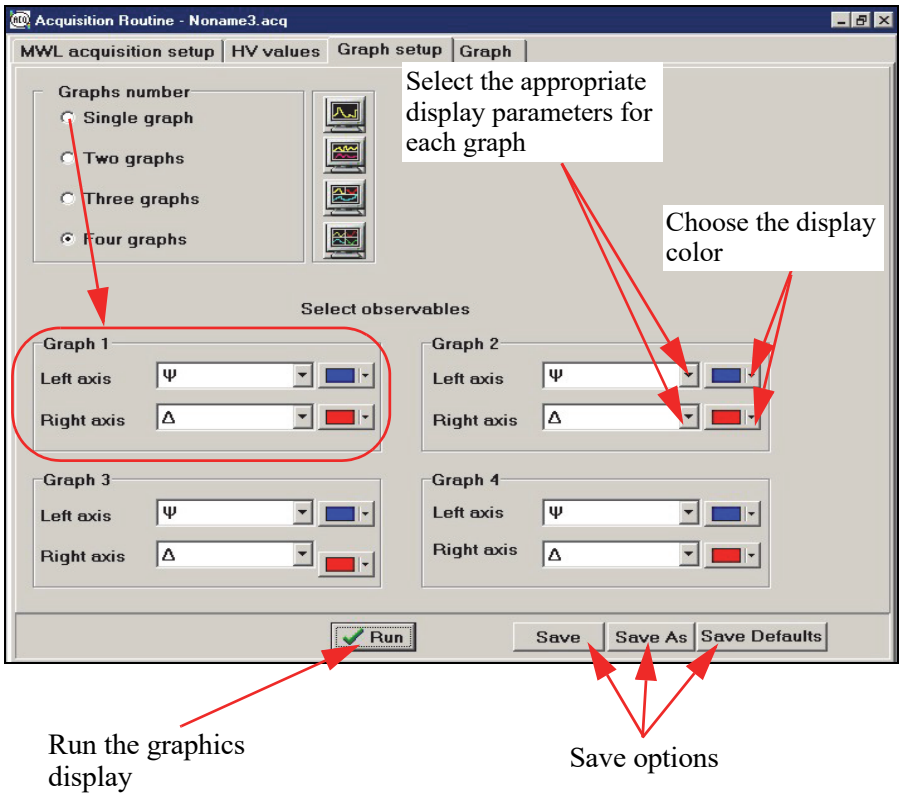



**Figure 4-8 Graph Setup**

### 4.4.4 Graph

If the **RUN** button is activated from the ***MWL acquisition setup*** or the ***Graph setup*** screen, an intermediate screen is then displayed which allows the user to fill in various information fields which generate the file name (see figure 4-9). the acquisition starts and the ***Graph*** tab is automatically switched to display acquisition results.

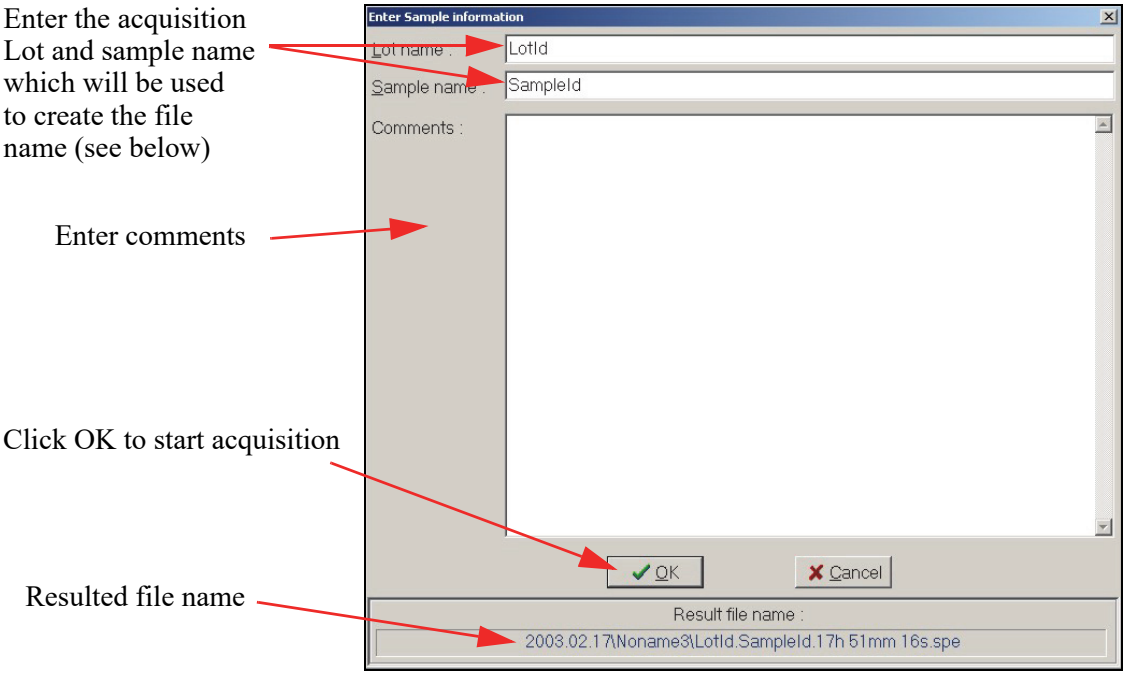



**Figure 4-9 Information screen**

Here below is an example of acquired spectra if four graphs are chosen:

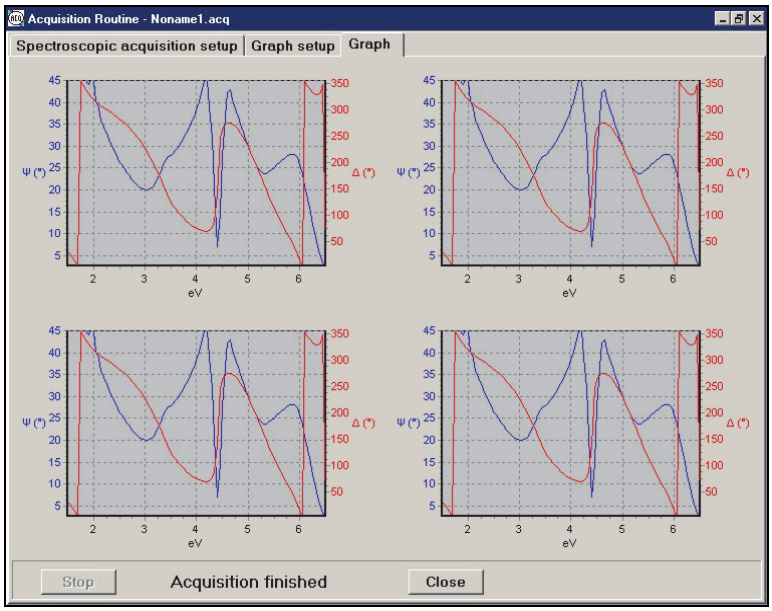


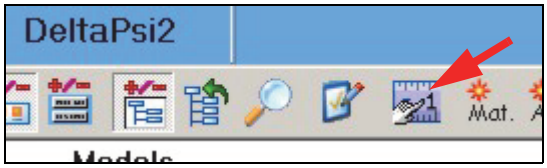

**Figure 4-10 Graph screen**

## 4.5 Views and Setup for UVISEL2 System

The UVISEL2 System is a complete automated System of integrated devices for acquisition and treatment of ellipsometric data.

### 4.5.1 Accessing

Double-click on the ***Manual Measurement***  icon from the ***Button Bar*** menu.




The following real time screen **Views** is displayed:

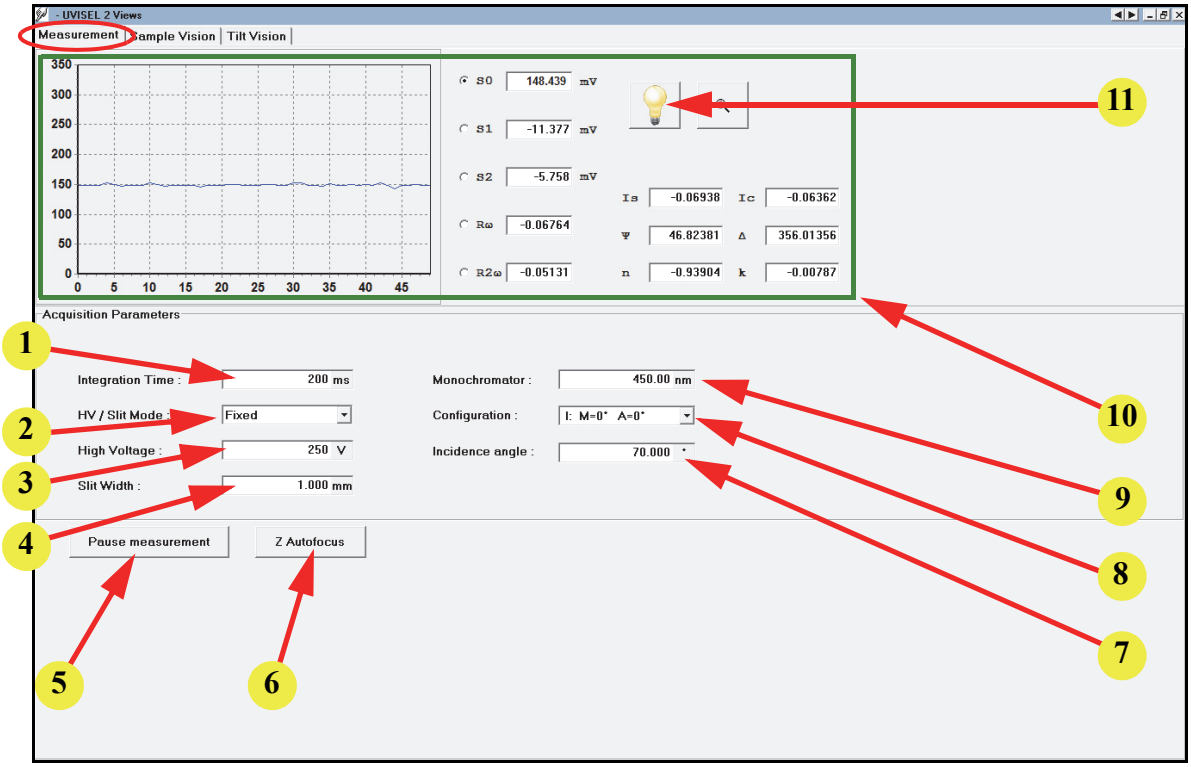



**Figure 4-11 UVISEL2 Views screen**

The Views screen allows the user to setup acquisition parameters and control the real time measured parameters.

The screen is organized with acquisition parameters described below:

**1** Integration time field

**7** Incidence angle field

2 High Voltage, slit mode: manual or automatic

3 High Voltage field

4 Slit width field

5 Pause the measurement

6 Run Autofocus

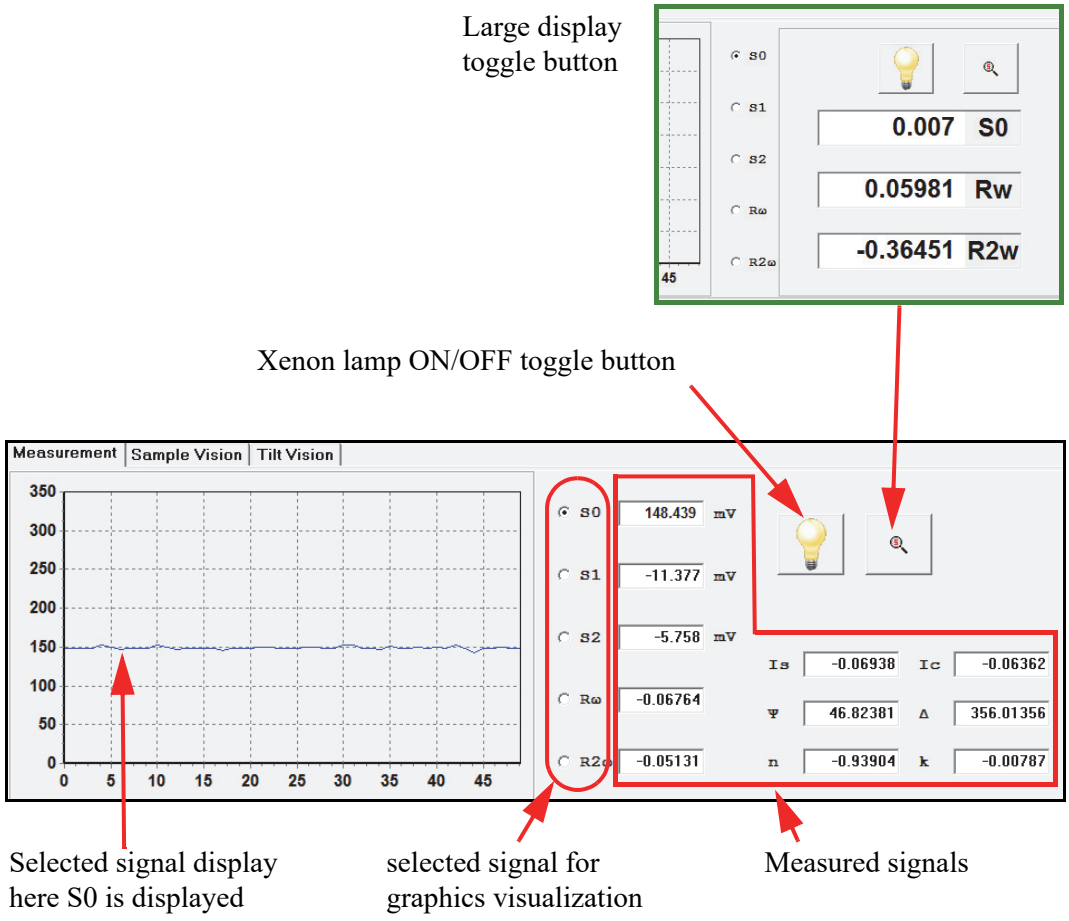
8 Ellipsometric mode selection

9 Monochromator Wavelength field

10 Real Time measurements panel

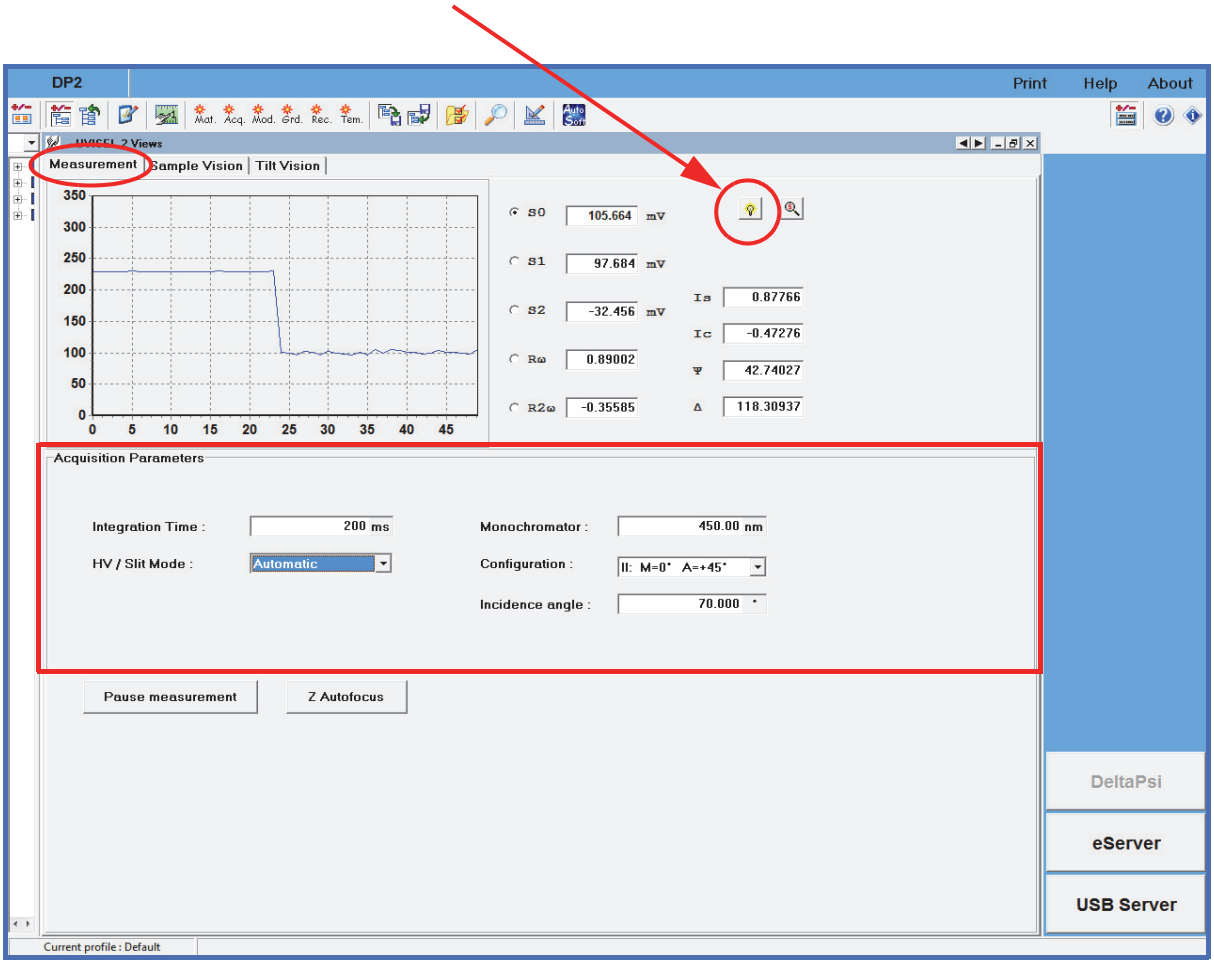
11 Xenon Lamp power ON/OFF

### 4.5.2 Real Time measurements panel



### 4.5.2.1 Measurement and Autofocus

**IMPORTANT: the icon**  **is a toggle button which turn the Xenon Light Source ON or OFF.**




The aim of this procedure is to perform the Autofocus.

The Autofocus is based on the level of the detected signal.

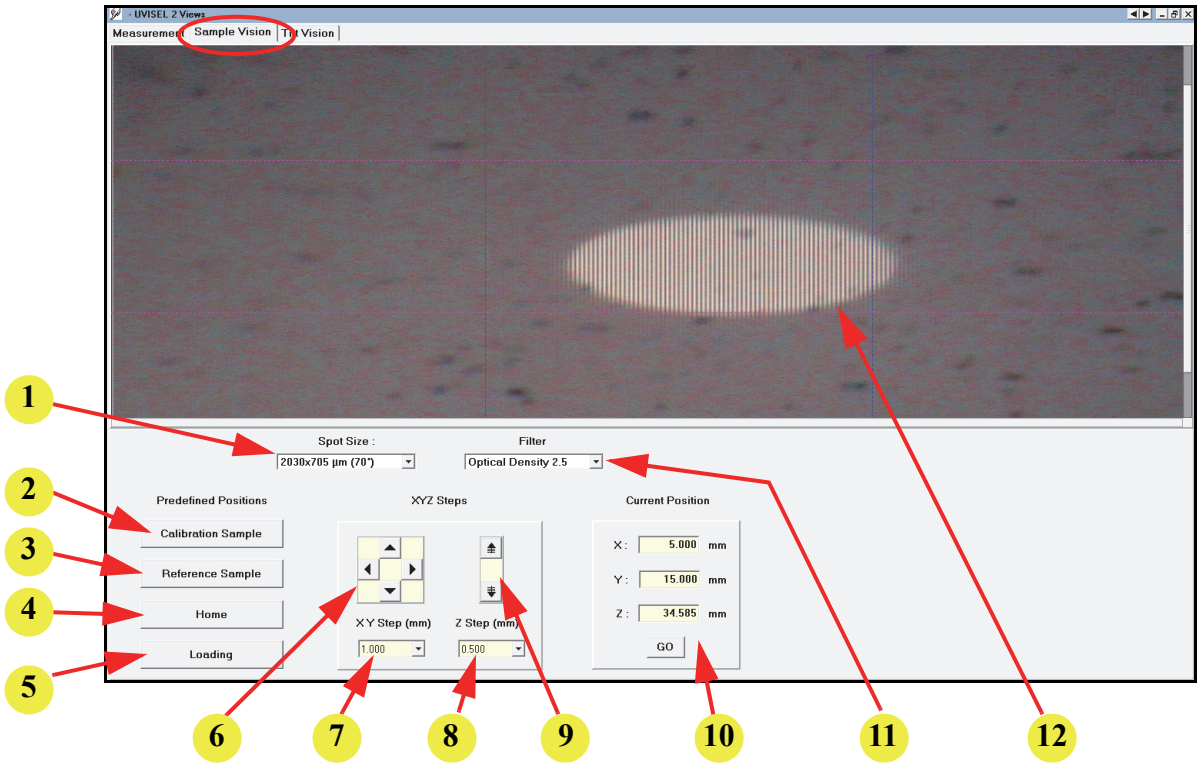
**1** Select the acquisition parameters (red frame) shown on the window above: the selected acquisition parameters are the «standard» which allows to control the overall measurements results.

**2** Place an Al sample on the Sample Stage.

Click on the Autofocus button  to perform the automatic Z Autofocus. Once done, your system is ready for the next step: Vision adjustment.

### 4.5.3 Sample Vision panel

The sample vision allows the user to adjust the sample area and the light source spot on this sample.




| | | | |
|---|---|---|---|
| 1 | Spot Size selection. Select the size of the spot on the sample. | 7 | Select or enter the XY step value |
| 2 | Move the Stage to the Aluminium sample (Calibration position) | 8 | Select or enter the Z step value |
| 3 | Move the Stage to the Reference $SiO_2$/Si sample. | 9 | Z stage joystick mode control |
| 4 | Move the Stage to the ***Home*** position (X,Y)=(0,0). A popup window asks for the sample thickness to position the Z axis. | 10 | Displays the XYZ current positions and editable fields to move manually the X, Y, Z positions. Click on the **Go** button to validate the entered values. |
| 5 | Move the Stage to the front to place or remove the sample. | 11 | ND (neutral density) filters selection |
| 6 | XY Stage joystick mode control | 12 | Visualization of the spot on the sample |

### 4.5.4 Tilt Vision

Third main step for setting the sample is the Tilt Vision control and adjustment. This powerful feature sets the horizontal tilt of the analyzed sample surface to reach the exact spacial reflection angles required for perfect analysis conditions.

**IMPORTANT: The Tilt adjustment must not be done on the reference sample (delivered and located near to the stage);** because this reference sample is not fixed on the mobile and adjustable sample plate.

**Procedure:**

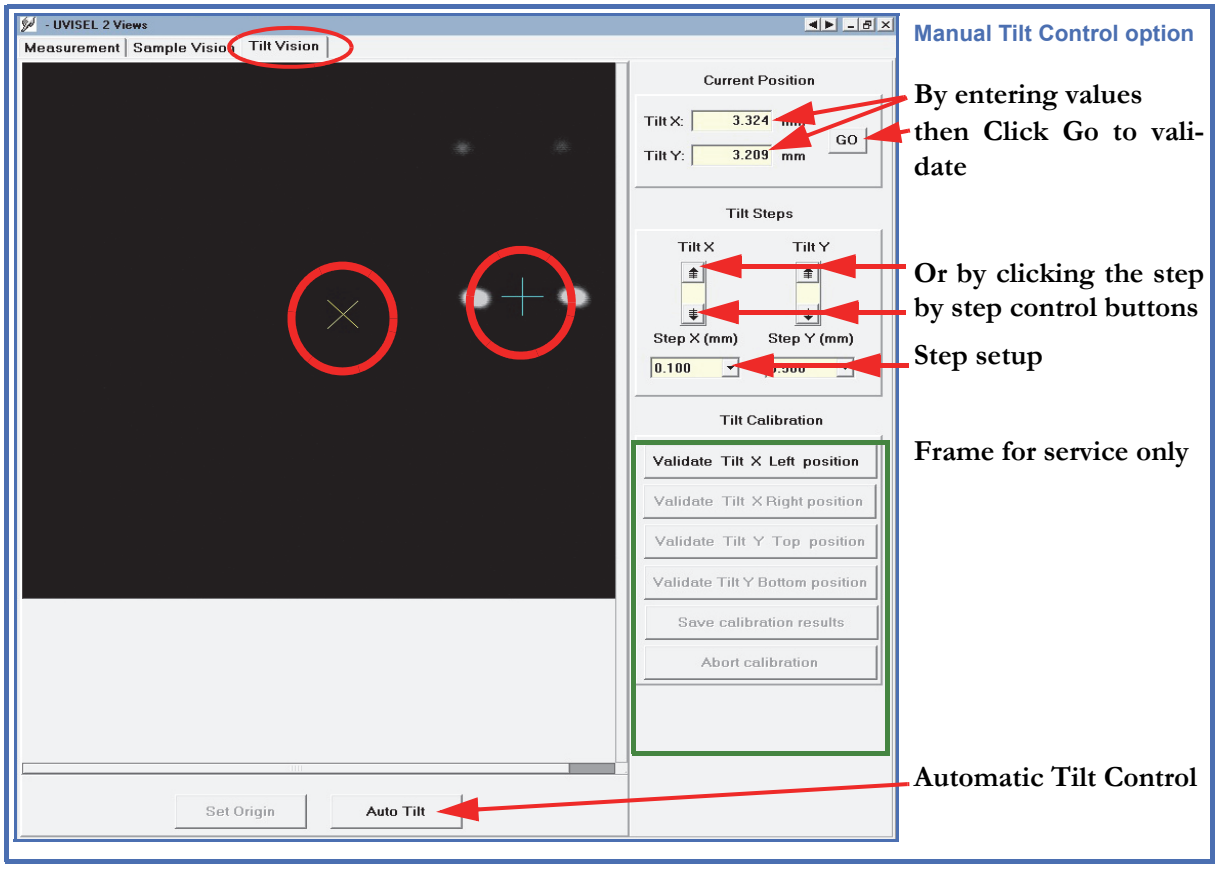
1 Verify that the laser (green) beam is oriented and reflected on the sample. If not, move the sample under the laser beam,

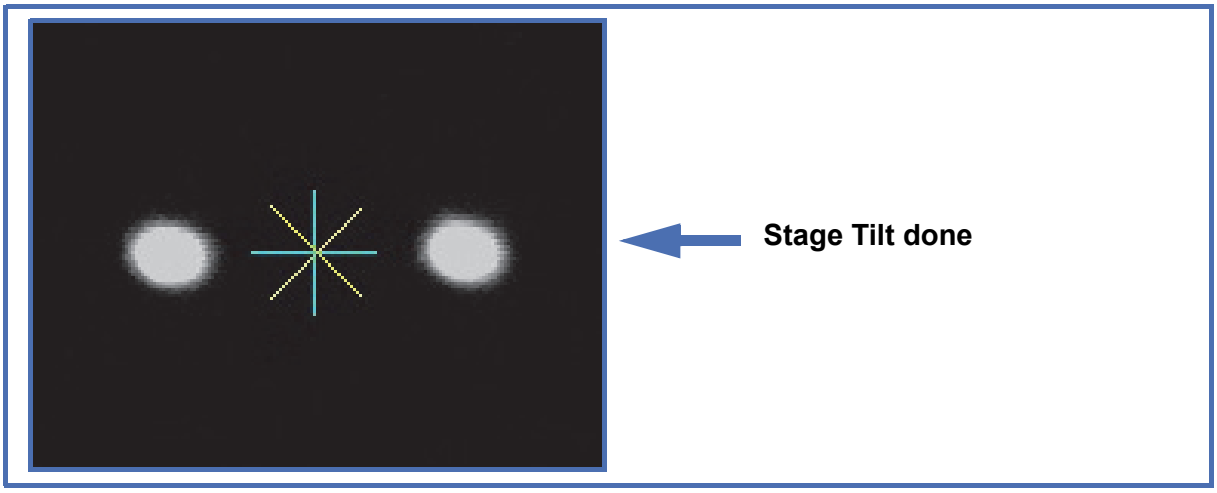
2 Using the Left/Right control buttons, move the beam to see it on the screen,

3 Click on the **Autotilt** button, twice or three times to see the blue and yellow crosses perfectly superimposed. The tilt is done.

4 **IMPORTANT**: Once **Autotilt** has been performed, it is important to launch the **Autofocus** again: if Autofocus changes the sample position, run again the Autotilt. This is due to the fact that the Autofocus and Autotilt settings are linked and interact with each other. See procedure in the chapter "Measurement and Autofocus" on page 67.

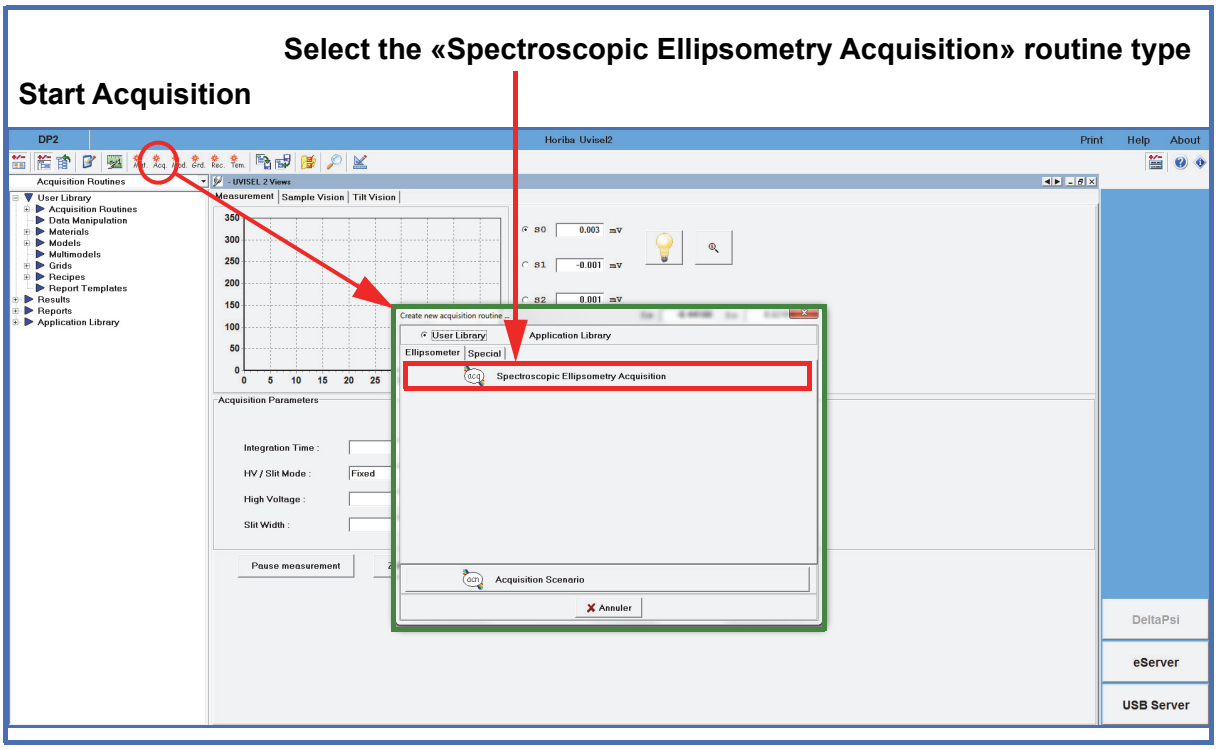
**NOTICE**: The tilt can be done manually using the control buttons located the right hand side of the screen (see figure below).





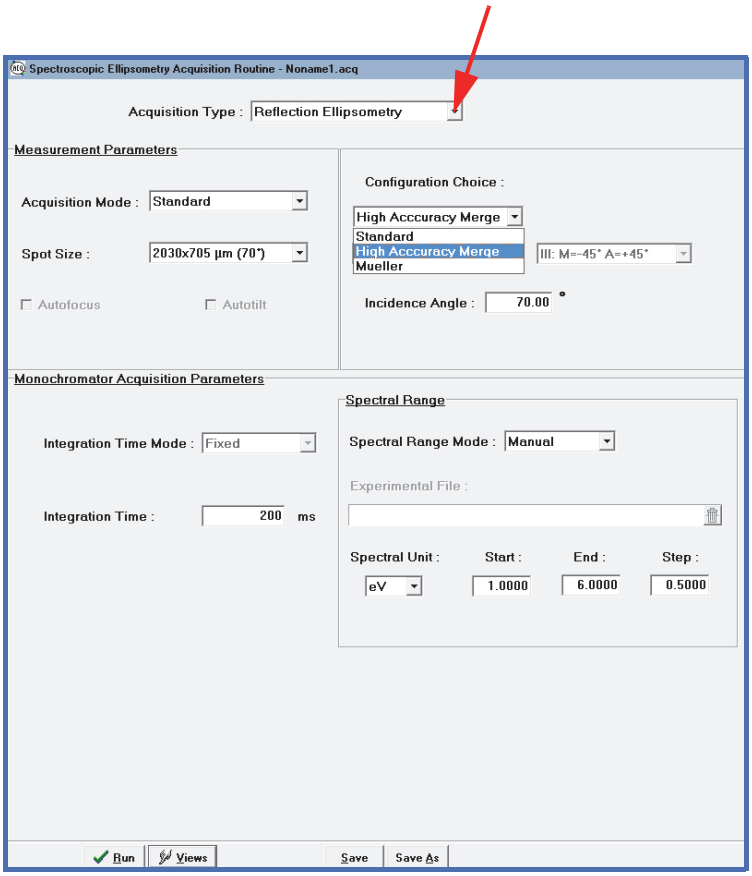

## 4.6 Acquisition with UVISEL2

Once the sample has been placed on the stage and once the complete setup has been made as explained in the chapter "Views and Setup for UVISEL2 System" on page 65, an acquisition can be performed. Follow the procedure explained below:




Once the acquisition routine type is selected, a window displays the available parameters need for acquisition.

**Select the appropriate acquisition type**




| | |
|---|---|
| **Routine Name** | By default, the «NonameX» is displayed. To rename the routine, click on the ***Save As*** button and enter the new name. |
| **Acquisition Mode** | **Standard:** The standard spectroscopic acquisition mode will be performed.<br>**Variable angle:** one spectroscopic acquisition will be performed for each PHI angle. Obviously, the «Modulator» and «Analyzer» angles change at the same time by the same angle increment. |
| **Spot size** | Programmed beam size can be set. The choices are:<br>2030x705 µm (70°)<br>585x525 µm (70°)<br>250x220 µm (70°)<br>182x120 µm (70°)<br>140x70 µm (70°)<br>85x35 µm (70°)<br>765x705 µm (70°) |

*05/20/2016*

| | |
|---|---|
| **Configuration choice** | Choosing a Merge method is useful to increase the precision on the Psi and Delta measurements according to the sample characteristics.<br>Additional choices are added to the chosen method.<br>**Standard merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second is done with Configuration III. The final spectrum is obtained in the following manner: Delta curve is taken from the Configuration II scan. Psi curve is taken from the Configuration III.<br>**High Accuracy merge:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained in the following manner: «Delta» curve is taken from the Configuration II scan. «Psi» curve is calculated by taking the Is and Ic from configuration III and Delta from configuration II with the following formula: Tg(2 * Psi) = IsII / [IcIII * Sin(DeltaII)],<br>**Mueller:** An acquisition method which includes 2 acquisition scans. The first scan is done with Configuration II. The second scan is done with Configuration III. The final spectrum is obtained like following: «Delta» curve is taken from the Configuration II scan. «Psi» curve is composed from several segments. The limits of segments are determined by some specified «Psi» threshold value crossed by the «Psi» curve (for example 40°). The segments with «Psi» less than this threshold value are taken from the Configuration II, if the values are over, they are taken from the Configuration III. |
| **Incidence angle** | Enter the incidence angle: angle determined between the Modulator, the sample and the Analyzer.<br>For information:<br>- Measurements for most semiconductors are usually made around 70°-75°,<br>- Measurements on transparent materials are made around 60°. |
| **Integration Time Mode** | **Fixed:** Enter the fixed integration time value in the related editor-field. |
| **Integration Time** | Enter the Integration Time value. The integration time is the amount of time allowed for the signal to be acquired for each measurement point. In general, select the shortest integration time that will give sufficient S/N for determining essential spectral features. |
| **Spectral Range Mode** | **Manual:** enter the Start/End/Step value (see below)<br>**From File:** load an acquisition file to retrieve the spectral Range parameters. |

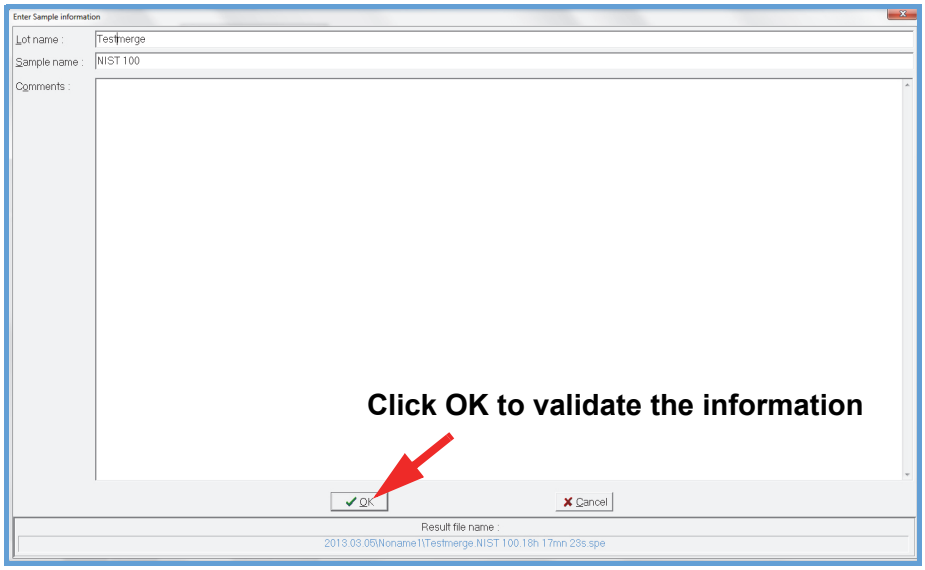
*05/20/2016*

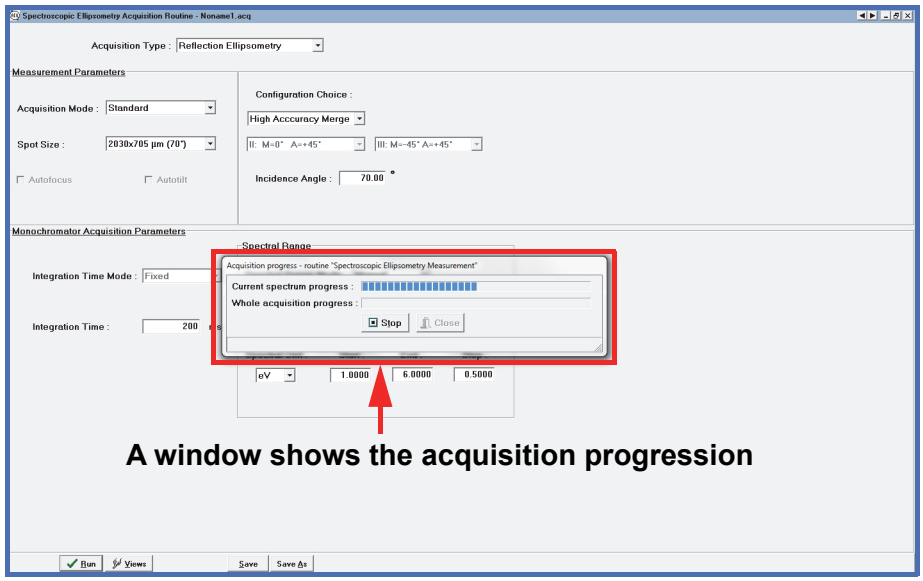
| | |
|---|---|
| **Spectral parameters** | **Spectral Unit:** select the working spectral unit, nm or eV.<br>**Start/End:** enter the working wavelength or wavenumber range.<br>It is important to know that limits are defined according to your System configuration.<br>It is recommended to really enter your minimal working coverage to increase scanning speed.<br>**Step:** enter the step on the spectral coverage. |

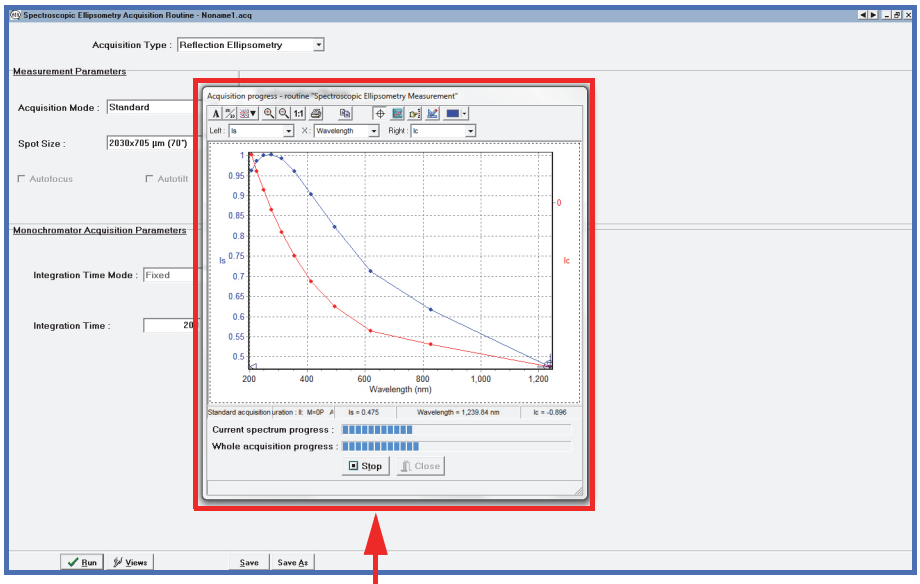
### 4.6.1 Launching acquisition

Once the acquisition parameters have been setup, click on the ***Run*** button to launch the acquisition.

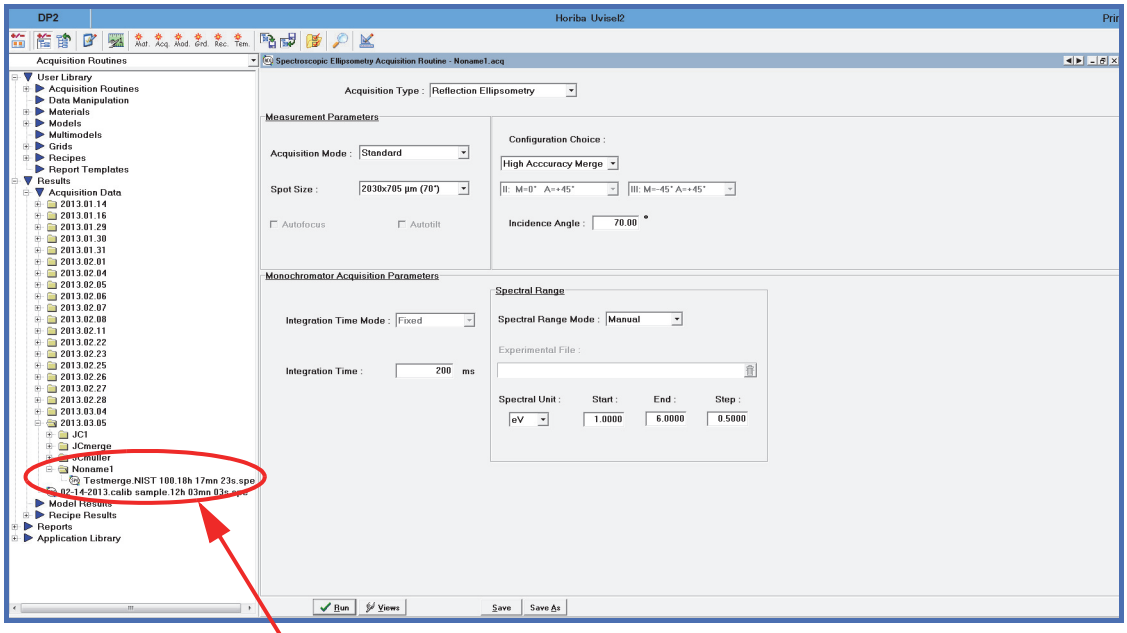
The following window will be then displayed: enter the ***Lot name***, ***Sample nam***e and the ***comments*** about the acquisition, then click ***OK*** to validate.






**Once the acquisition is done, the results are displayed**




**Default location of the acquisition data files, but can be changed according to the «Software Options» setup: see "Software Options" on page 14.**

05/20/2016

## 4.7 Views for MM-16 Ellipsometer

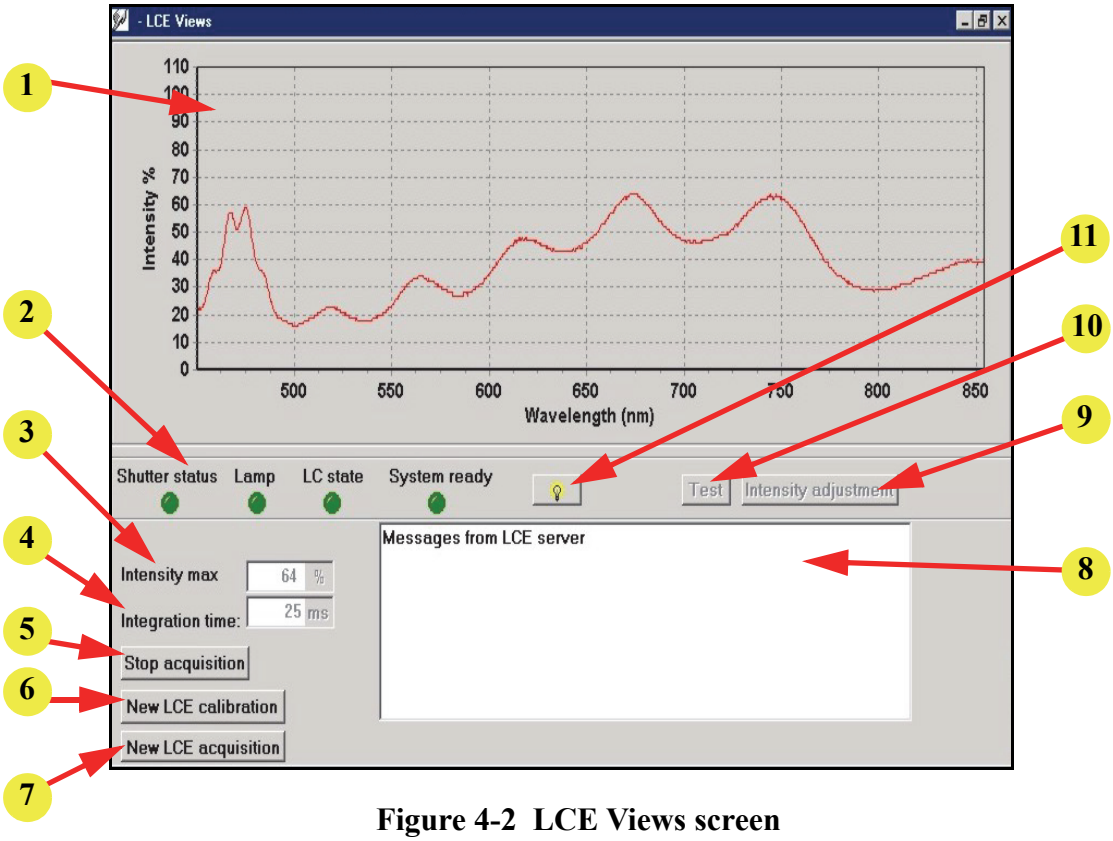



**Figure 4-2 LCE Views screen**

The ***LCE Views*** screen allows the user to setup acquisition parameters and control the real time measured parameters.

The screen is organized by operating sections described below:

**1** Graphics display area,

**2** Status indicator lamps,

**3** Signal Intensity max field,

**4** Integration time field,

**5** Start/Stop acquisition button,

**6** activates the automated calibration,

**7** Displays the acquisition routine screen,

**8** area for the messages delivered by the LCE server,

**9** Intensity Adjustment test,

**10** System Test,

**11** Light Source ON/OFF (shutter activation),

*05/20/2016*

### 4.7.1 Acquisition Parameters Test

## 1 Preliminary

Before any acquisition setup and measurements, it is important to test the MM-16 System integrity. Follow the procedure described in this chapter.

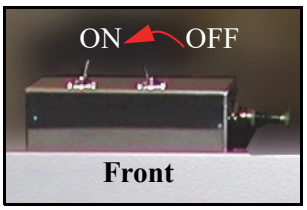
Verify that this  button (light ON) is displayed: that means the Light Source beam is delivered to the Input Head (default is activated),

## 2 Preparing the sample

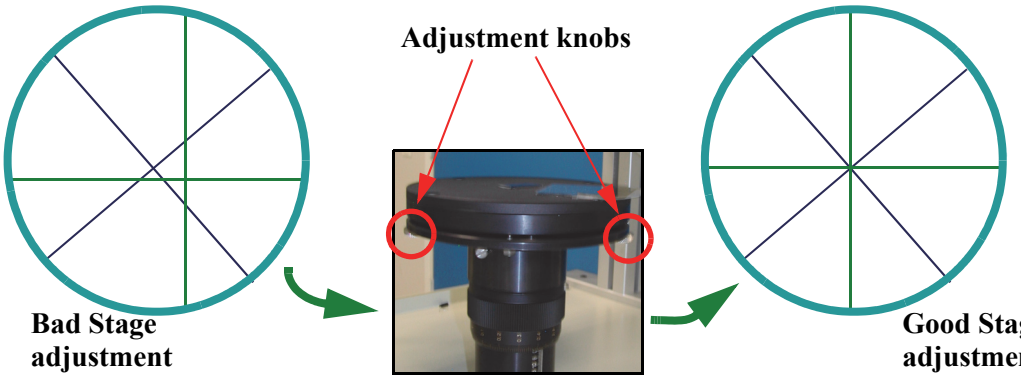
To perform a calibration and a test, place an Aluminium sample (delivered with the System) on the middle of the stage. This sample must be now adjusted to reflect the exit beam on the Exit Head.

The Aluminium sample is suitable for calibration because its reflection covers a large wavelengths range.

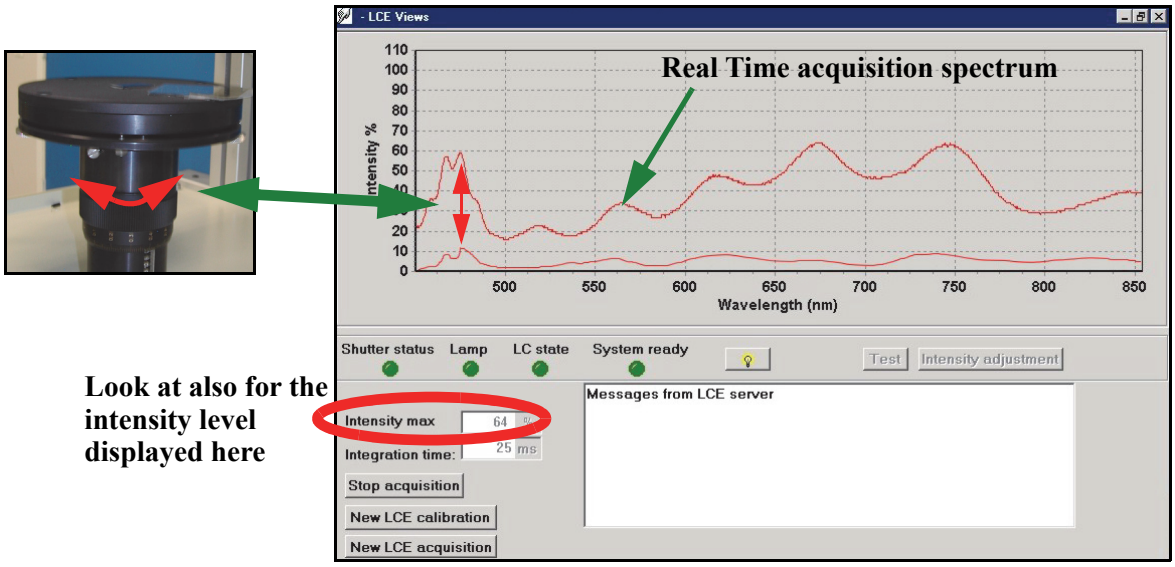
- **Power on the Autocollimator; the vacuum activation is not required,**



- **Look at inside and turn the two knurled knobs located below the stage to meet the middle of the crosses.**




- **Turn the Z stage adjustment to increase and find the maximum signal level of the spectrum displayed on the Views screen (Real Time display).**




- **press the** Test **button to test the overall MM-16 System: the «comments area»** 8 **should answer «LC Test Ok» message. If an error occurs, a message will be delivered.**

- **Press the** Intensity adjustment **button to perform a signal intensity test: the results will be displayed in the «intensity max» and «Integration time» fields. This test can only be performed if a sample is placed on the stage.**

During all these tests, the status lamps should be stay in green color. If not, an error has been detected. In this case, an error message is displayed.

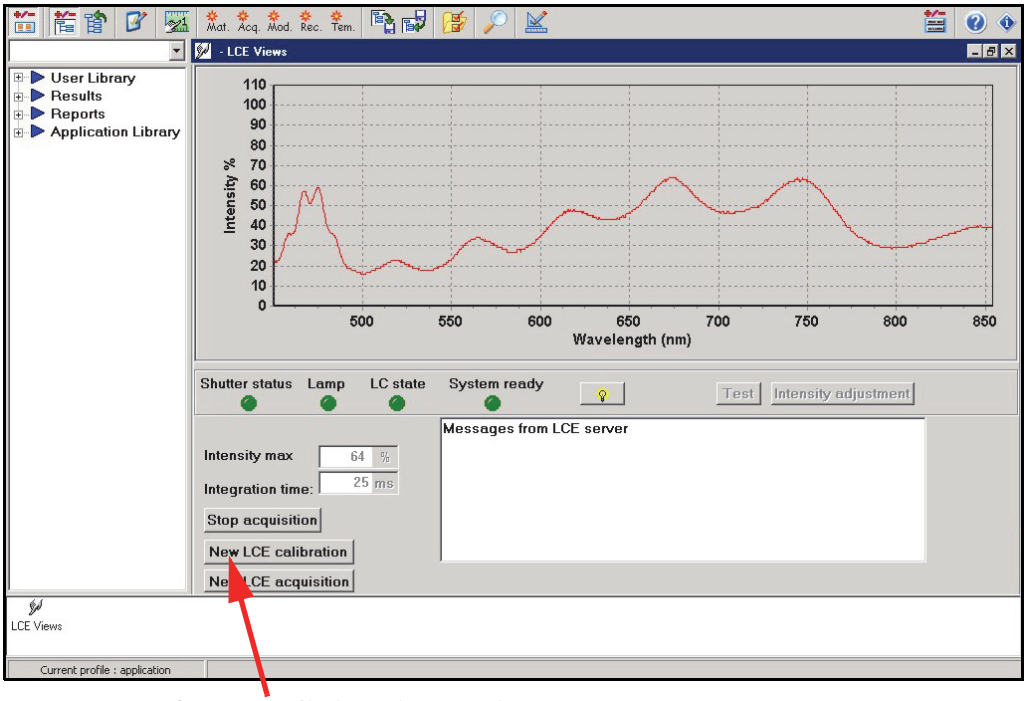
05/20/2016

During the tests, a sample **must** be placed on the stage.

### 4.7.2 System Calibration

Performs the steps described previously in the "Acquisition Parameters Test" on page 76. The System is now ready to be calibrated from the DeltaPsi2 Software.

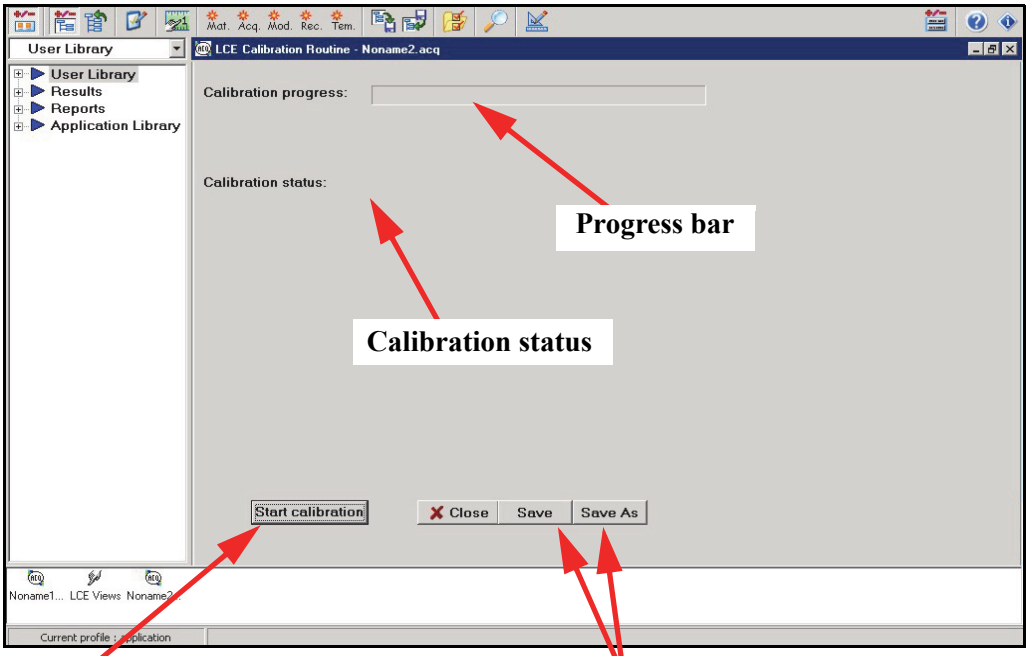
- Click on the **New LCE calibration** button to open the calibration screen.




**Open the Calibration routine screen**

## 3 Launching the Calibration

Click on the ***Start calibration*** button to launch the automated calibration routine. If a pop-up screen is displayed (see figure 4-3, page 79 below), click on the ***OK*** button. Wait for about 10 minutes and the message «successfully ended» will be displayed. The System calibration is now performed and the parameters automatically saved.




If the software displays the following error message «**Calibration failed: Integration time evaluation error, underflow. Calibration step stopped. No error description**», that means the aluminium sample has not been adjusted on the sample stage. See Z stage adjustment on **"Turn the Z stage adjustment to increase and find the maximum signal level of the spectrum displayed on the Views screen (Real Time display)." on page 77.**

The ***Save*** and ***Save As*** buttons will NOT save the calibration; they save the current Acquisition Routine for a further use.

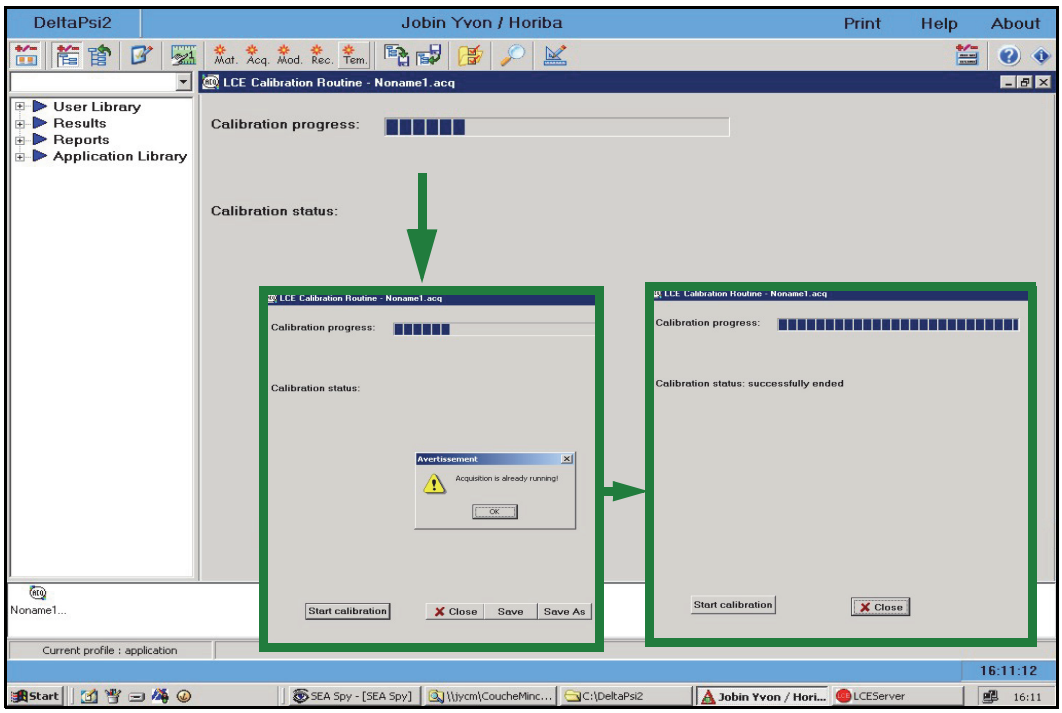
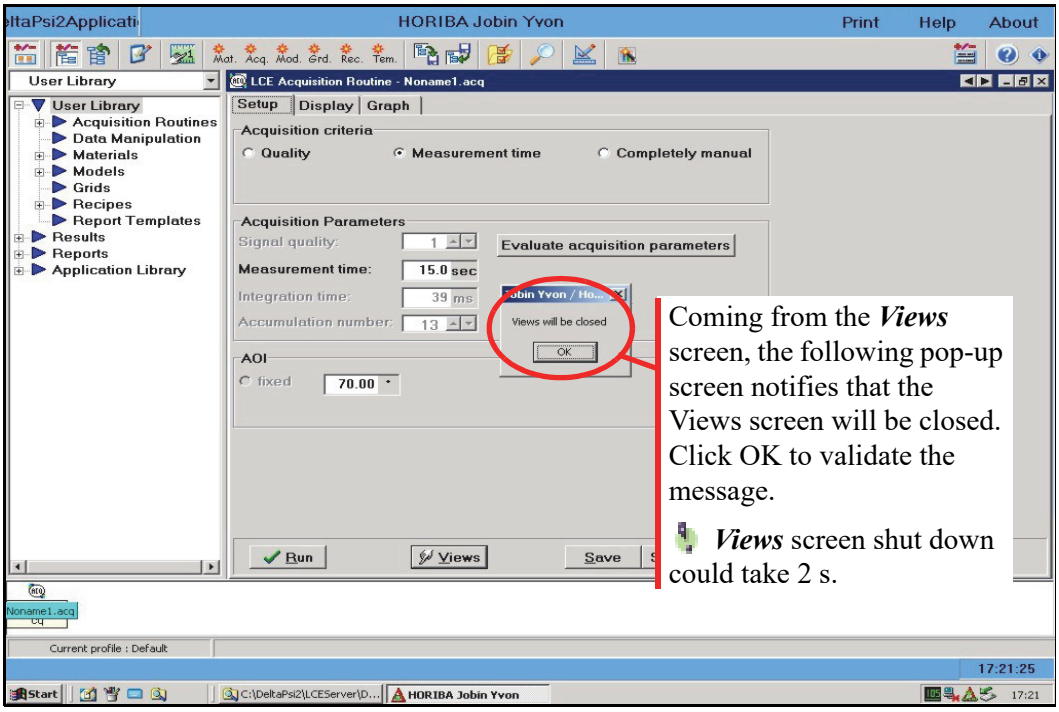



**Figure 4-3 Automated calibration steps**

### 4.7.3 Acquisition Setup routines

Do not touch the sample and the stage adjustment previously set for the Calibration.

From the ***Views*** screen, click on the New LCE acquisition button or from the ***TreeView*** right-click on the ***Acquisition Routine*** and select ***New LCE Acquisition*** choice. The following screen will be then displayed:




# 4 Acquisition parameters setup

Before starting an acquisition, the parameters must be set according to the sample. For our test with the Aluminium sample, the following procedure will be performed. With another sample, the same procedure should be performed as a start action; usually this automated procedure will be enough to reach good measurement conditions.

If the software displays the following error message «**LCE server: Spectroscopic spectrum error: Acquisition failed: Underflow**». Three reasons could generate this message:

1 The sample has not been adjusted on the sample stage. See Z stage adjustment on "Turn the Z stage adjustment to increase and find the maximum signal level of the spectrum displayed on the Views screen (Real Time display)." on page 77.

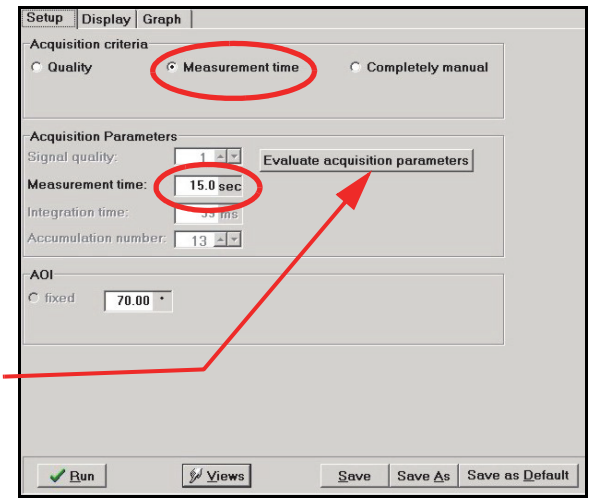
2 The sample is not enough reflective.

3 If the Evaluate acquisition parameters has not been activated. See below the description of this feature.

- **The procedure described below will use a standard safely setup to reach good results.**
- **The screen includes three tabs: *Setup* which concerns the hardware, *Display* and *Graph* which concern the displayed format.**

**1 Check these boxes first**

**2 Then click on the evaluate acquisition parameters**



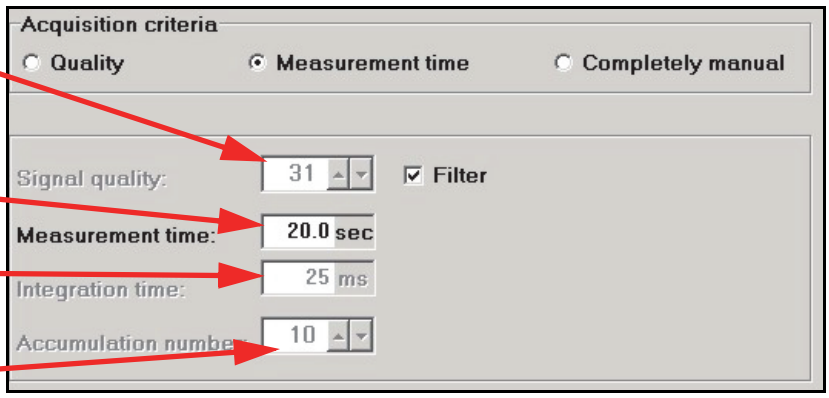

- The automated mode with «Measurement Time» priority set the acquisition parameters according to the entered total acquisition time. Filter activation is highly recommended. Clicking on the Evaluate acquisition parameters button will modify the acquisition parameters according to the time limit. Thus for a specific use, it becomes easy to choose the «Quality» priority or the «Completely manual» mode to modify the parameters. **In our test, we will choose the «Measurement time» mode. Pressing the Evaluate acquisition parameters button will generate results shown on the figure below:**

**Higher is the level better is the signal quality**

**Manually fixed limit time (in this mode)**

**Better integration time found by the program**

**Better accumulation number found**




- **Click on the Save or Save As button to save the Acquisition routine.**

# 5 Specific Setup

The Step 1 shows a «safe» acquisition setup for a good general results. In practice, it could be useful to use the two other modes: «Signal Quality» and «Completely Manual».

In the acquisition setup screen, the ***Acquisition criteria*** area shows three different modes: ***Quality***, ***Measurement Time*** and ***Completely Manual***.

- **«Quality» mode uses the quality as a prior user parameters. The user can enter a value from 0 to 100. 100 offers the best quality. In practice this mode is useful if two or several samples are measured. If the quality is fixed, then the results could be really compared between the samples acquisition results.**
- **«Measurement Time» mode is useful for two applications:**
  - As a start acquisition setup (see in the Step1)
  - To limit the acquisition time: if several thousand measurements must be performed on a wafer, it could be interesting to find the minimum acquisition time for an acceptable result.
- **«Completely Manual» is a mode for advanced users and specific conditions.**

***Quality***: select a number between 1 and 100. Higher is the number, better will be the quality.

***Measurement time***: time for an acquisition.

***Integration time*** (completely manual mode only): enter the desired integration time value. The integration time is the amount of time allowed for the signal to be acquired for each spectrum. In general, select the shortest integration time that will give sufficient S/N for determining essential spectral features. This entered value is the CCD Integration Time and not the measurement time of the Ellipsometer.

***Accumulation Number*** (completely manual mode only): enter the number of appropriate accumulations, taken into account that the signal/noise ratio may be improved by performing more accumulations per spectrum, where each accumulation is an individual integration time. Again, for n number of accumulations, the signal/noise ratio will improve by the square root of n. The trade-off is that the experiment takes n times longer.

# 6 Advanced Acquisition (Option)

The Advanced Acquisition setup has been developed to create multiple recipes with very different parameters setup. This feature is an option delivered with a licence key. Two different steps have been created to reach the desired flexibility:

- **Initial Acquisition routine**: this acquisition screen can be easily recognized by its green color header. From this screen two main choices are available: ***Quality*** or ***Measurement time*** priority. By default, the ***Evaluated Values*** of the background acquisition is set automatically, but the user can set it manually (***User Target*** choice). The Angle Of Incidence (AOI), can only be set (available only from motorized goniometer). Once the Run button is activated, a complete parameters optimization is performed. They can be saved and used in the second step. The figure 4-4 shows the Initial Acquisition screen.

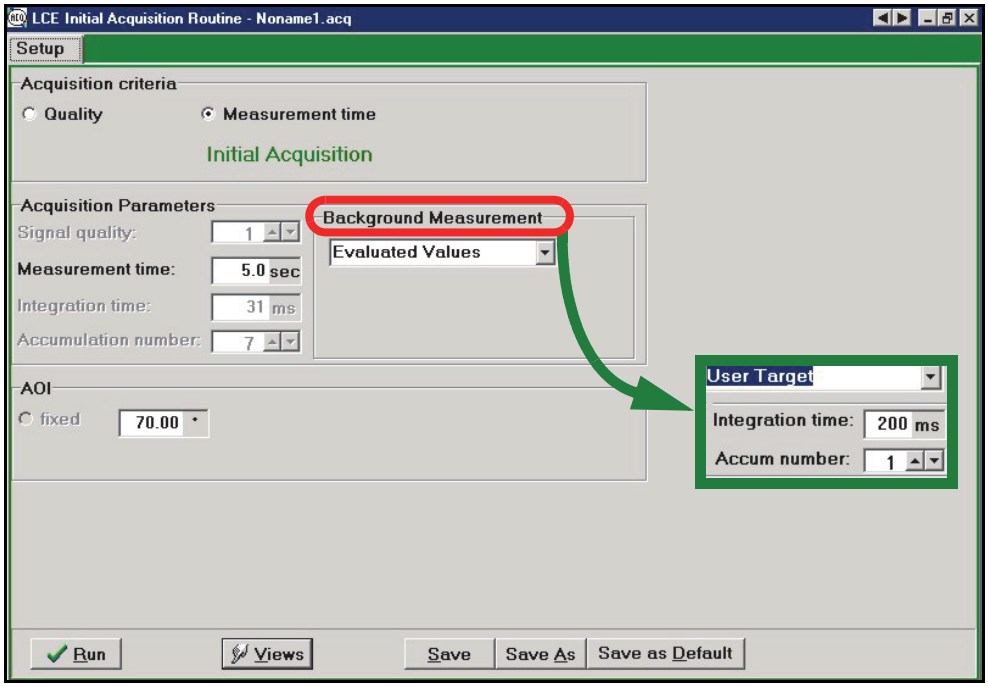



**Figure 4-4 Initial Acquisition routine**

- **Advanced Acquisition routine**: This routine can be easily recognized by its brown-green color header. Logically, this routine performs the second step. It retrieves a ***previous run Initial Acquisition*** routine, calculates then displays the ellipsometric measurements. Using the Recipe feature, it becomes possible to add several ***Initial Acquisition*** routines followed by an ***Advanced Acquisition*** routine. An Advanced routine will always retrieved the results parameters of the previous Initial routine. The following figure shows the setup for this typical use:

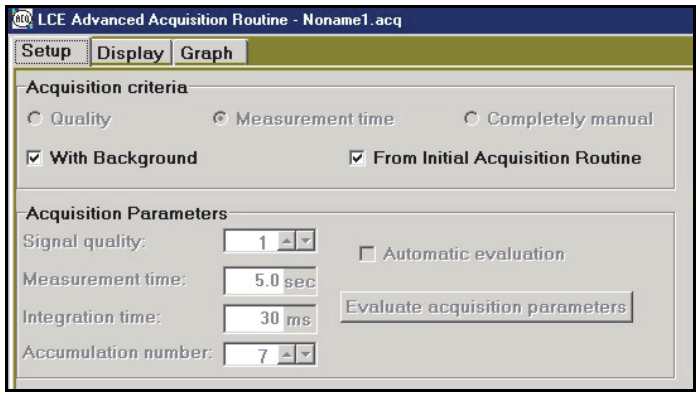



**Figure 4-5 Advanced acquisition routine**

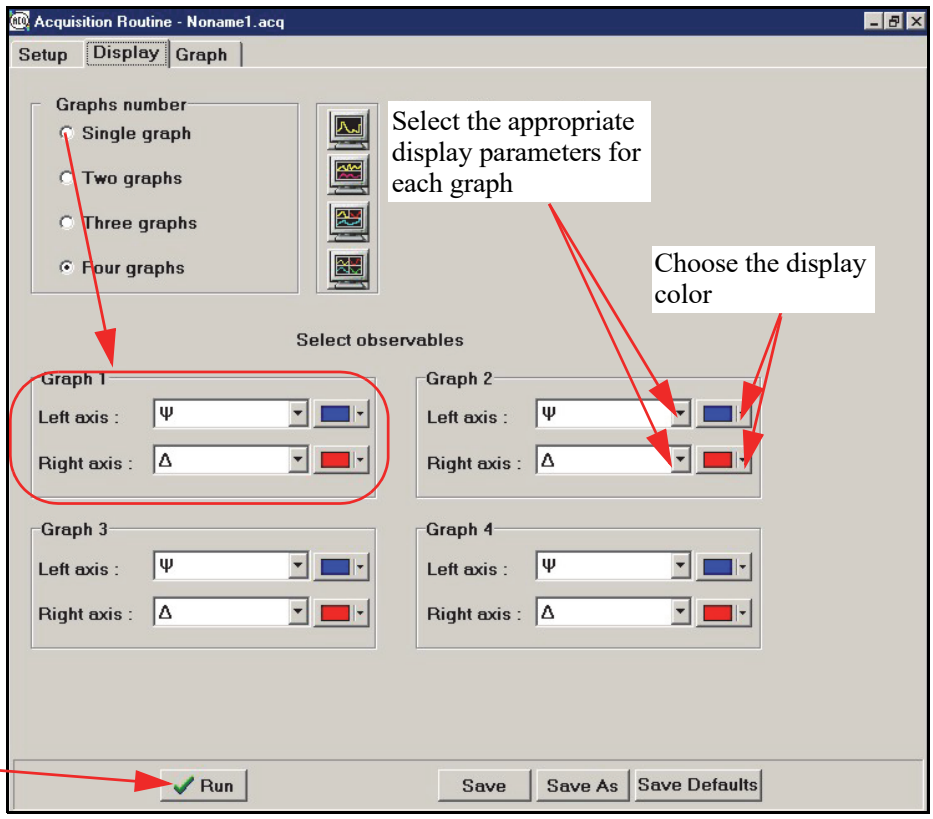
Therefore, the Advanced Acquisition routine allows the user to perform a stand alone measurement. All of the Standard Acquisition routine choices are available and some others have been added:

***With background***: if checked, the background will be automatically measured then subtracted from the acquisition.

***Automatic evaluation***: if checked, each time the ***Run*** button is activated, the routine will evaluate the parameters then performs the acquisition. One click for results!

# 7 Display Setup

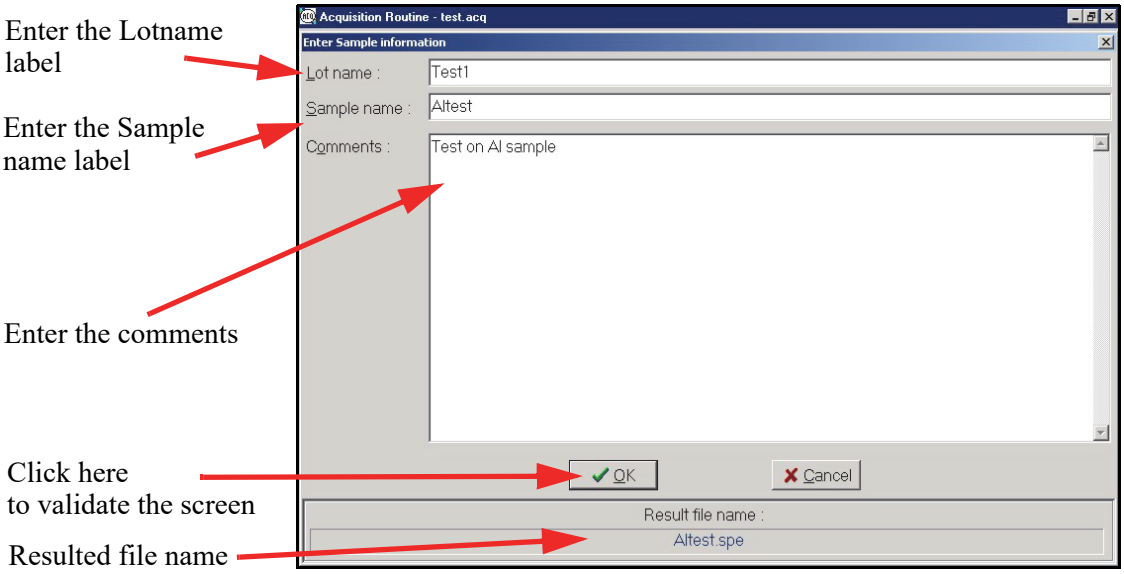
***Display Setup*** tab allows the user to setup the acquisition graphics presentation.




# 8 Acquisition

Click on the Run button to launch the acquisition process. The next screen will be displayed:

05/20/2016




The ***Lotname*** and ***Sample name*** information are used to organize the files on the hard disk. Please refer to the chapter "File name formats" on page 15.

The Comments will be included inside the file and could be useful later as information.

Once the information screen is validated, the acquisition is launched and the acquisition graphs are displayed. Then the following final acquisition results screen is displayed:

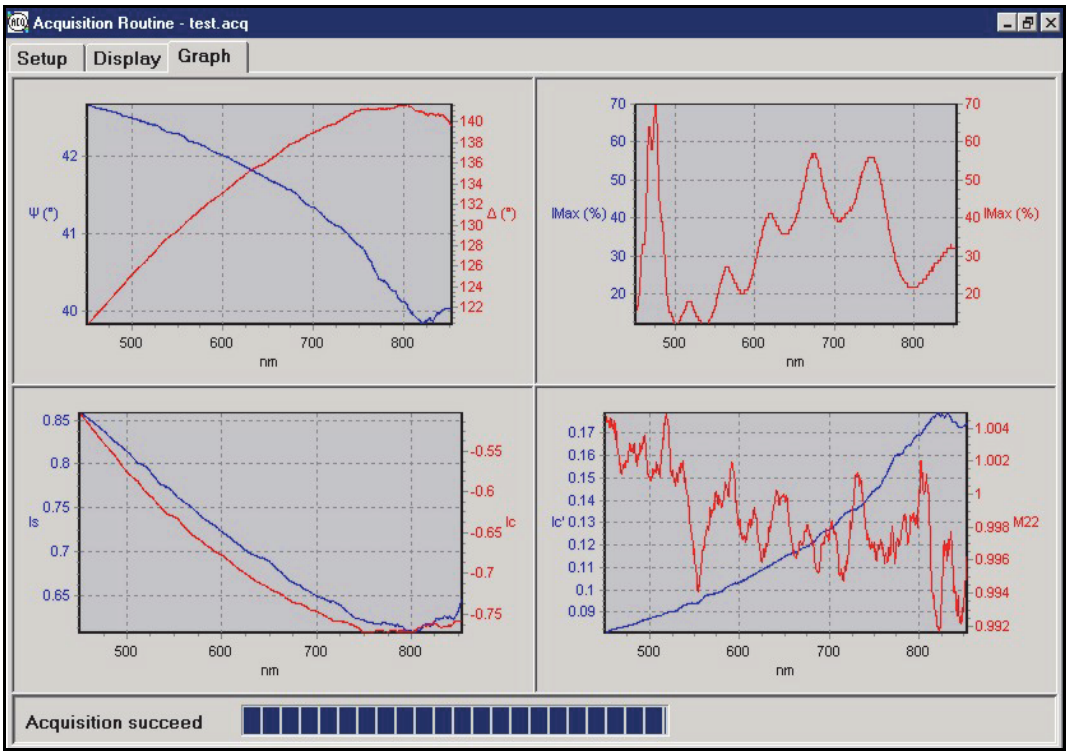



**Figure 4-6 Acquisition results**

## 4.8 Views for Reflectometer (UVISEL VIP)

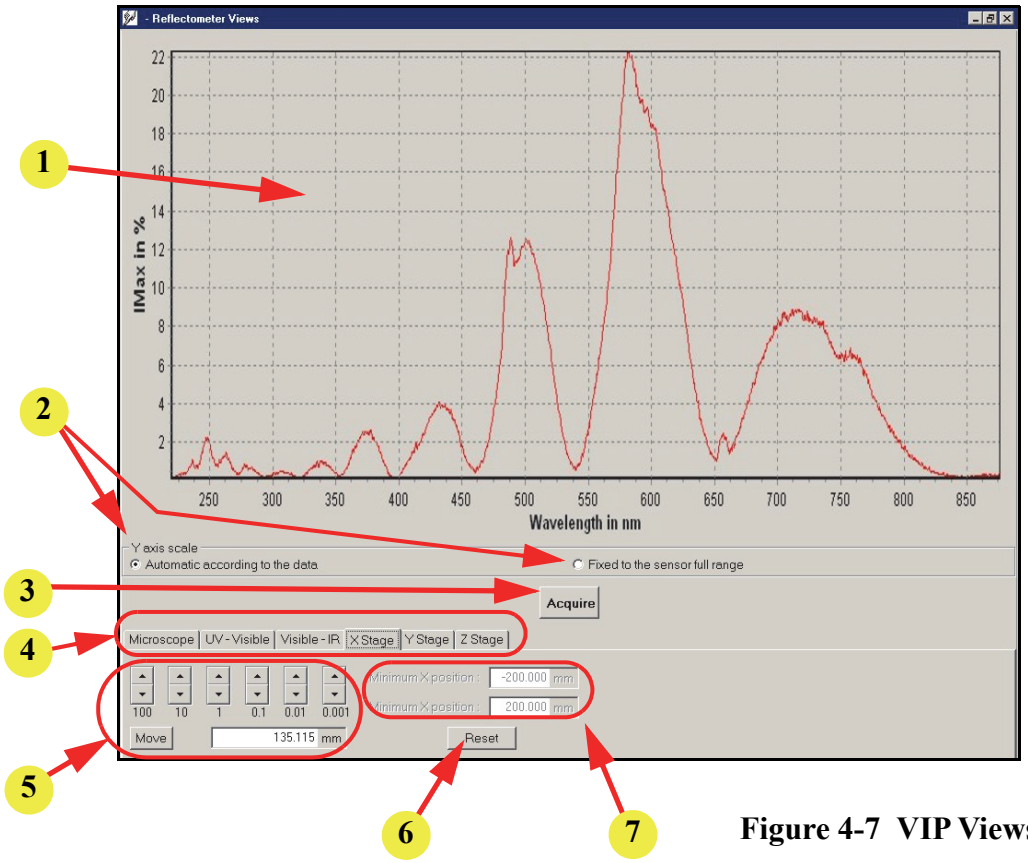



**Figure 4-7 VIP Views screen**

The ***VIP Views*** screen allows the user to setup acquisition parameters and control the real time measured parameters.

The screen is organized by operating sections described below:

**1** Graphics display area.

**2** Y axis scale:

**Automatic**: scale repositioning according to the measured data,

**Fixed**: sensor range is displayed as it.

**3** Toggle button: Start/Pause measurement.

**5** Motors setup panel: according to the selected motor device, adjust the value and click on the move button to activate the changes.

**6** Reset the changes.

**7** Information field: minimum/maximum values allowed for setup the selected device.

05/20/2016

4 Setup (Spectrometers, Stage...) switching tabs: select the device to setup.

## 4.9 Acquisition for Reflectometer (UVISEL VIP)

### 4.9.1 Accessing

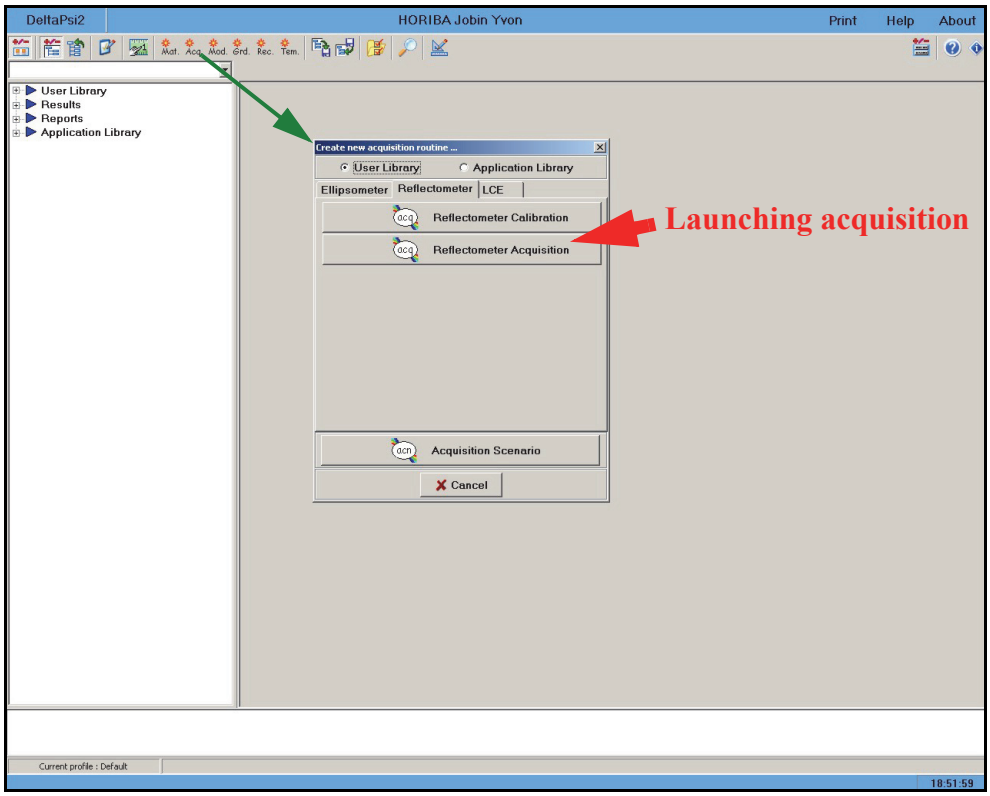



**Figure 4-8 Launching acquisition**

# 5 Reflectance/Transmittance Measurements on UVISEL

## 5.1 Definition

The purpose of this technique is to measure the intensity modification upon reflection or transmission on or through a sample.

$I_r$ = intensity of reflected beam

$I_t$ = intensity of transmitted beam

Typically given as relative quantities:

$$R = \frac{I_r}{I_0}$$

$$T = \frac{I_t}{I_0}$$

with $I_0$ = Intensity of incident beam


**Change of polarization upon reflection (or transmission)**

$$\rho = \frac{R_p}{R_s} = \tan\Psi \cos i\Delta$$

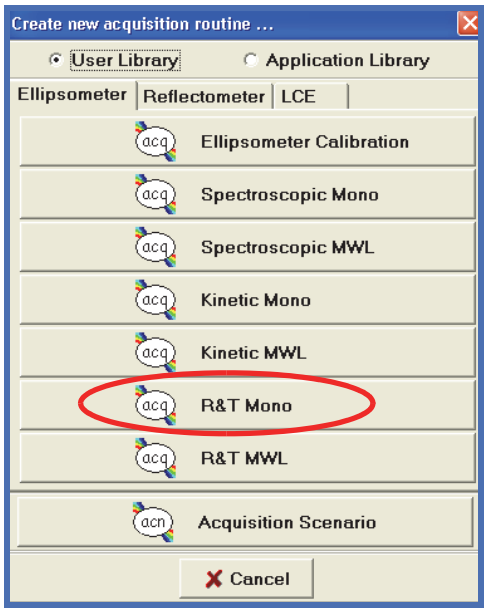
p = parallel to plane of incidence

s = perpendicular to plane of incidence

## 5.2 Measuring the reflectance of your sample

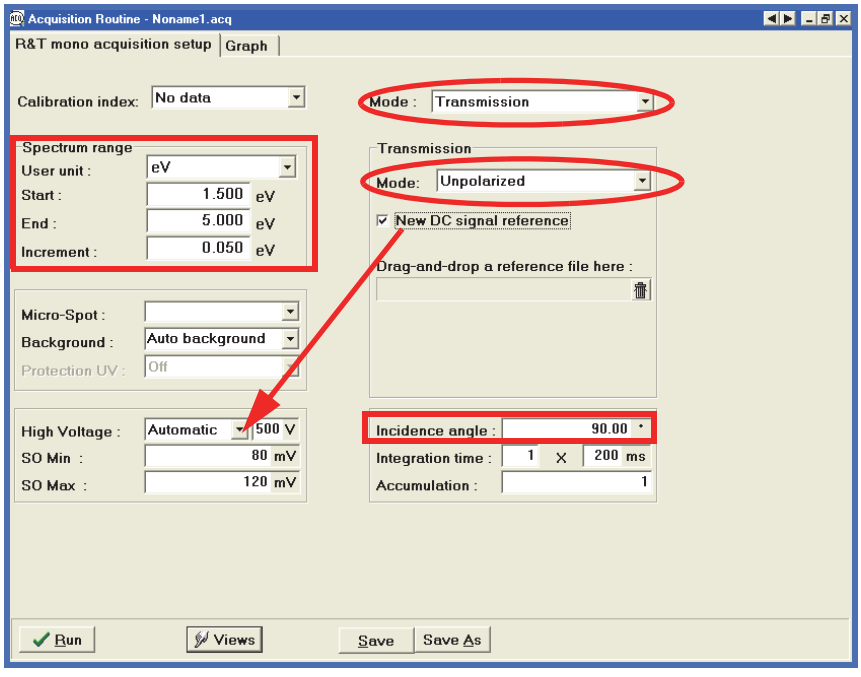
- Click on the acquisition tab 

(the seventh button from the left on the top tool bar)
- Select the “R&T mono” Acquisition mode in the menu




- Select the Reflection mode (the 1st box in the right column) and then the polarized or unpolarized mode. You will measure respectively either Rp and Rs (polarized R mode) or R (unpolarized R mode) using these 2 modes.

- Select your spectral range and the incidence angle wished – the other parameters are set by default, you can keep them.




The Reflectance is calculated from the following equation:

$$Reflectance = \left( \frac{S_{0\,experimental}}{S_{0reference}} \right) \times Reflectance_{reference}$$

The reflection routine allows to measure these both parameters in the following three steps:

05/20/2016

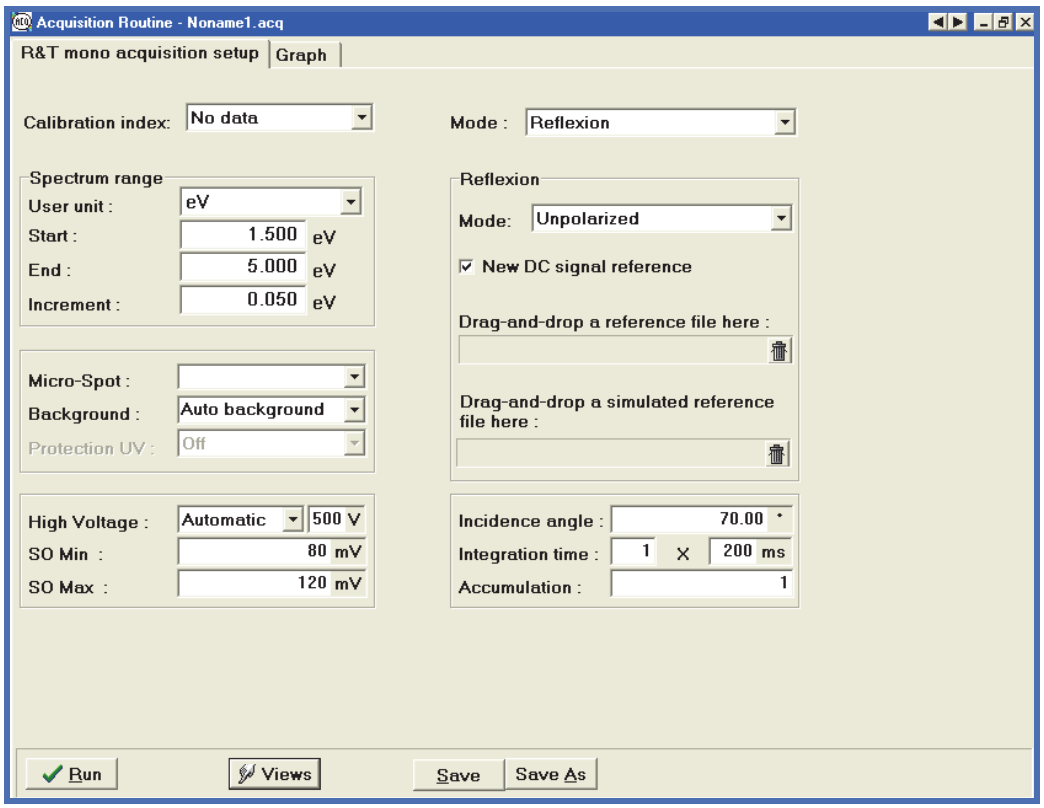
**1** The 1st step is the measurement of the So reference: Aluminium mirror or native oxide sample Si substrate.

➔It consists in the measurement of the So and Hv values of sample reference on the defined spectral range.

**2** The 2nd step is the modeling of this sample reference in order to calculate its reflectance R, Rp & Rs

**3** The 3rd step is the measurement of the So experimental of your sample.

➔It consists in the measurement of the So of your sample in the defined spectral range by keeping the values of the High Voltage measured for the sample reference in the 1st step.




# 1 Acquisition of $S_0$ reference – the reference sample is an aluminium mirror/native oxide sample (Si substrate).

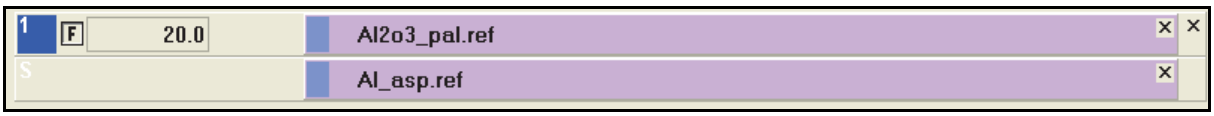
- Place the reference sample on the sample stage
- Check the box New DC signal reference
- When clicking on this box, the automatic mode of the high voltage appears in black. This is set by default and you have to keep it.
- If you want to save this routine, click on save as and enter a name.
- Click on Run. The So and Hv values of the sample reference will be measured on the spectral range defined.

- When the acquisition is finished, the file is located in the left hand side menu in the branch Results / acquisition data.

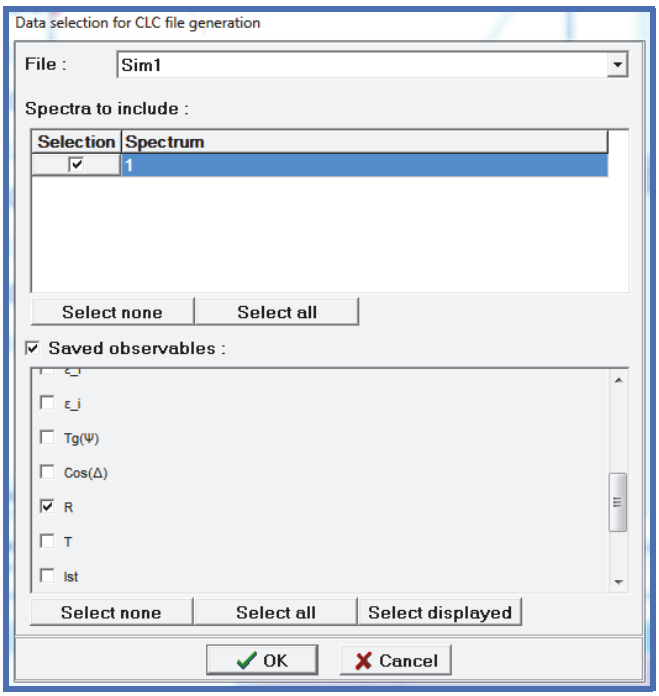
# 2 Creation of the model of the reference sample to calculate the Reflectance of it

- Click on the mod tab 

(the eighth button from the left on the top tool bar),
- Select the ***Spectroscopic model*** in the menu,
- Build a model composed of a 40-50 Å aluminium oxide on Al / 18Å SiO2/Si by using the reference files for both materials located in the application library in the branch materials.




- Run a (R, Rp,Rs) simulation of this model,
- Click on this tab  to save the reflectance results,
- Select the file called Sim and check the box saved observables and select photon energy, wavelength, R , Rp and Rs,
- Click on ***OK*** and save this file. The file will be located in the User Library in the left hand side of the menu, in the folder Results / Acquisition data




05/20/2016

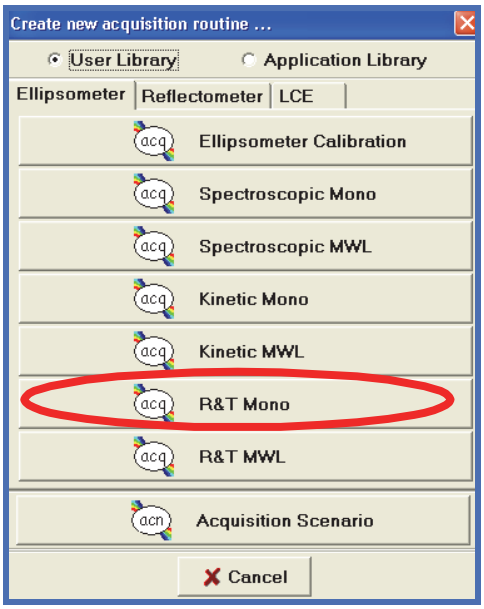
### 3 Acquisition of $S_0$ of your sample

- Drag and drop the experimental measurement of the reference sample to the box named «**Drag-and-drop a reference file here**» (performed in the step 1)
- Drag and drop the reflectance calculation file to the box named «***Drag-and-drop a simulated reference file here***» (performed in the step 2)
- Uncheck the box "***New DC signal reference***"
- Place your sample on the sample stage
- Click on ***Run***
- The measurement gives the reflectance R=f(λ) of your sample

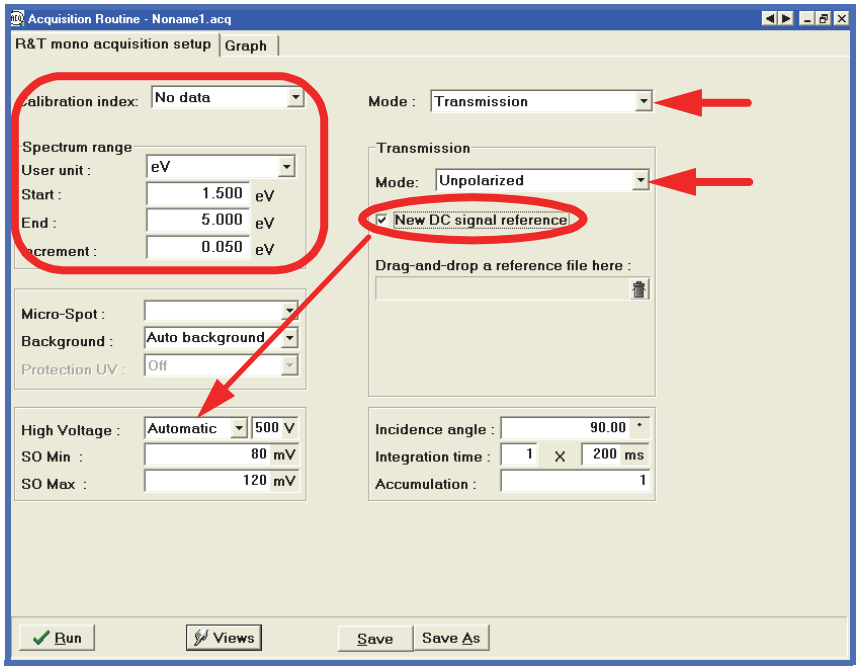
## 5.3 Measuring the transmittance of your sample

- Click on the acquisition tab 

(the seventh button from the left on the top tool bar),
- Select the "R&T mono" Acquisition mode in the menu,




- Select the Transmission mode (the 1st box in the right column) and then the polarized or unpolarized mode for the transmission. You will measure respectively either Tp and Ts or T using these 2 modes.
- Select your spectral range (the other parameters are set by default, you can keep them).
- In the transmission mode, the ellipsometric heads are positioned at 90°, this will be handled automatically by the software.




Transmittance is calculated from the following equation:

$$Transmittance = \left( \frac{S_{0\,experimental}}{S_{0\,reference}} \right)$$

The transmission routine allows to measure these both parameters in two steps.

**1** The 1st step is the measurement of the So reference: the reference sample used is the air.

➔ It consists in the measurement of the So and Hv values of air on the defined spectral range.

**2** The 2nd step is the measurement of the So experimental of your sample.

➔ It consists in the measurement of the So of your sample in the defined spectral range by keeping the values of the High Voltage measured for the air reference in the 1st step.

## 1 Acquisition of $S_0$ reference – the reference sample is measurement of air.

- Check the box New DC signal reference
- When clicking on this box, the automatic mode of the high voltage appears in black. This is set by default and you have to keep it.
- If you want to save this routine, click on save as and enter a name.
- Click on the ***Run*** button. Both ellipsometric heads will move automatically at 90°. The So and Hv values of air will be measured on the spectral range defined.
- When the acquisition is finished, the file is located in the left hand side menu in the folder Results / acquisition data.

# 2 Acquisition of $S_0$ of your sample

- Drag and drop this file to the box named «***Drag-and-drop a reference file here***»
- Uncheck the box «***New DC signal reference***»
- Place your sample on the sample stage
- Click on run
- The measurement gives the transmittance T = f(λ) of your sample

# 6 Reflectance/Transmittance Measurements on AutoSE

## 6.1 Definition

The purpose of this technique is to measure the intensity modification upon reflection or transmission on or through a sample.

$I_r$ = intensity of reflected beam

$I_t$ = intensity of transmitted beam

Typically given as relative quantities:

$$R = \frac{I_r}{I_0}$$

$$T = \frac{I_t}{I_0}$$

with $I_0$ = Intensity of incident beam


**Change of polarization upon reflection (or transmission)**

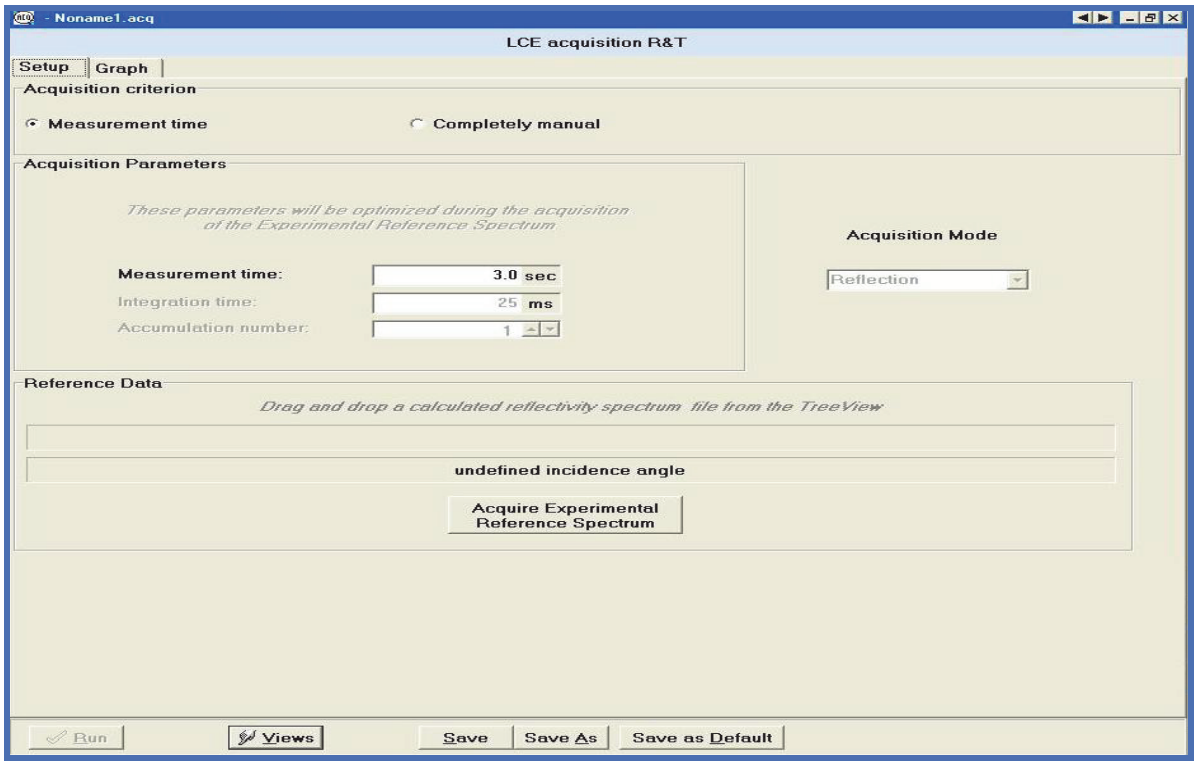
$$\rho = \frac{R_p}{R_s} = \tan\Psi \cos i\Delta$$

p = parallel to plane of incidence

s = perpendicular to plane of incidence

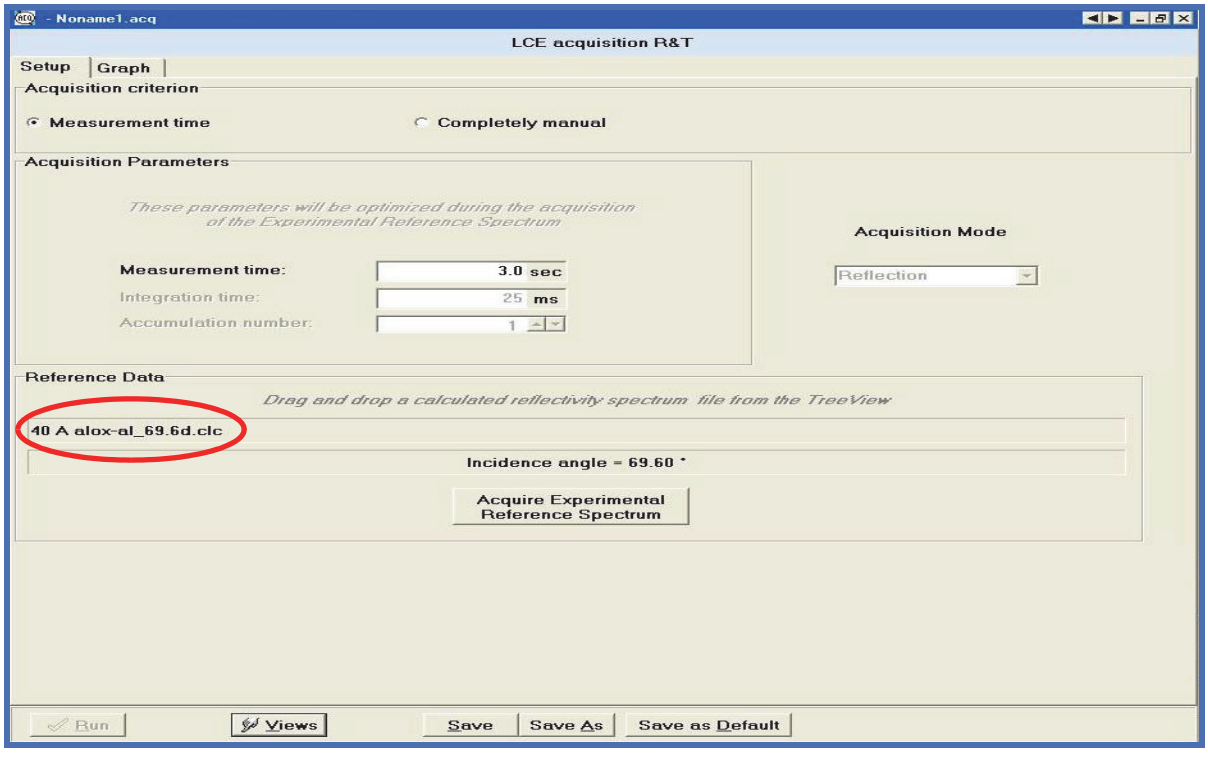
## 6.2 Measuring the reflectance of your sample

1. Create a new "***LCE R&T acquisition routine***"
2. Select "***Reflection***" Acquisition mode
3. Check "***Measurement Time***" for acquisition criterion
4. Enter the measurement time value.




**5** In the ***Reference data*** area, drag and drop a simulated file R=f(λ) of your reference sample at the AOI of your system (for transmission the simulated data not needed).

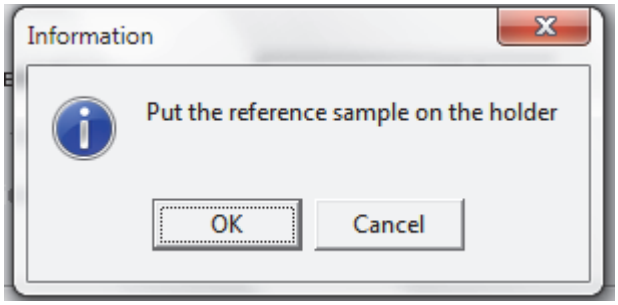
The screen shown below is displayed



**6** Then, put your reference sample on the stage and using «***Auto SE views***»

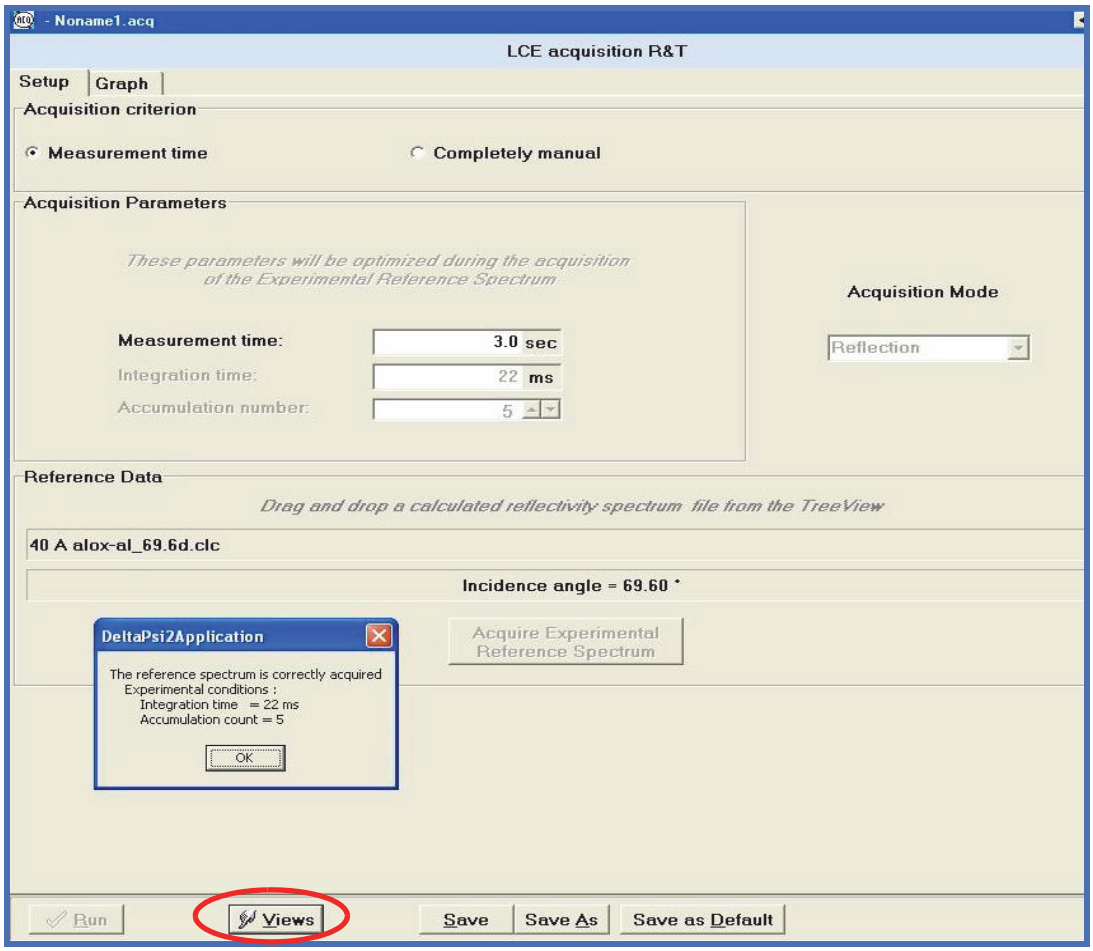
- Choose the desired spot size. (It must be the same for the reference spectrum & sample)
- Adjust & optimize the height of the stage

**7** Then, click on «***Acquire Experimental Reference Spectrum***» button, and click on «***OK***» if your reference sample is correctly located.




**8** The measurement of reference sample is started.

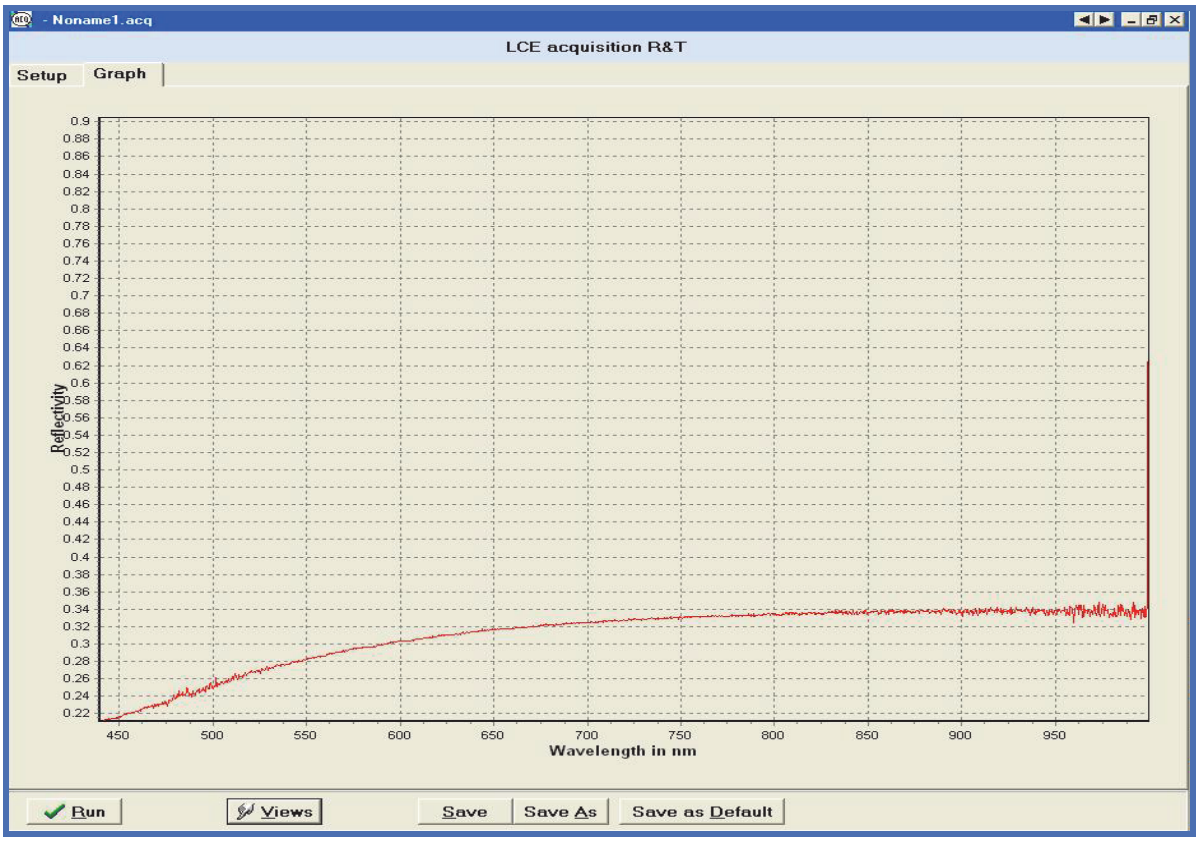
**9** Then an information screen is displayed with the integration time & accumulation number.




**10** Then, put your sample and adjust the height of the stage,

**11** And click on "***Run***" button.

**12** The measurement is launched.




## 6.3 Measuring the transmittance of your sample

**The measurement of transmittance T is performed the same way as for the reflectance measurement. The difference is the necessity to use a specific accessory to put the sample on the stage and there is no need of reference file.**

# 7 Reflectance/Transmittance Measurements on UVISEL2

## 7.1 Definition

The purpose of this technique is to measure the intensity modification upon reflection or transmission on or through a sample.

$I_r$ = intensity of reflected beam

$I_t$ = intensity of transmitted beam

Typically given as relative quantities:

$$R = \frac{I_r}{I_0}$$

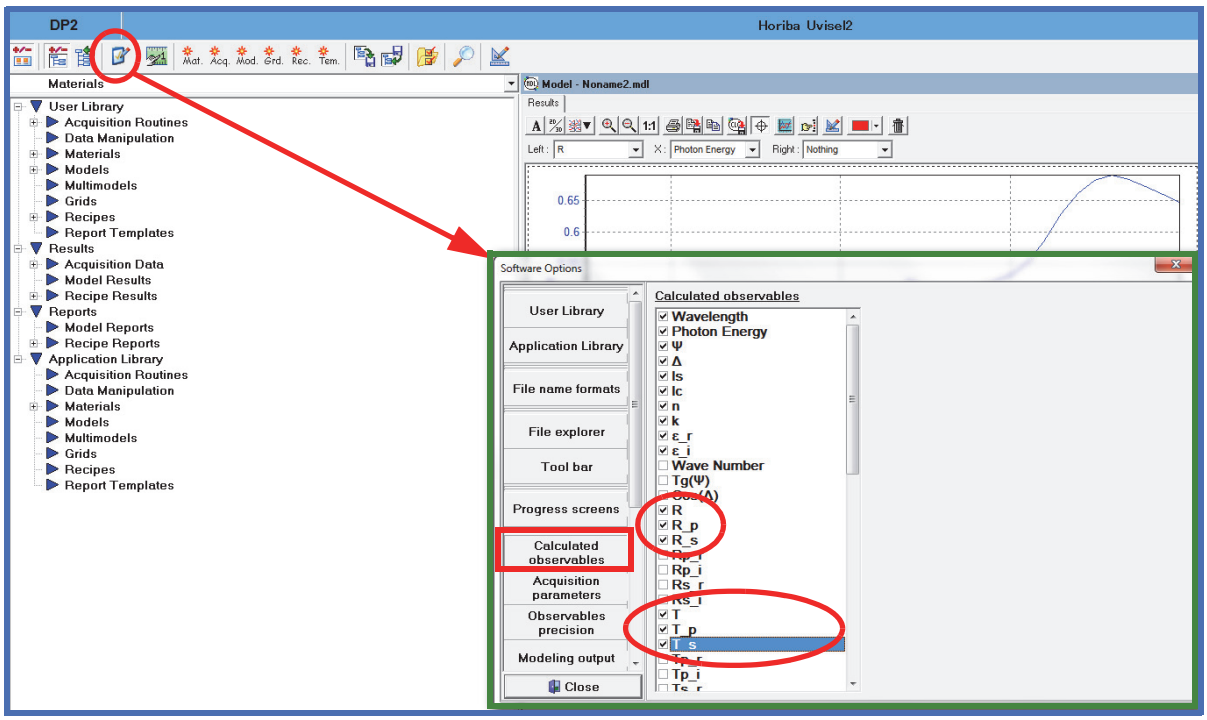
$$T = \frac{I_t}{I_0}$$

with $I_0$ = Intensity of incident beam


**Change of polarization upon reflection (or transmission)**

## 7.2 Preliminary

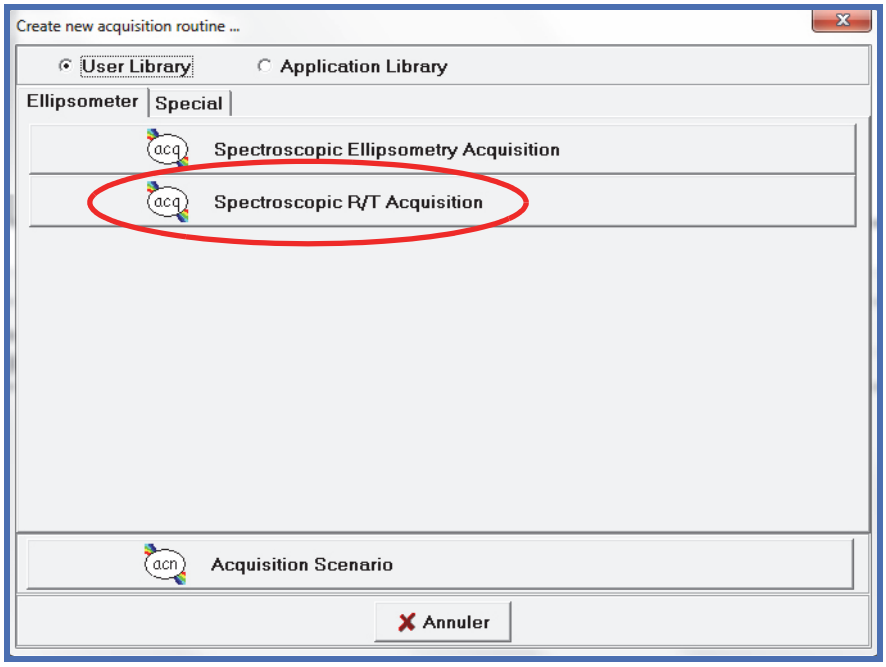
- Before any action on the Reflectance/Transmittance measurements, verify that the R, Rp, Rs, T, Tp and Ts observables are ticked in the ***Software Options*** setup (see figure below).




## 7.3 Measuring the reflectance of your sample

- Click on the acquisition tab 

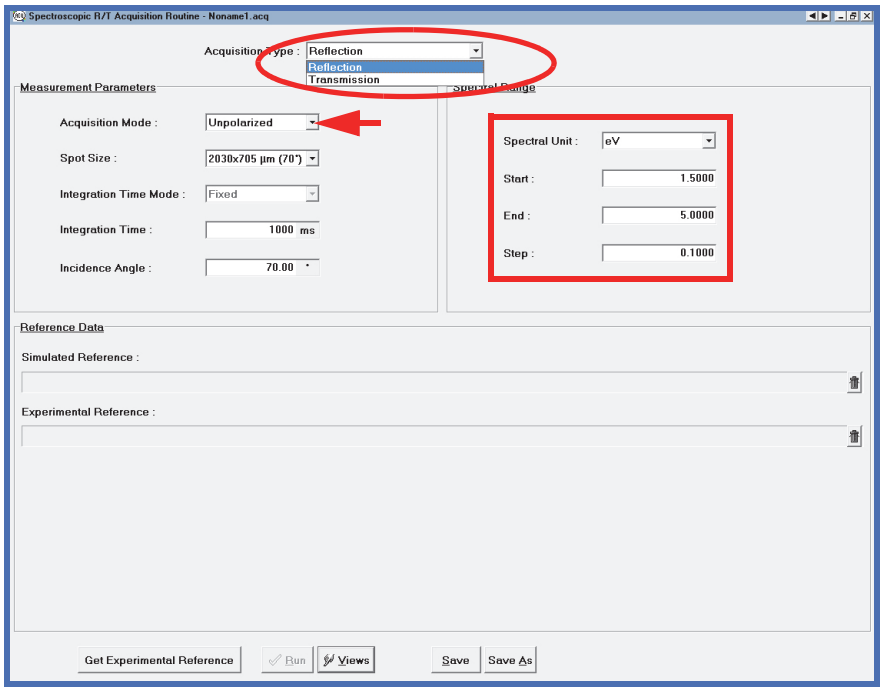
(the seventh button from the left on the top tool bar).
- Select the «***Spectroscopic R/T Acquisition***» Acquisition mode in the menu.




- Select the Reflection mode (the 1st box in the middle) and then the polarized or unpolarized mode. You will measure respectively either Rp and Rs or R using these 2 modes.

05/20/2016

- Select your spectral range and the others parameters according to your sample characteristics.




- To perform the Reflectance measurement you need a ***Simulated Reference*** file. If you don't have this file, the procedure below shows how to create it.

- At this step, there are two possibilities: you know well your sample and then you can create the **Model** of your sample. If you don't know your sample and precisely the layers thicknesses, you must create a **Model** and launch an **ellipsometric measurement** to find the real layers thicknesses of your sample.

**About the Simulated Reference file required before Reflectance measurement**

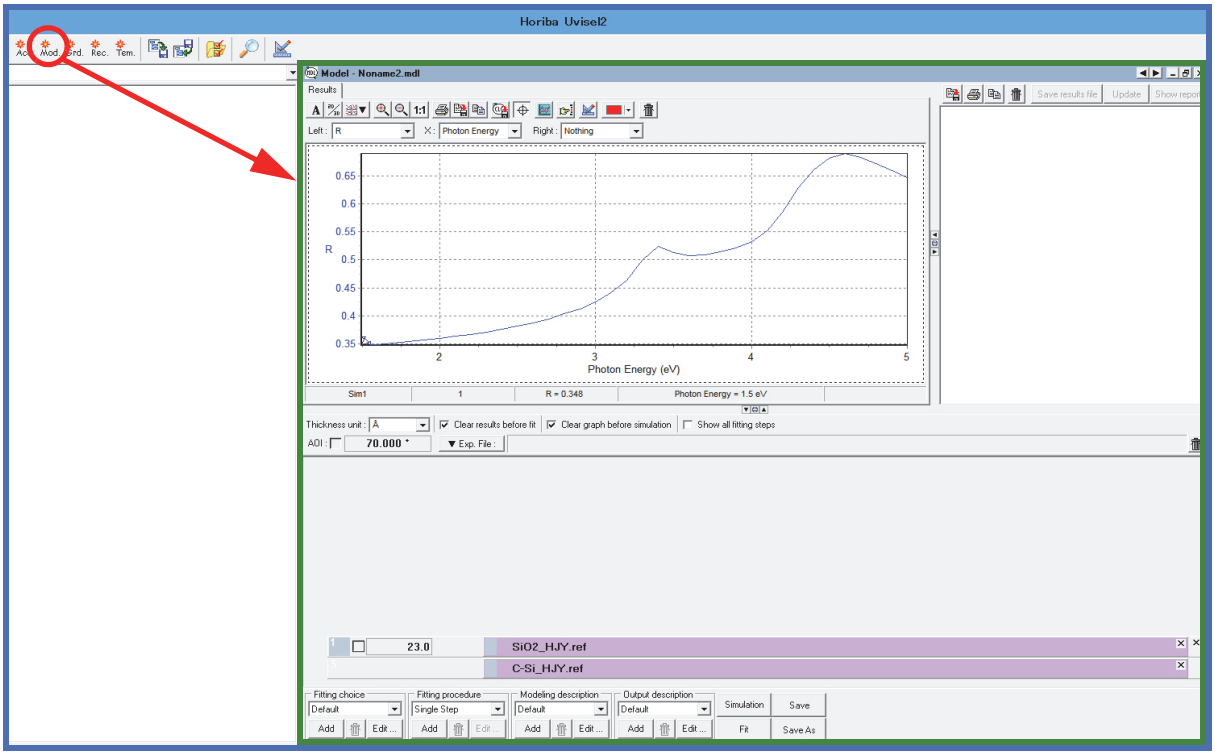
| You already have a **Simulated Reference** file of your sample | You don't have a Simulated Reference file of your sample but **you know** precisely the optical constants and thicknesses of the layers | You don't have a **Simulated Reference** file of your sample but you **don't know** precisely the optical constant and thicknesses of the layers |
|---|---|---|
| Go to the **Step 2** | Go to the **Step 1** | Go to the **Step 3** |

**Better you know the real thicknesses of your Reference sample, better will be the Reflectance measurement accuracy.**

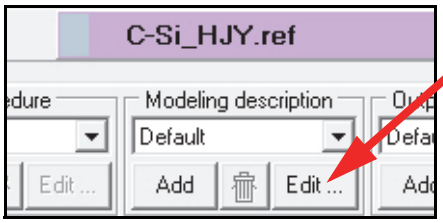
*05/20/2016*

# 1 Simulated file creation

- Click on the Model icon  located in the upper left hand of the icon bar. As shown on the windows below, you have to fill and setup each layer of your Model.
  Follow the Model building detailed in the chapter **"Model Building" on page 166**.




- Once the layers have been defined (components, thicknesses, AOI etc.),

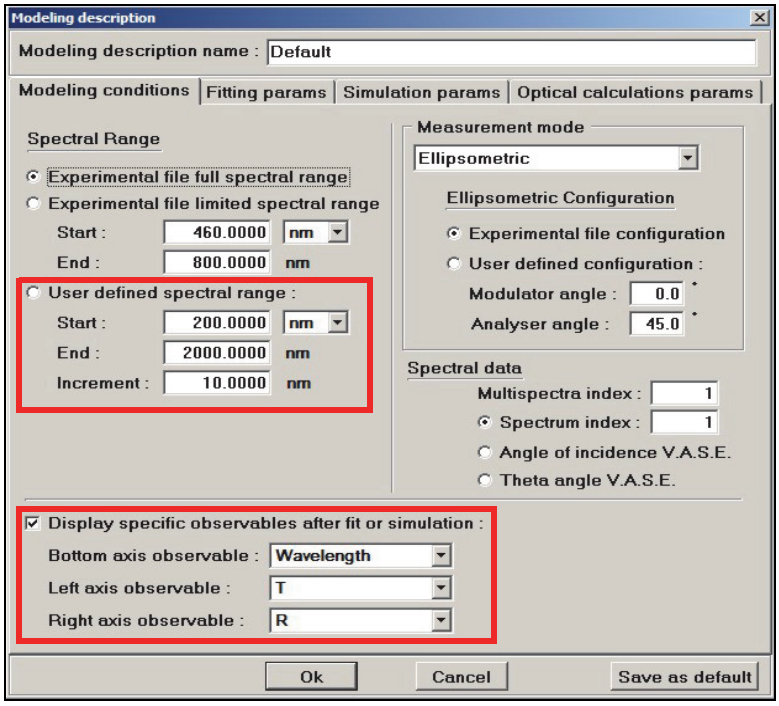



**Click here to open Simulation Setup Screen**

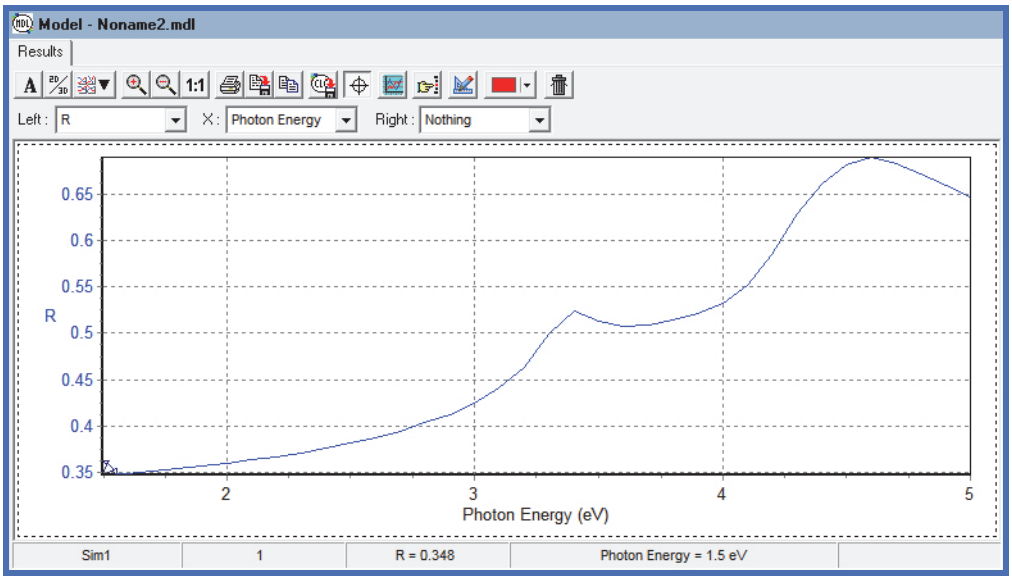
- Click on the Edit button (***Modeling description***) to open the setup screen. From the ***Modeling Condition*** tab screen, tick on the ***User defined spectral range*** and set the values (these values must be the same for the reference measurement and sample R measurement). Tick on also and select the observable to display after the simulation.

05/20/2016

- Once done, click OK to validate.

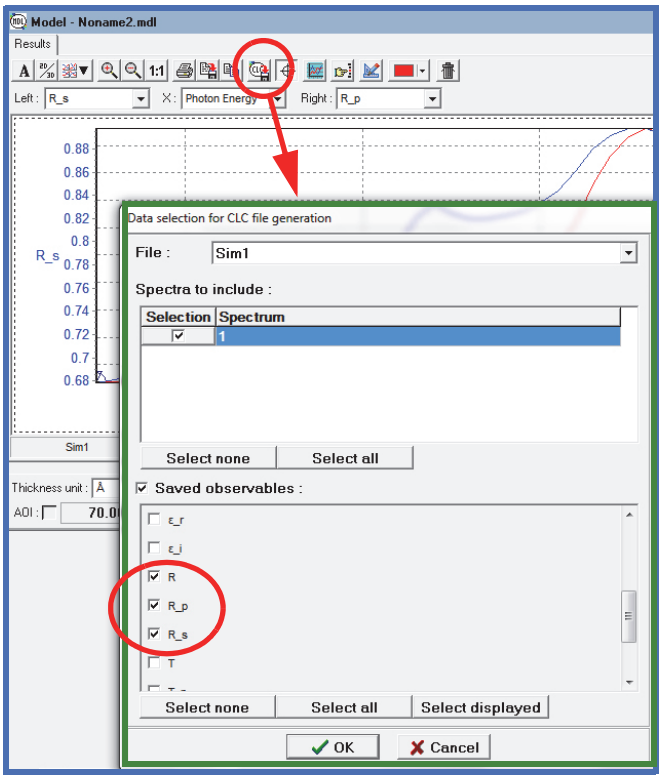



- From the ***Model*** window, click on the Simulation button to launch the simulation process which will end with the result curve (see below).

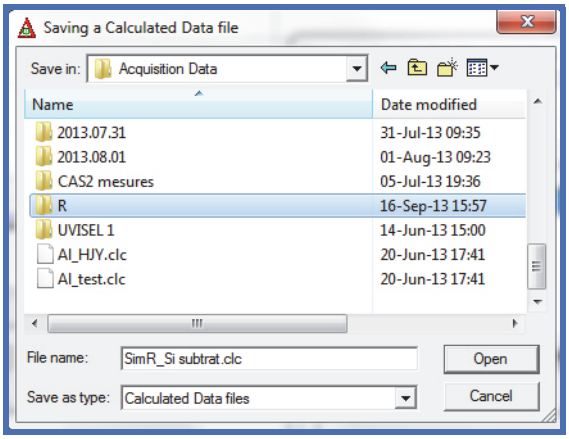



05/20/2016

- From the ***Model*** window, click on the CLC icon  . The ***Data selection for CLC file generation*** window will be opened (see below). Select the observables which will be saved with the simulated file. Tick on the R, Rp and Rs observables, then click OK to validate.

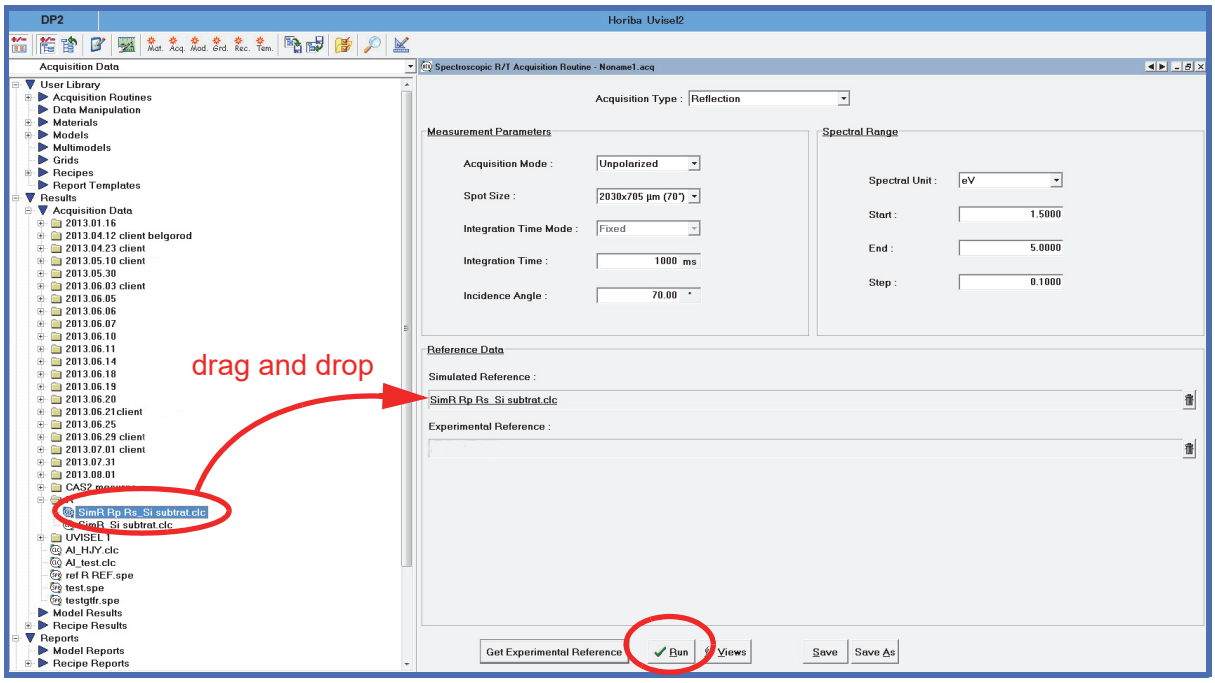



- A new pop-up screen will ask you for a name of this «Calculated Data» file. The file will be saved in the selected folder.



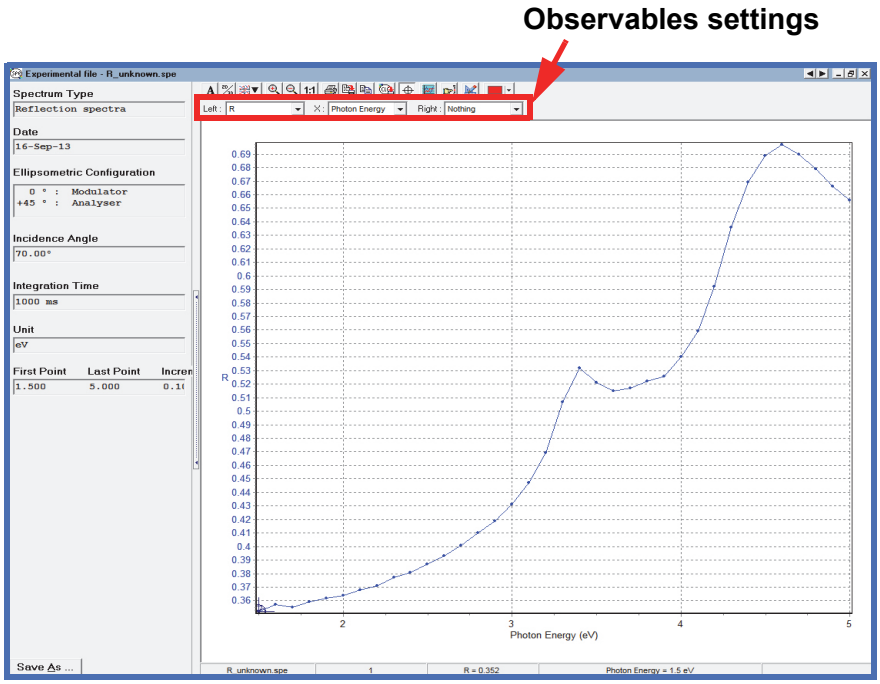

# 2 Reflectance measurement

- Open back the «Spectroscopic R/T acquisition routine» window. In the TreeView, find the previously saved «*Reference Simulated*» data file (.CLC file), then drag and drop it to the Simulated Reference box as shown on the figure below:




- Click on ***Get Experimental Reference*** button: the $S_0$ and HV will be acquired. At the end, the **Run** button will become accessible (not greyed).

- Click on the ***Run*** button to launch the Reflectance measurement. The results will be then displayed and a generic xx.spe filename is generated. The observables of the curve can be set at will by selecting them in the frame located above the curve.



3 **Model Creation with thicknesses not defined**

- To obtain better Reflectance measurements, it is necessary to create a ***Simulated Reference*** file with real known layers thicknesses. If your sample is not well known, the better way is to perform a spectroscopic ellipsometric measurement of your sample.

- Follow the chapter "Modeling", page 157 and build the model and fit it to obtain the layers thicknesses. At the end the .CLC file must be created and will be used as ***Simulated Reference*** file. See the end of the **Step 1** "Simulated file creation", page 102. Then follow the **Step 2**.

## 7.4 Measuring the transmittance of your sample

### 7.4.1 Principle

- In the transmission mode, the ellipsometric heads are positioned at 90°, this will be handled automatically by the software.
  Transmittance is calculated from the following equation:

$$Transmittance = \left( \frac{S_{0\,experimental}}{S_{0\,reference}} \right)$$

The transmission routine allows to measure these both parameters in two steps.

**1** The 1st step is the measurement of the So reference: the reference sample used is the air.

➔ It consists in the measurement of the So and Hv values of air on the defined spectral range.

**2** The 2nd step is the measurement of the So experimental of your sample.

➔ It consists in the measurement of the So of your sample in the defined spectral range by keeping the values of the High Voltage measured for the air reference in the 1st step.

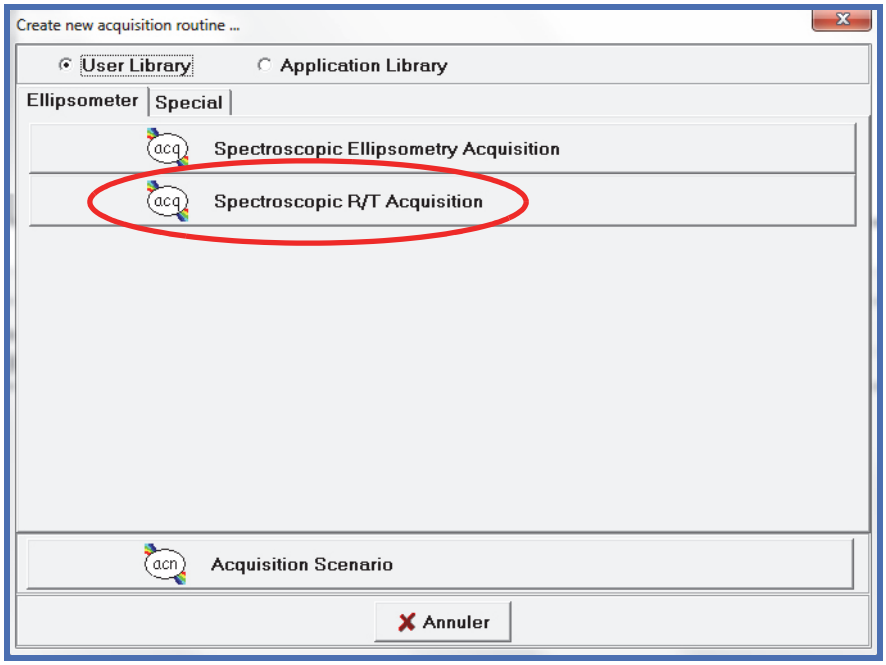
### 7.4.2 Procedure

The «Transmittance» procedure is very similar to the Reflectance measurement. All the steps are the same with the exception of the ***Simulated Reference*** file which is not necessary for the transmittance measurement.

- Click on the acquisition tab 

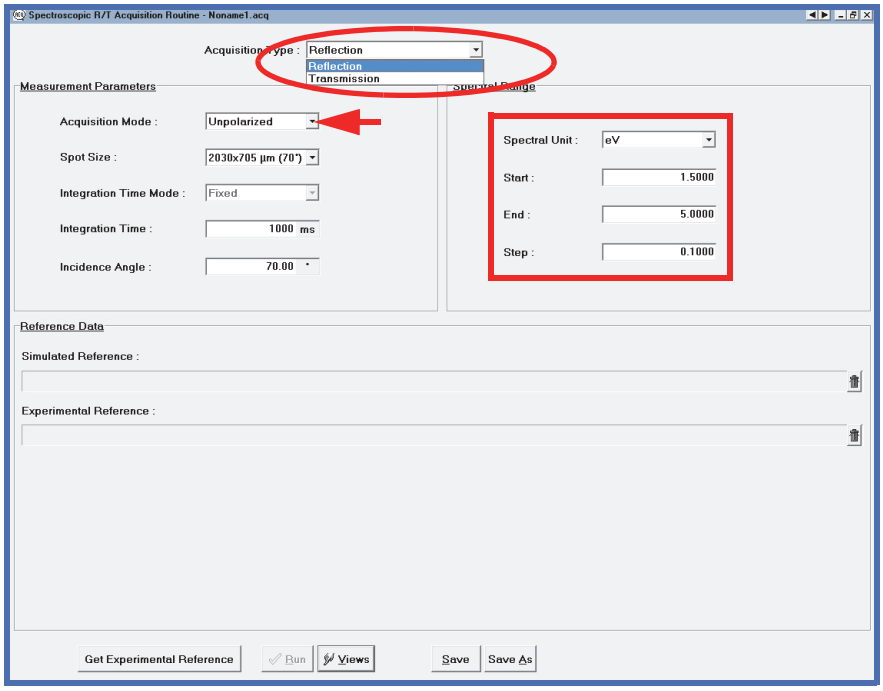
(the seventh button from the left on the top tool bar).

- Select the «***Spectroscopic R/T Acquisition***» Acquisition mode in the menu.



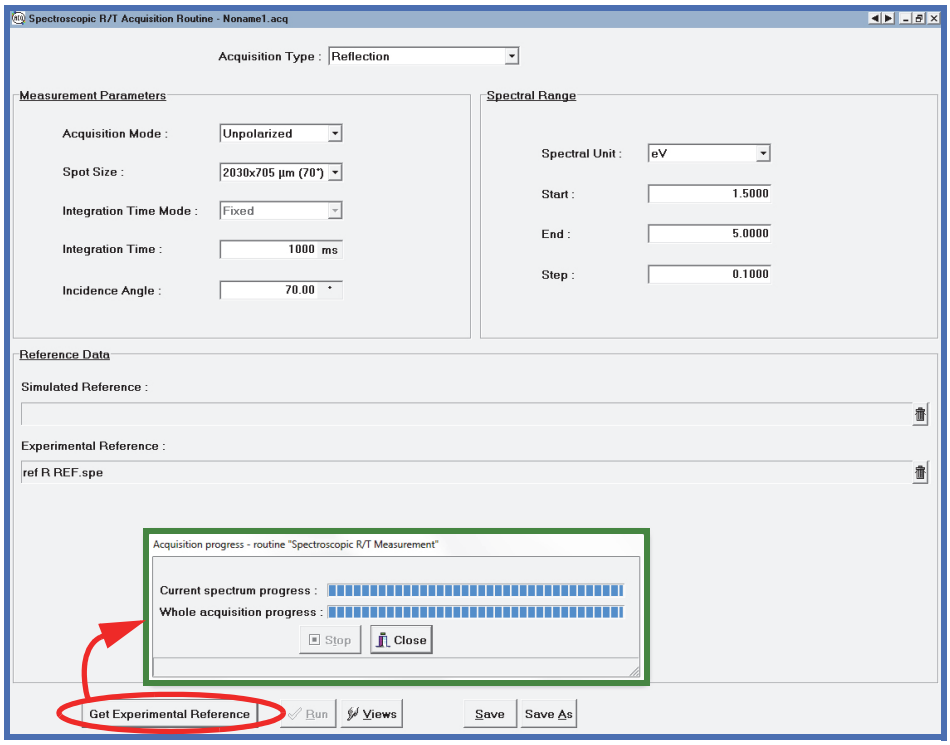

- Select the Reflection mode (the 1st box in the middle) and then the polarized or unpolarized mode. You will measure respectively either Tp and Ts or T using these 2 modes.

- Select your spectral range (the other parameters are set by default, you can keep them).




- To measure the transmittance of a sample, you **don't need** any Simulated Reference file. Click on ***Get Experimental reference***.

- Then click on the Run button to launch the transmittance measurement.


Spectroscopic R/T Acquisition Routine - Noname1.acq
Acquisition Type : Reflection
Measurement Parameters
Acquisition Mode : Unpolarized
Spot Size : 2030x705 µm (70°)
Integration Time Mode : Fixed
Integration Time : 1000 ms
Incidence Angle : 70.00 °
Spectral Range
Spectral Unit : eV
Start : 1.5000
End : 5.0000
Step : 0.1000
Reference Data
Simulated Reference :
Experimental Reference :
ref R REF.spe
Acquisition progress - routine "Spectroscopic R/T Measurement"
Current spectrum progress :
Whole acquisition progress :
Stop
Close
Get Experimental Reference
Run
Views
Save
Save As

05/20/2016

# 8 Simulated Acquisition

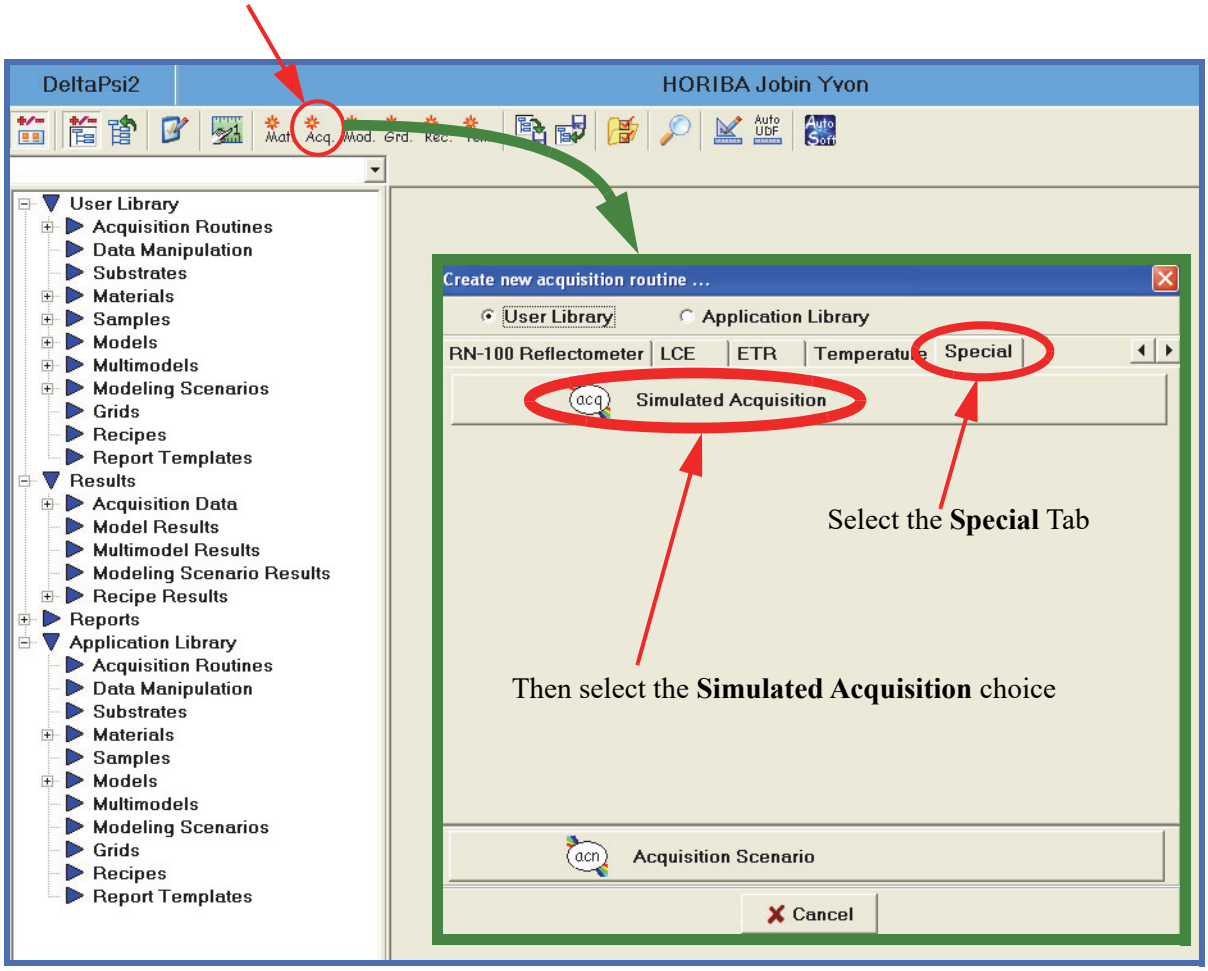
## 8.1 Definition

The Simulated Acquisition is a useful tool which converts an already acquired experimental data or a Recipe to a data source. This action can be done on a computer which is not connected to an ellipsometer.

## 8.2 Accessing

There are two ways to access to the ***Simulated Acquisition*** window:

Click on the ***Acq*** icon located on the Main icons bar

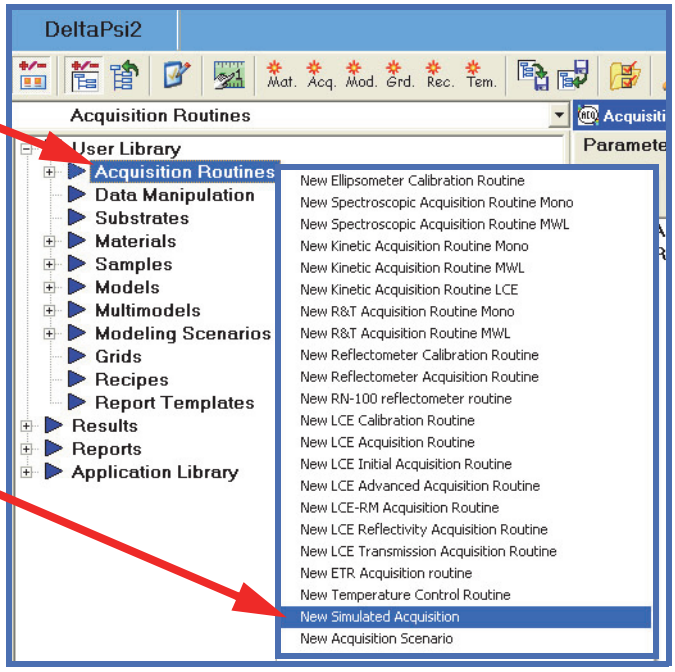



05/20/2016

**OR**

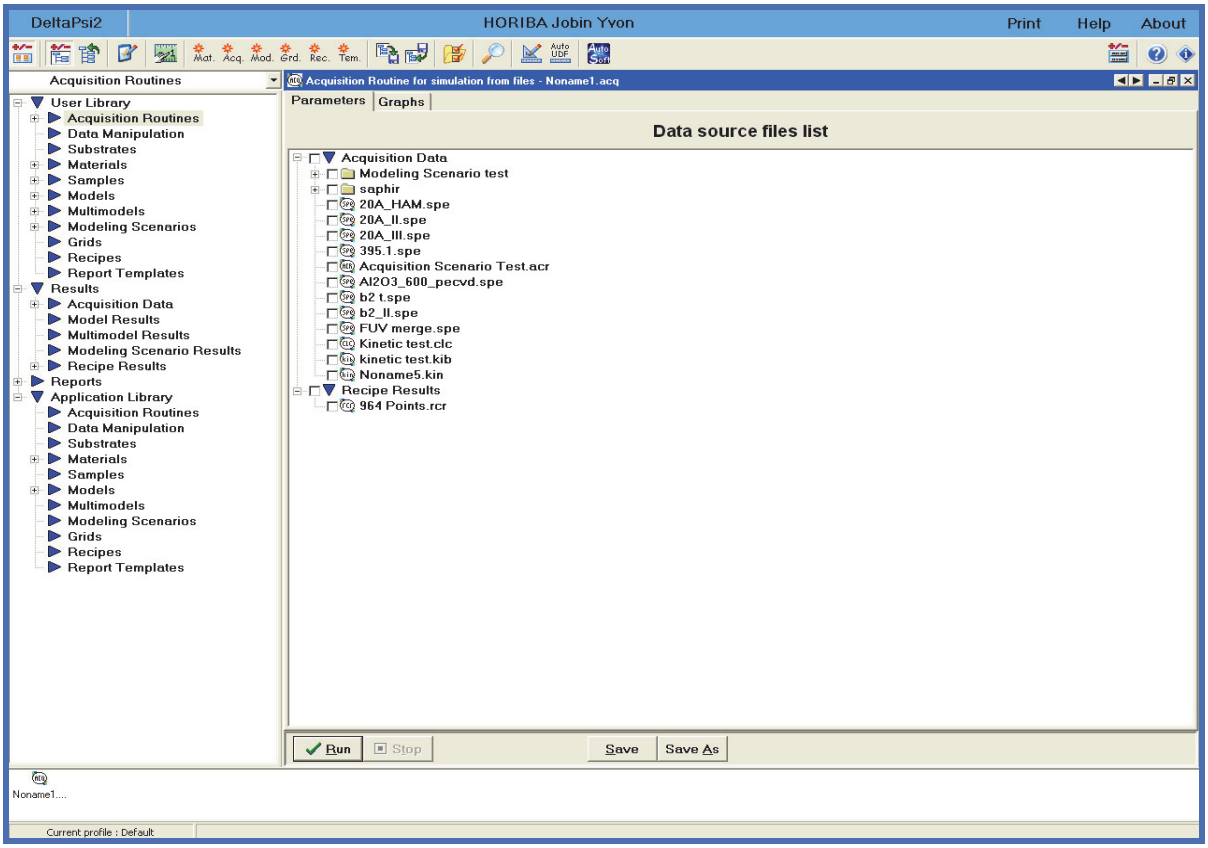
Right-Click on the Acquisition Routine choice

Then, select the New Simulated Acquisition choice




## 8.3 Description

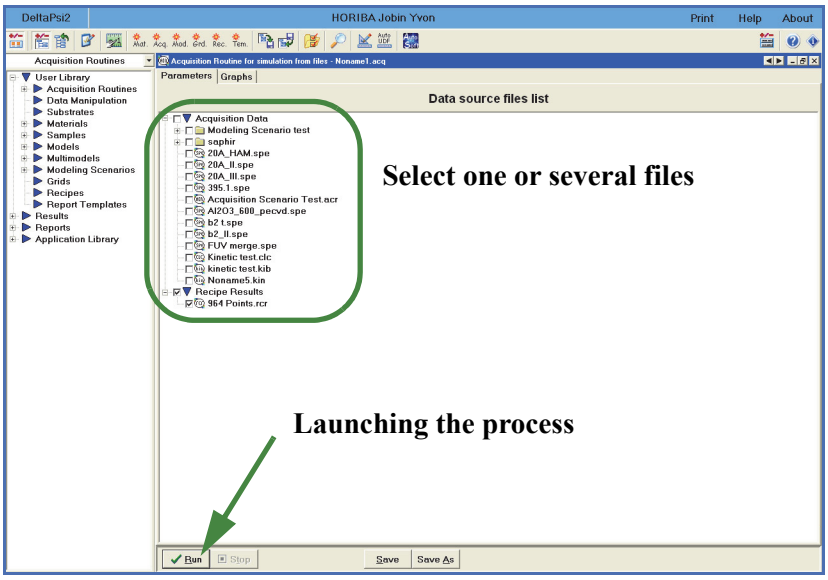
The following screen will be displayed:

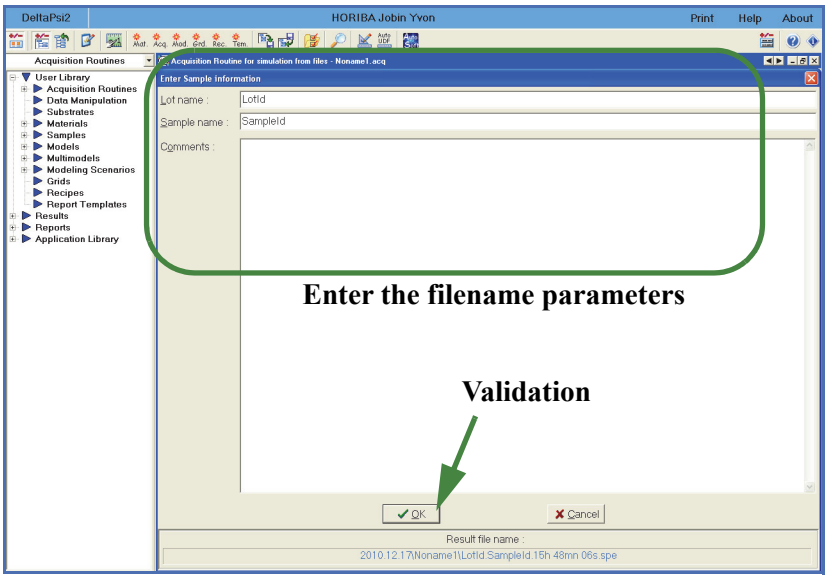


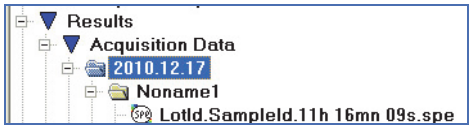
As shown on the figure above, the main window displays the **Acquisition Data** and **Recipe Results** tree. It becomes easy to select the files to process.

Once selected, click on the ***Run*** button to generate a new **Acquisition Data** which is automatically saved in a new folder labelled with the current date under the **Acquisition Data** folder.

The following figures summarizes the actions.



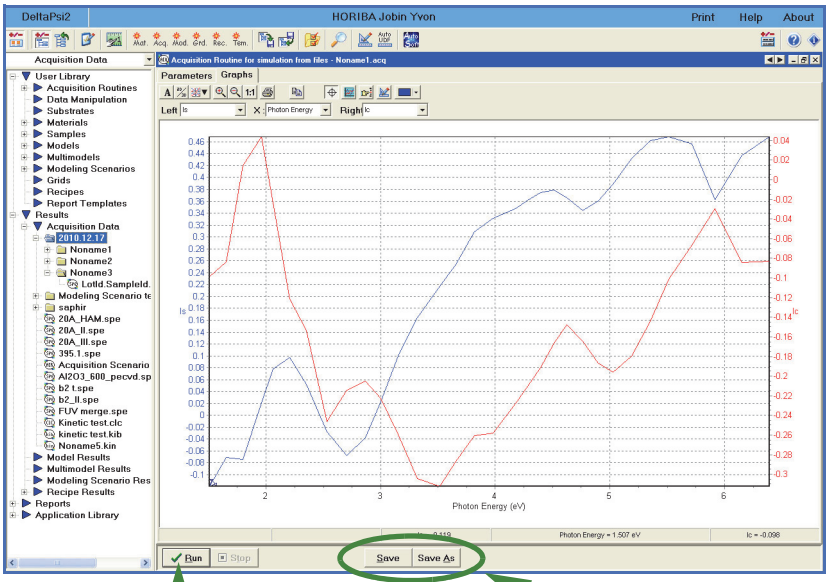




This figure shows where and how the new Acquisition Data is saved in the DP2 TreeView. Please notice how the sample information entered before the process is implemented to the data file.

05/20/2016

**Results**:




**Save the Acquisition Routine**

Click on the ***Save As*** button to save the new Acquisition Routine extracted from the Acquisition Data. This Acqusition Routine is saved in the Acquisition Routine folder.

**Launch the next Acquisition Data to process (if several Acquisition Data have been selected or in case of Recipe).**

**The Simulated Acquisition routine is programmed as a loop. To avoid the same process from the beginning of the selected files, it is recommended to write the checked files in the same order than the TreeView to know which data is currently processed.**

**NOTICE**: a recipe result file with N steps provides N data sources and is equivalent to N acquisition result files.

*05/20/2016*

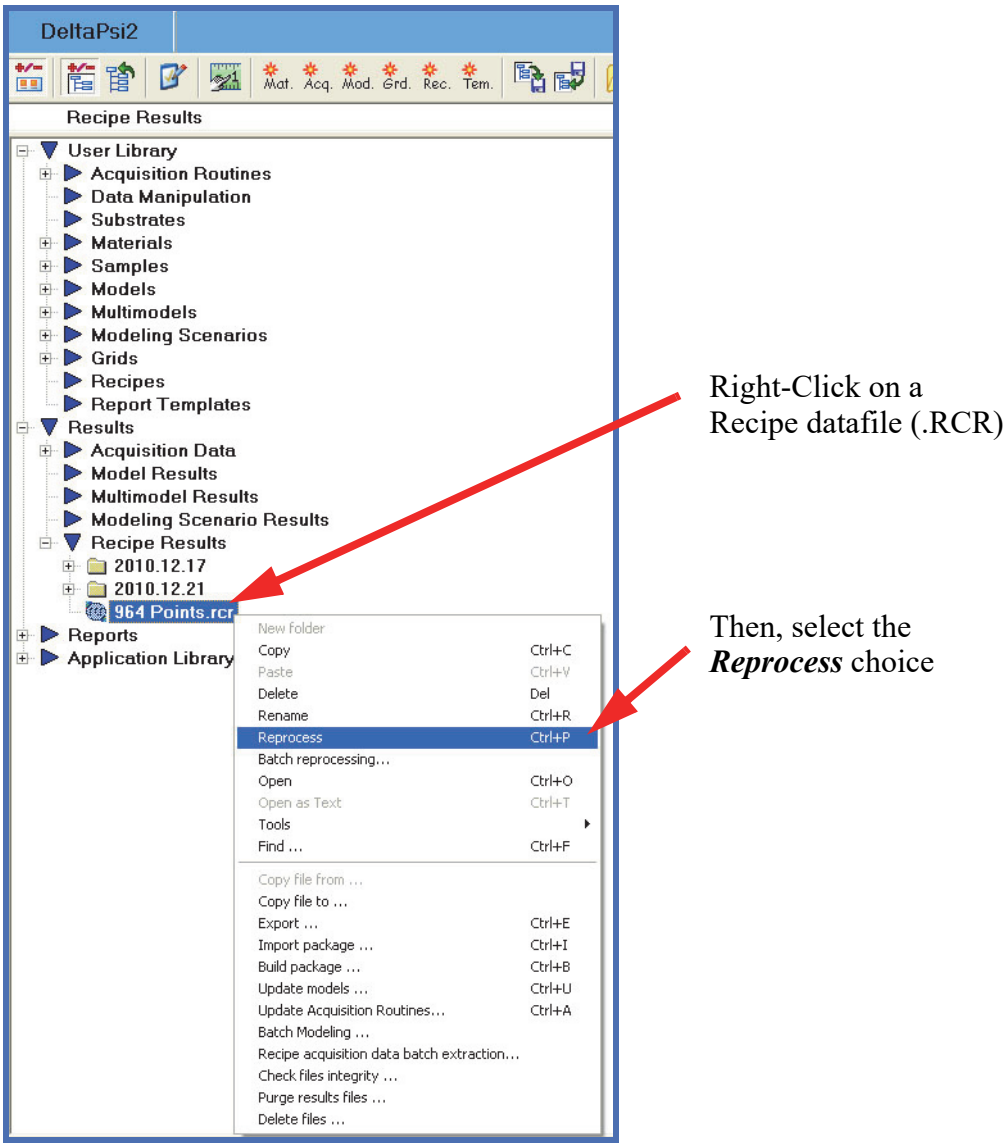
# 9 Reprocessing

## 9.1 Definition

The Reprocessing is a useful feature which reprocesses resulted data (including acquisition and modeling) from a new model. This can be done with a single Immediate Recipe or a Mapping Recipe. With this feature, it becomes possible to sharpen a result by using another model; this action does not require to be connected to an ellipsometer, a DP2 software installed on a PC is enough.

## 9.2 Accessing

From the TreeView, right-click on a ***Recipe Results*** **.RCR** file and choose the ***Reprocess*** choice:



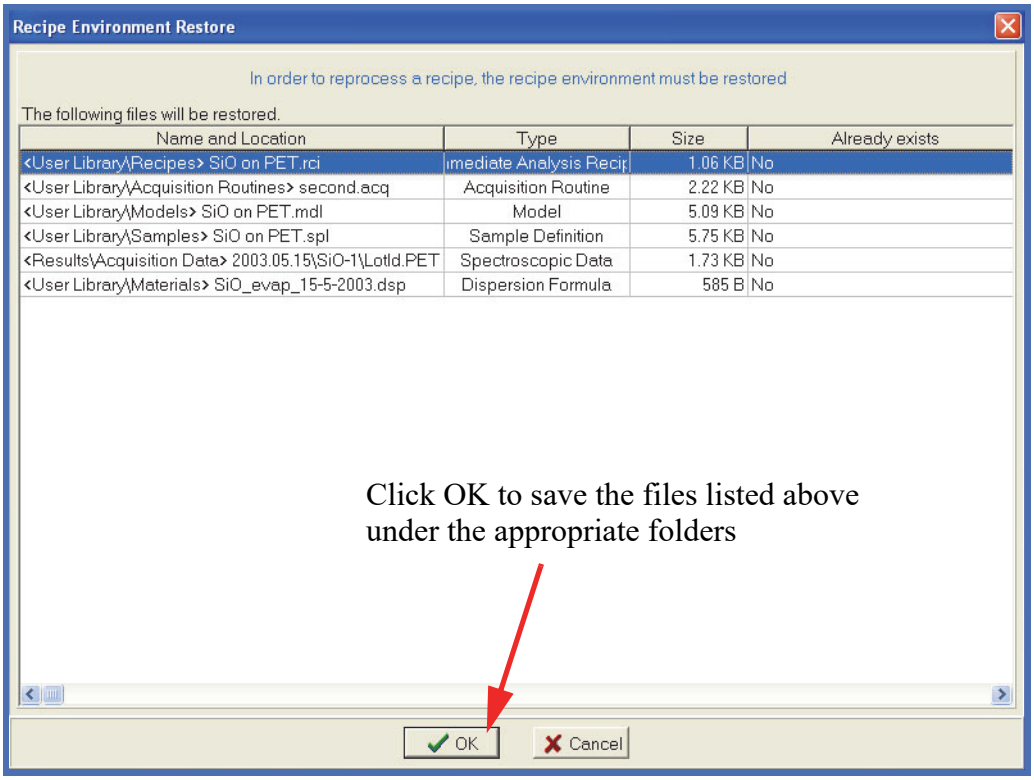

05/20/2016

## 9.3 Description

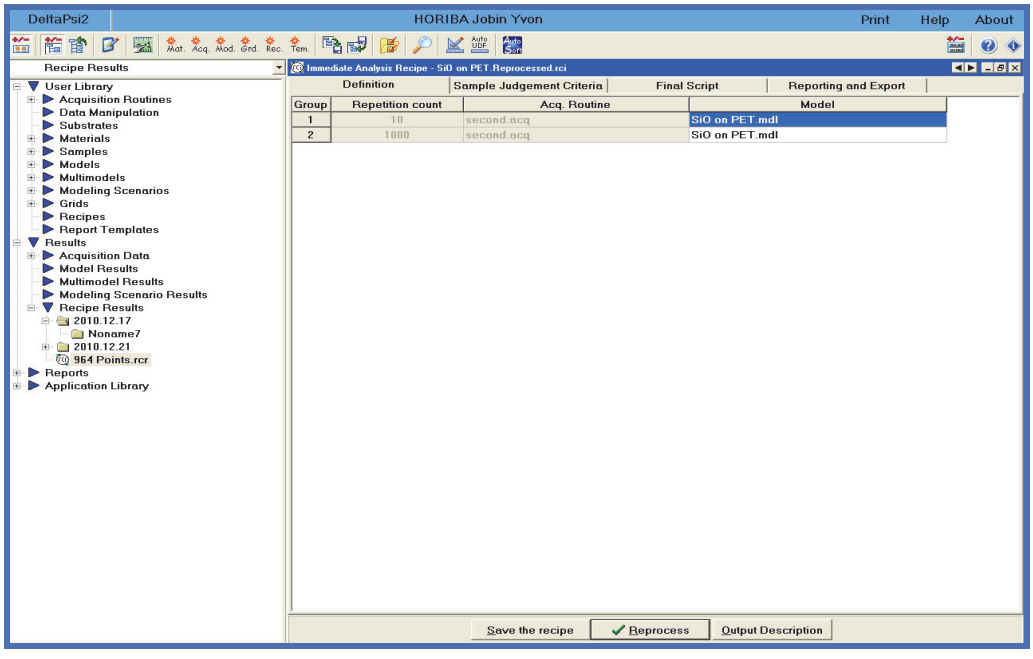
Once the .RCR file is selected, the ***Recipe Environment Restore*** window will be displayed. This window lists the files need to run the Recipe and which are required for a reprocessing. These files are generated from the .RCR file and will be saved under the appropriate folders of the TreeView.

**CAUTION**: do not overwrite already existing files with the same name. To prevent this, the last column indicates if a same filename is already existed.

Click on the **OK** button to save the files to the TreeView.



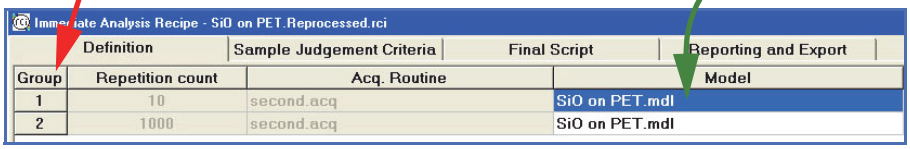

Once done, the following window will be displayed:

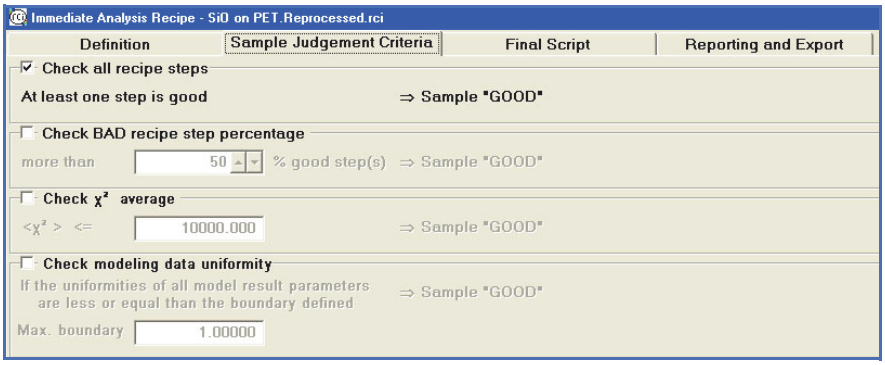


The windows shows the identification of the Recipe, here is an ***Immediate Analysis Recipe***. Below, four tabs displays the Recipe parameters. The parameters which can be modified for a reprocess are accessible. Of course, the acquisition parameters, the number of data acquisition, the acquisition routine cannot be modified. The parameters which concerns the post acquisition process, which includes the model characteristics or the statistics and acceptance criteria can be changed.

Two groups of samples

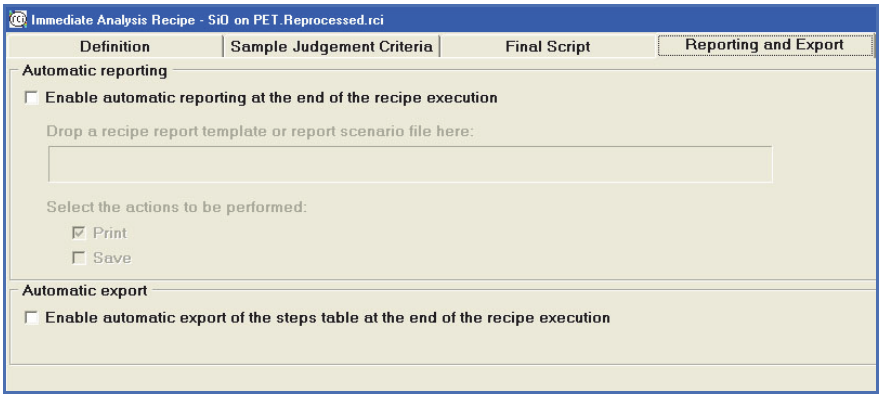
Drag and drop a new model to reprocess




Changing the Judgement criteria

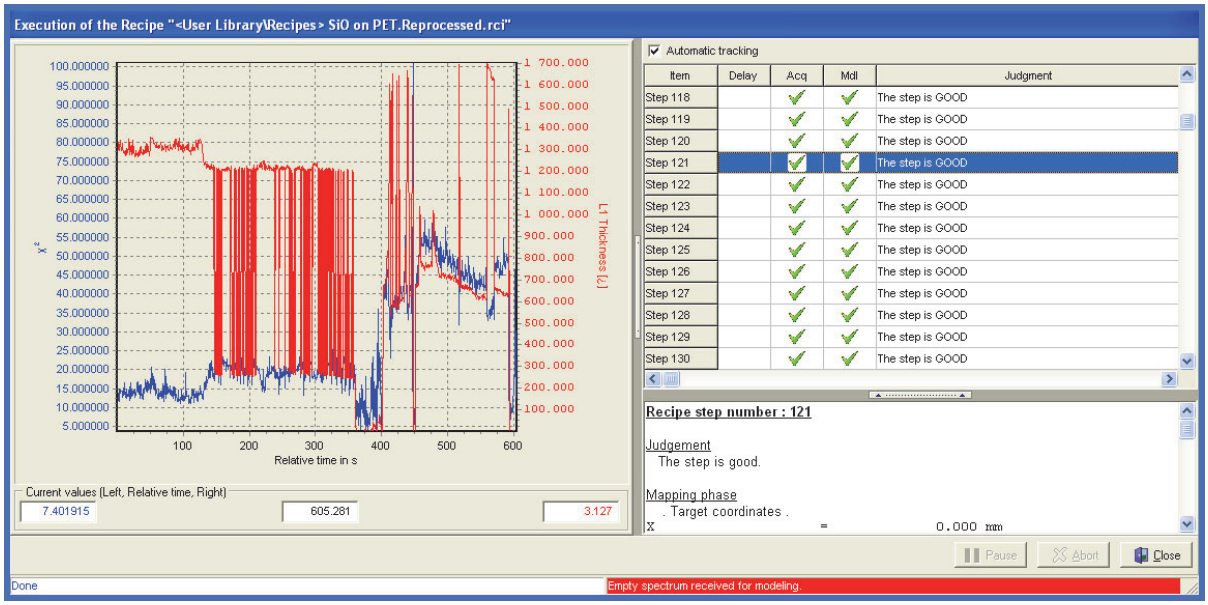

Additional Script activation
A final script can be added. Click on the ***Edit script*** button to display the script editor with a context semi-automatic programming




Automatic reporting enabling

Automatic export enabling

Once the parameters have been setup, click on the ***Reprocess*** button to launch the reprocessing. The figure below shows an example of the final screen results.




The results are automatically saved in the ***Recipe Results*** folder using the syntax defined from the ***Software Options «File name formats»*** (see "File name formats" on page 15).

The figure below shows an example of folders and files generated by the reprocessing feature.


on 27th december 2010, four reprocessings have been generated from the SiO on PET file

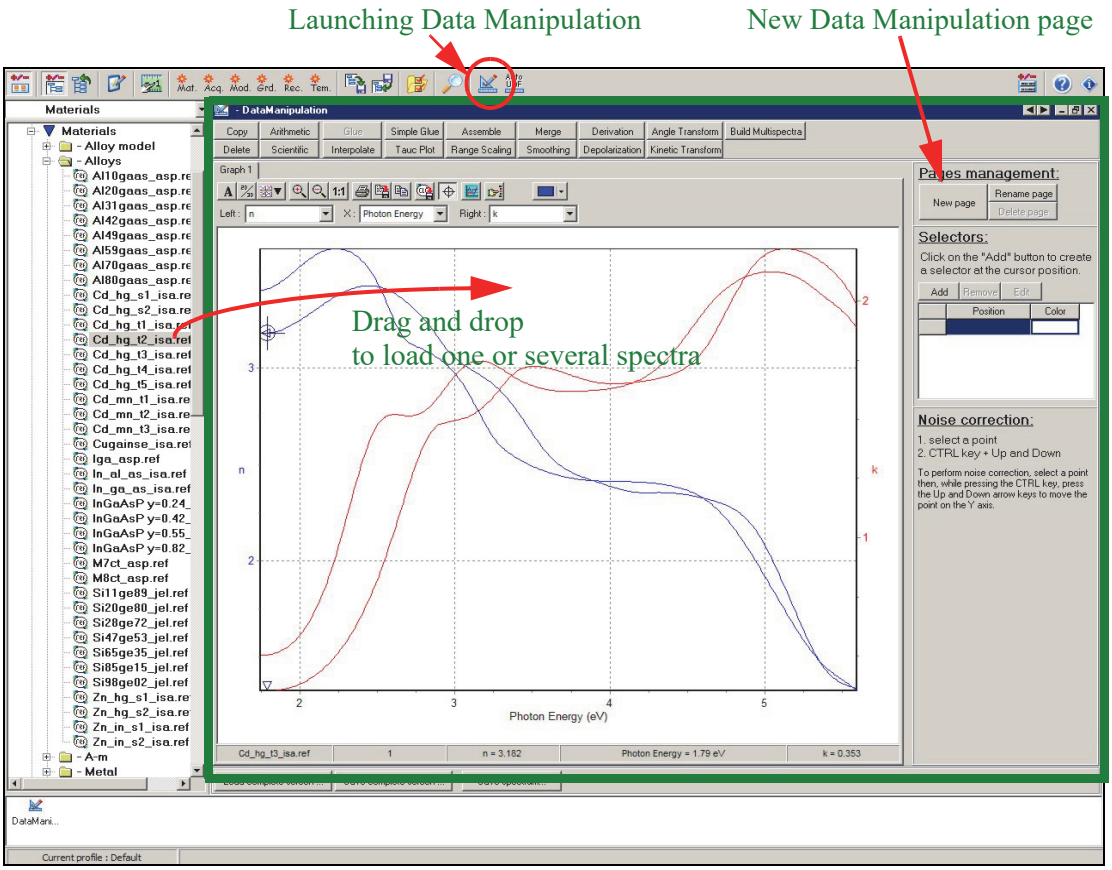
05/20/2016

# 10 Data Manipulation

The Data Manipulation is a powerful feature which allows a user to modify spectra. This chapter describes all of the parameters and resulted effects.

## 10.1 Accessing

Click on the Data Manipulation icon to display the data Manipulation screen.




The ***Save Spectrum*** and ***CLC*** button perform the same action: save current data to «Calculated» type file.

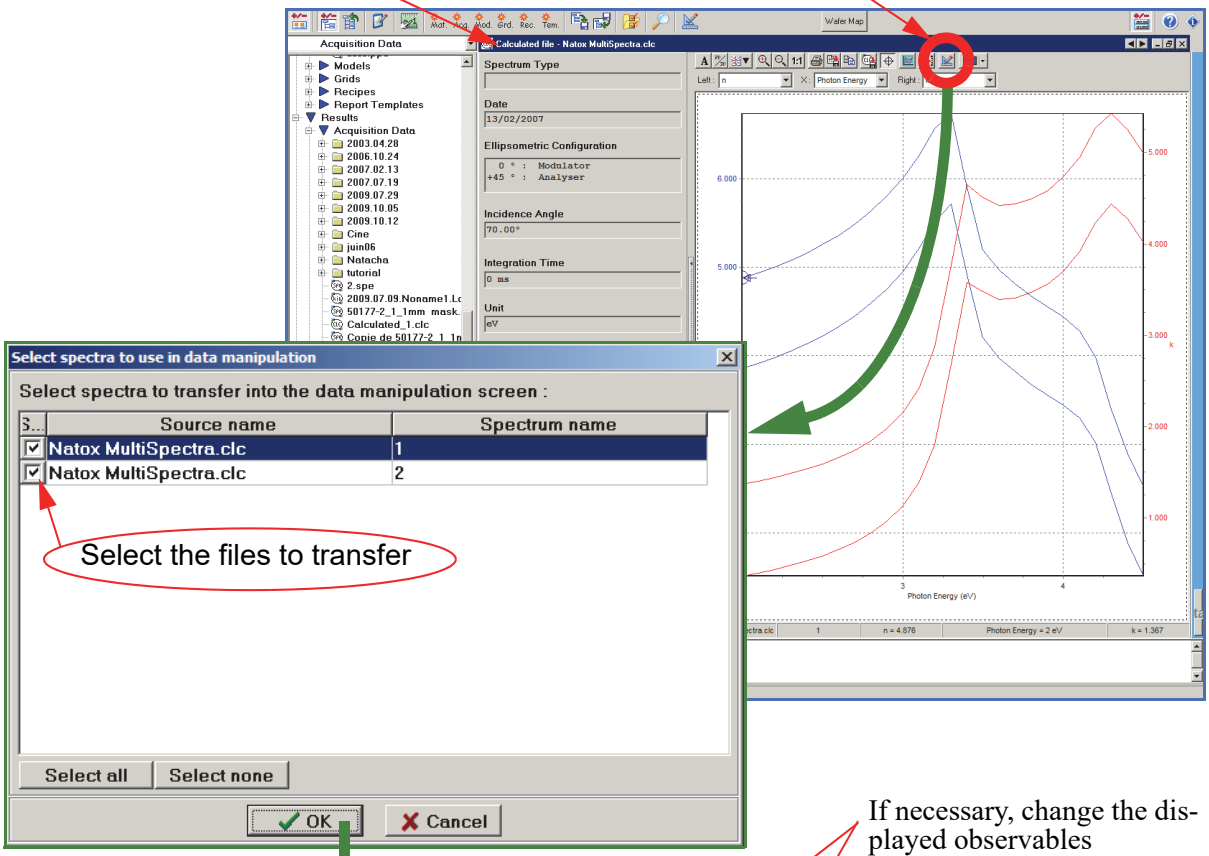
**IMPORTANT**: Access to Data Manipulation is also possible from each DP2 graphics screen by clicking on the same icon located above the graphs window. In this case the displayed graphs will be automatically retrieved inside a new Data Manipulation window. The figure below shows the procedure.

**NOTICE**: on the bottom main screen, three buttons perform useful general actions. The ***Save complete screen*** allows to save the manipulation screen, as it, with all the "Graph X" tabs. This is useful if the work is not finished and must be continue later. The ***Load complete screen*** will allow to retrieve the complete Manipulation screen already previously saved. The ***Save spectrum*** button saves the current Graph tab.

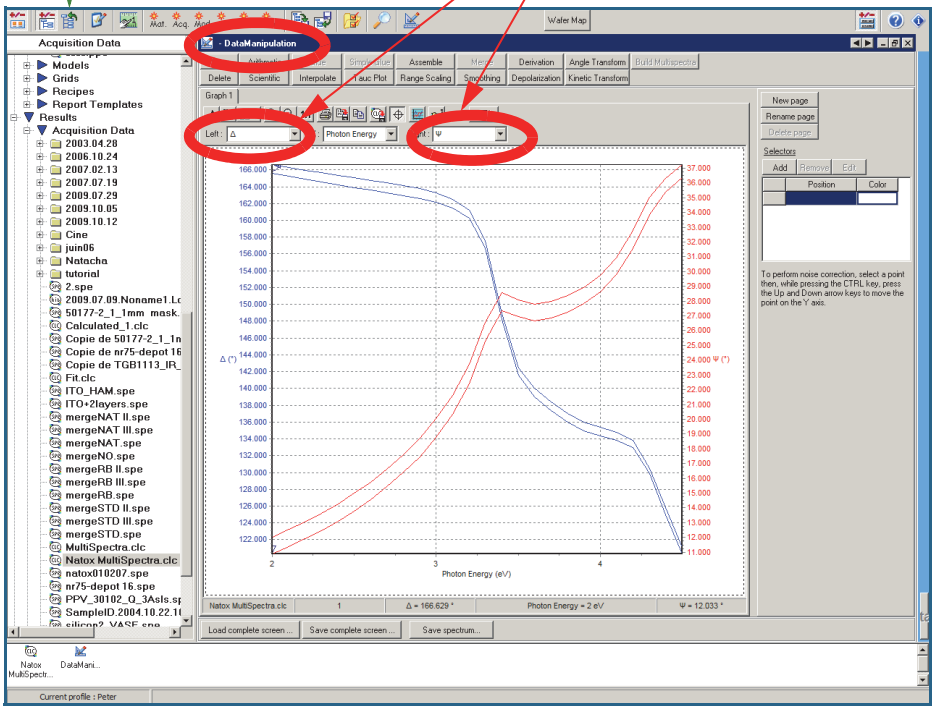
05/20/2016

Standard opened graphics screen

Data Manipulation icon: clicking on it will transfer the graphs to Data manipulation window




If necessary, change the displayed observables



## 10.2 Spectrum logic comprehension in DP2

In DeltaPsi2, it is important to know what are spectra, multispectra, files, calculated files, source, observables, graph page and how they are managed. The comprehension of this structure are required to use the manipulations features.

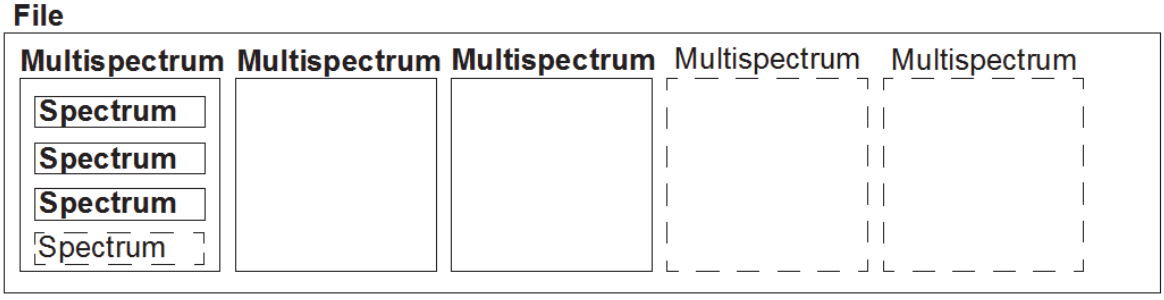
What is a spectrum ?

A spectrum is composed of acquired data during one acquisition period. These data are the datapoints for each observable which can be numerous and depend on the software setup (see "Observables Setup", page 17).

**Table 1: DP2 data containers**

| Container | information | where to visualize |
|---|---|---|
| **file** | Large container with adaptive structure from single spectrum to multiple Multispectra.<br>All of the datapoints of the selected observables are recorded.<br>Multispectra: Kinetic mode, Vase Φ and Θ, results which include acquisition scenarios or other combinations. | Double-click on the file from the TreeView: 1 or 2 observables will be displayed. The observables selector can be used to display 1 or 2 observables among those recorded inside the file. |
| **spectrum** | acquired datapoints during 1 acquisition period. | When a file is opened, all the spectra (if existing) are displayed (up to 2 observables). |
| **Multispectra or Source** | acquired data using the Kinetic mode, the Vase mode or by manual creation (Data Manipulation). | Double-click on the file from the TreeView: 1 or 2 observables will be displayed. All of the spectra are displayed excepting for Kinetic mode in which two cursor selectors go over the time and the wavelengths. |
| **Graphics screen** | can potentially displays all the spectra and up to 2 observables. | 1 or 2 observables can be displayed. The units selector can be used to display 1 or 2 other observables among those recorded inside the file. |

The figure below illustrates one typical data container used in DP2. This representation shows the most complex case.



# 10.3 Data Manipulation Operations

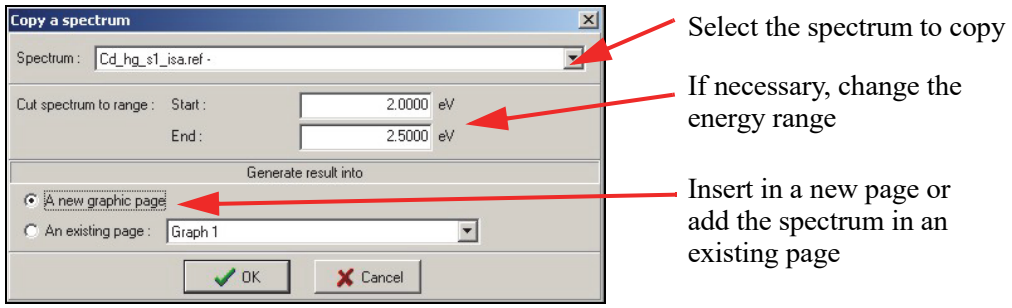
## 10.3.1 Edit Feature

From the Data manipulation main screen, it becomes possible to edit and correct a curve to remove uncoherent data or correct the noise. To edit a data, follow the procedure below:

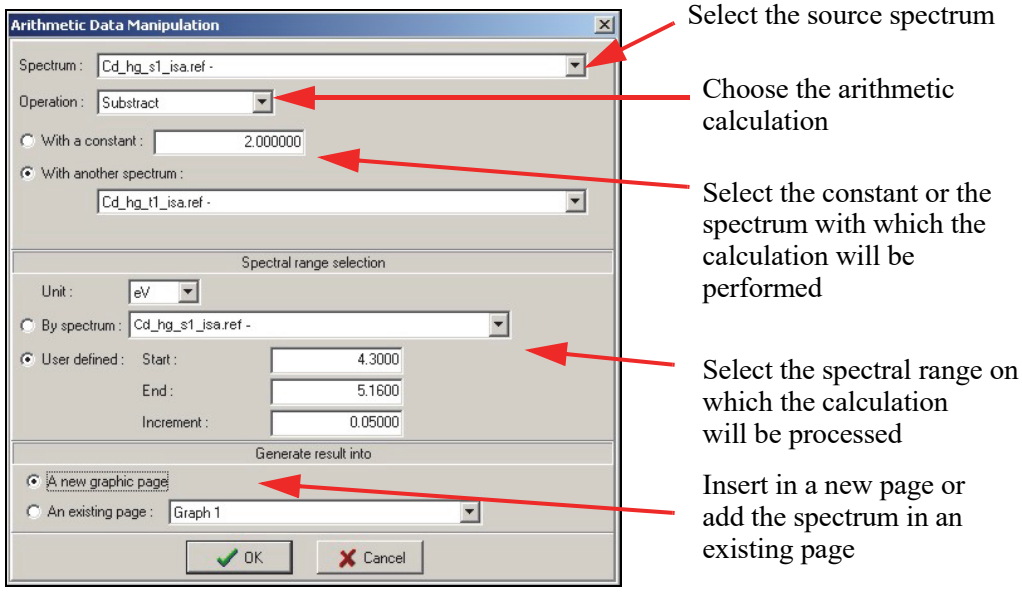
- Using the mouse, select the point to modify,
- Press simultaneously the Ctrl and Up or Down keys to move the point along the y axis.

1 **Copy** duplicate loaded spectrum, eventually limiting the range, to the same or a new page.

This feature is useful to prepare a spectrum for a specific manipulation, e.g. cutting a range and/or duplicate a file to keep original.



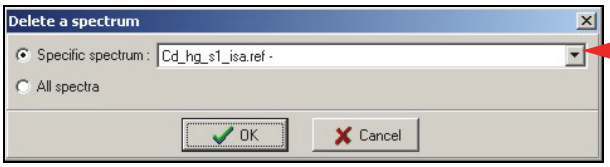

2 **Arithmetic** perform the 4 standard arithmetic calculations between a spectrum and a constant or using another spectrum.




05/20/2016

## 3 Delete

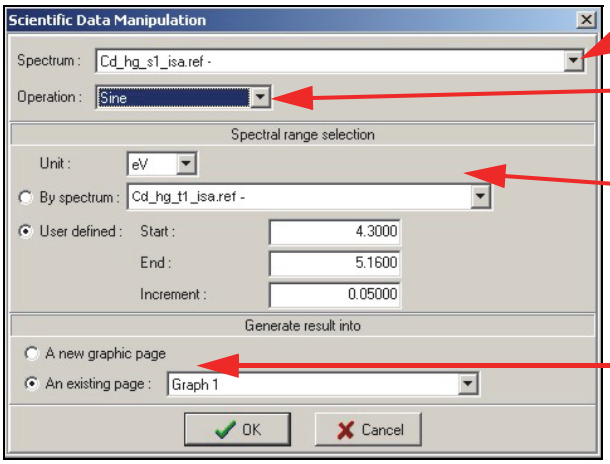
delete a spectrum.




Choose the spectra to delete

## 4 Scientific

perform the scientific calculations on a spectrum.




Select the source spectrum

Choose the Scientific calculation

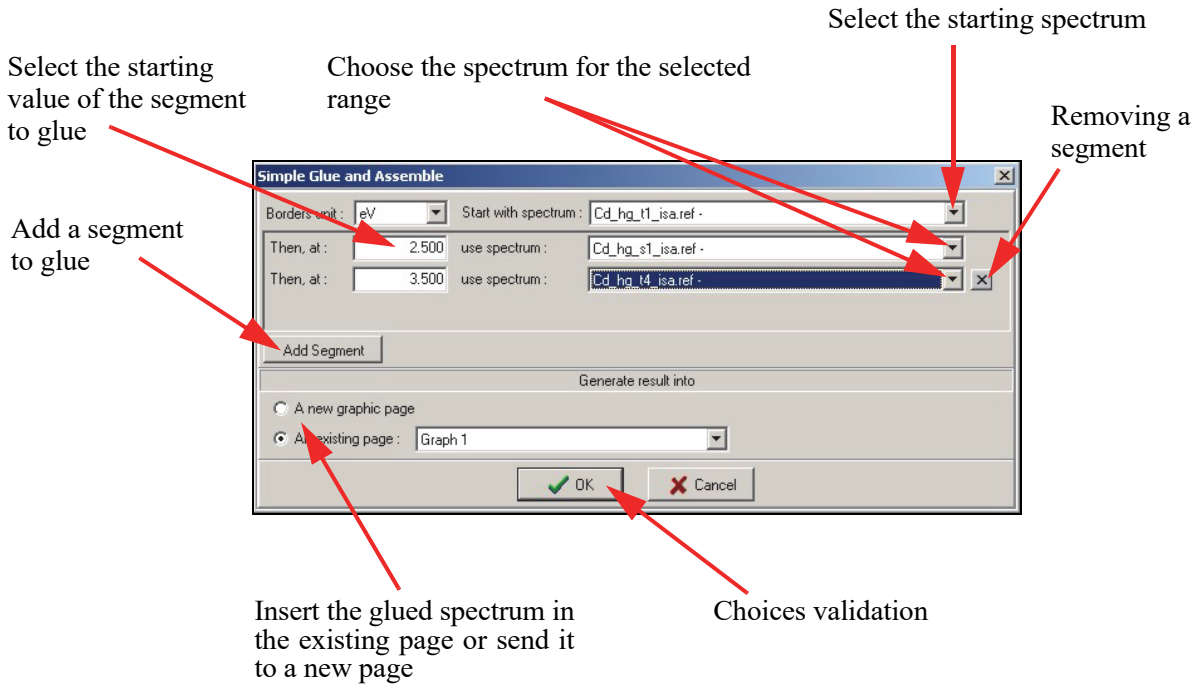
Select the spectral range or which the calculation will be processed

Insert in a new page or add the spectrum in an existing page

## 5 Simple Glue

glue several part of spectra according to the selected wave-length or energy range.

**Requirement**: at least 2 spectra must be present in the same graph window. If not, the "***Simple Glue***" button is inactive.



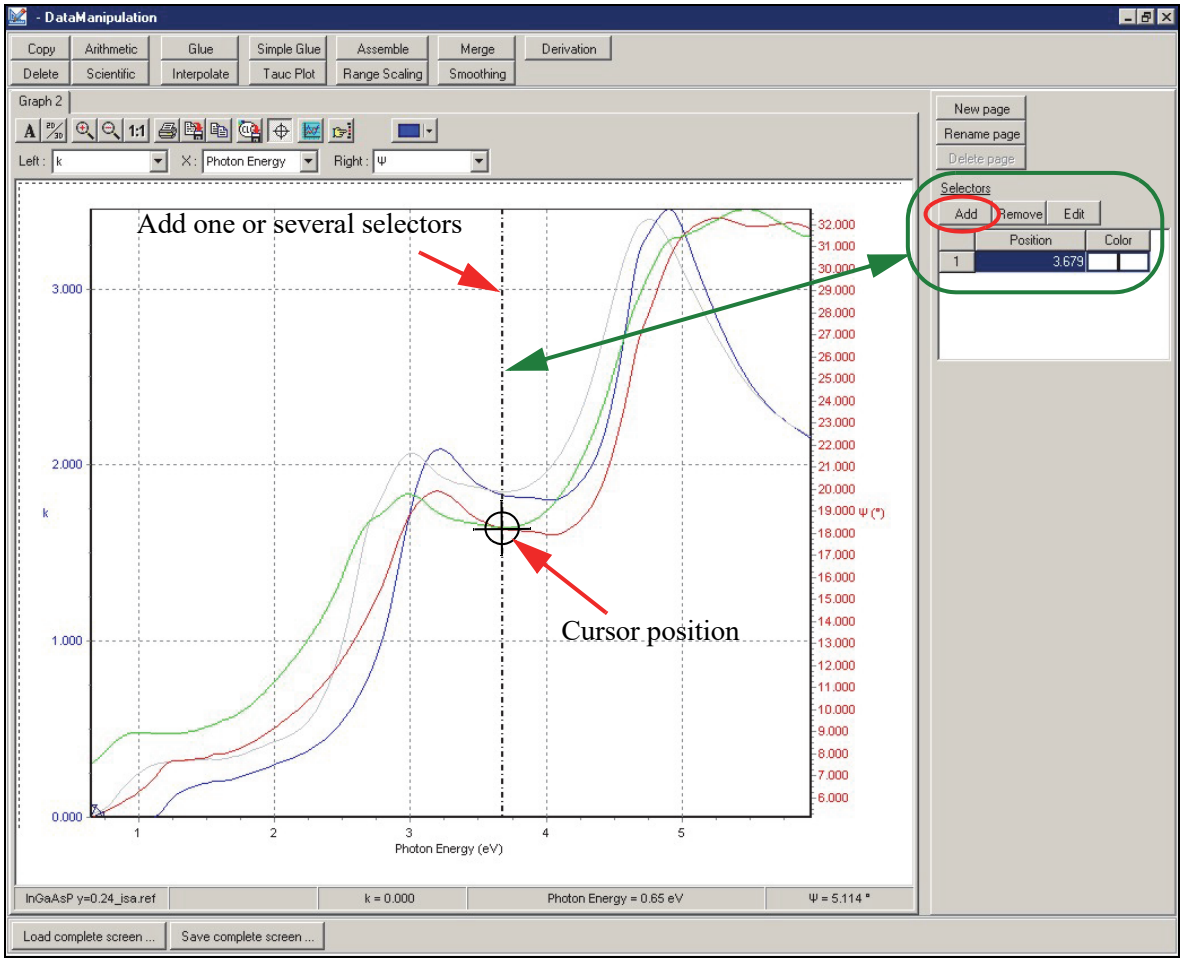

## 6 Glue

Same as Simple Glue with improved graphics selection.

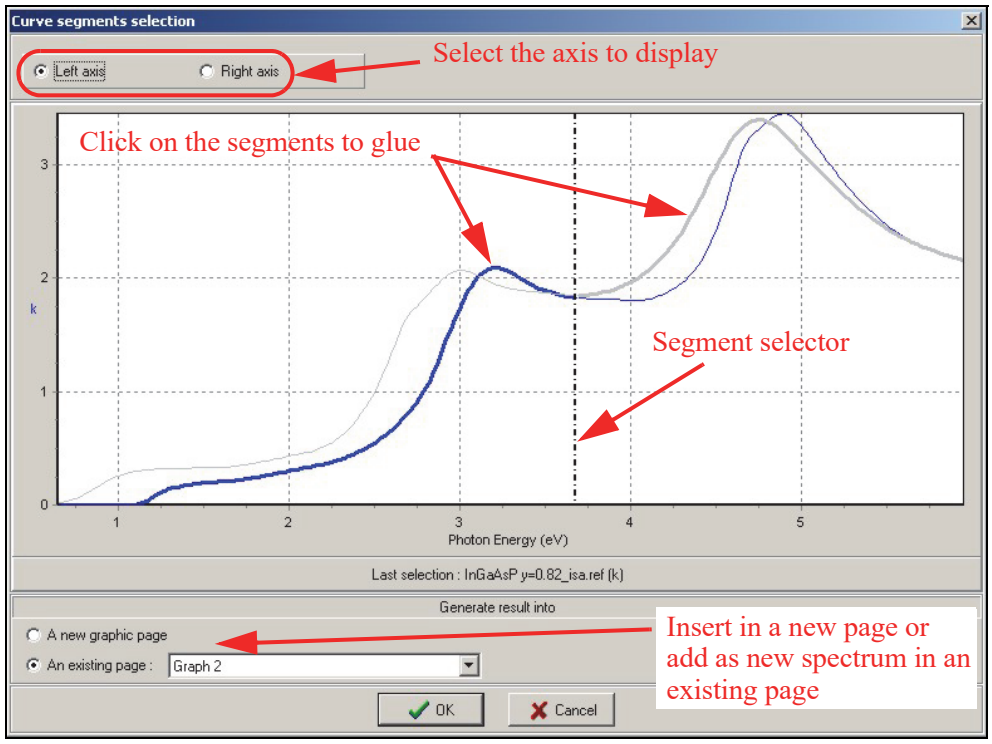
The Glue button becomes active only if at least one selector has been inserted on the graphics screen.

IMPORTANT: Once a selector is added, it is inserted at the cursor position. If the cursor is on location (x=0, y=0) or (x=max, y=max), the added cursor will not be visible. Using the mouse, move it then from this extreme position to the wished location.

NOTICE: The Glue feature operates on only one observable at a time. When several glued spectra are created for different observables, they can be assembled using the ***Assemble*** manipulation feature.

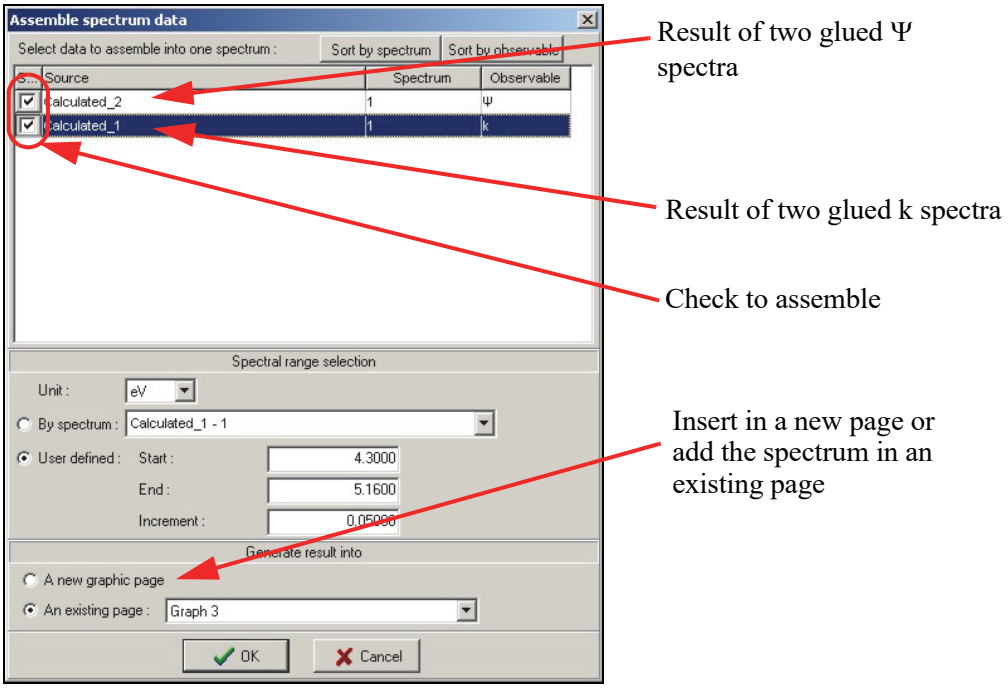



When the selectors have been inserted and setup, click on the ***Glue*** button to display the segments selection




# 7 Assemble

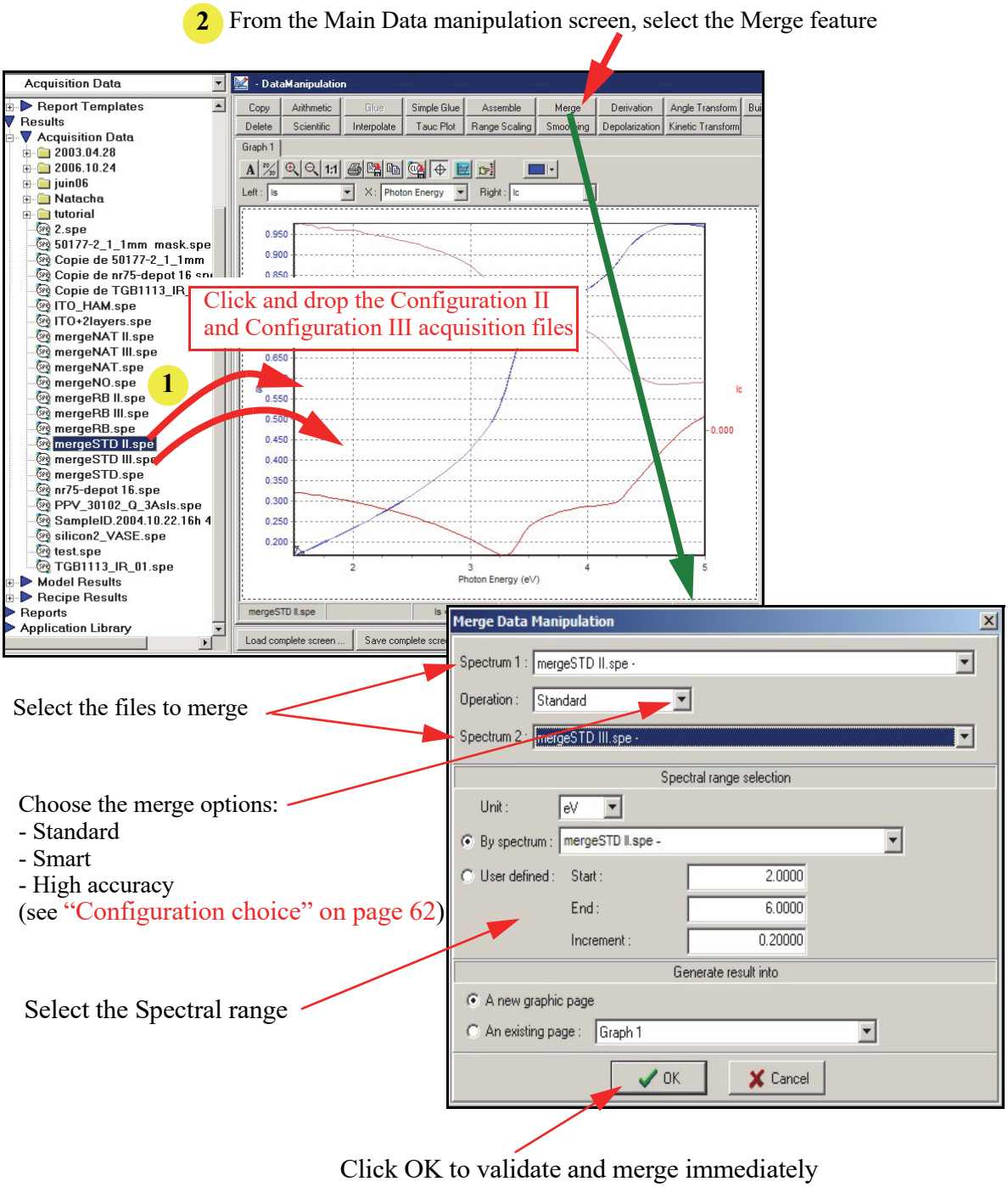
When a Glue has been performed with at least two observables (for example Psi and Delta), two calculated and separated spectra are created. Using the ***Assemble*** feature will generate one spectrum.




05/20/2016

# 8 merge

Perform a merge between a «**Configuration II**» and a «**Configuration III**» files using different Merge options. These options have been already detailed in the acquisition chapter which explain how to acquire spectra with the Merge option: see "Configuration choice" on page 61.



## 9 Derivation

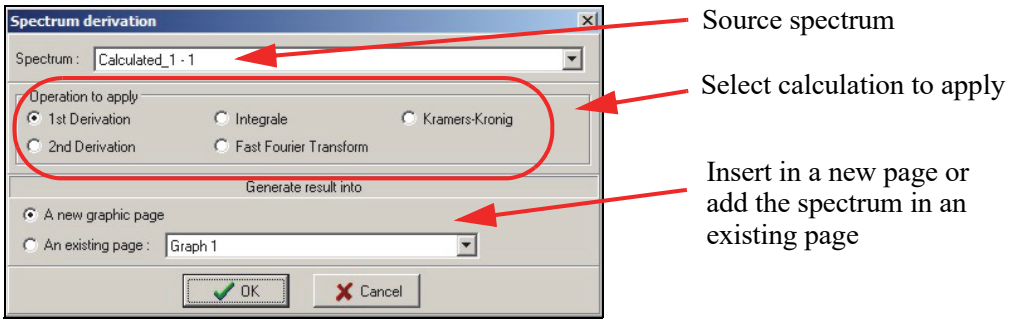
perform a «derivation family» calculations on a spectrum. Available calculations are: 1st derivative, 2nd derivative, Integral, Fast Fourier Transform and Kramers-Kronig.

These different processes can increase or reveal some properties about the layers. Many abstracts, which can be found on internet, explain the properties and the uses of such algorithms.

**Kramers-Kronig**

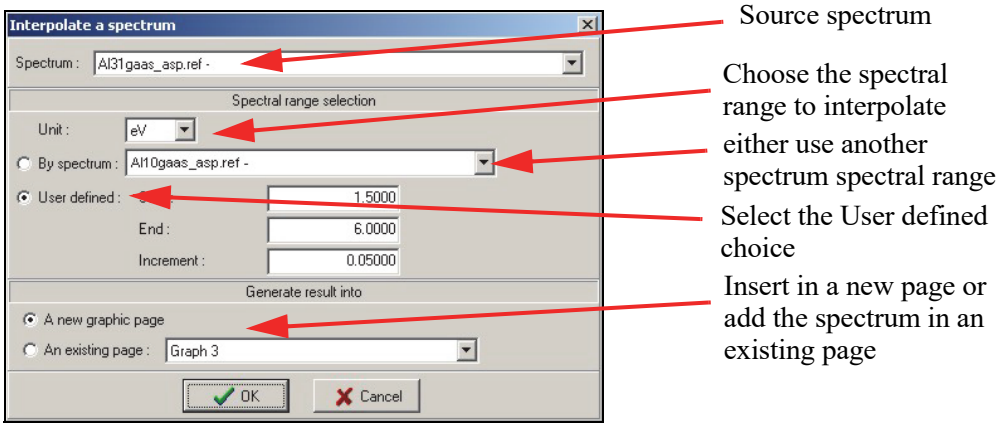
Kramers–Kronig (K–K) relations have constituted one of the principal tools in the optical spectroscopy for the assessment of the optical properties of media from measured spectra. The underlying principle for the existence of the K–K relations is causality. Thanks to the K–K relations we have achieved a better understanding of both macroscopic and microscopic properties of media.

Recently, various kinds of modified K–K relations have been presented in the literature. Such relations have been applied, e.g. to the nonlinear optical properties of polymers. A typical advantage of these generalized K–K relations is that the measured data do not need to be manipulated as in the case of the traditional K–K relations. Hence, the accuracy of the inverted data on linear or nonlinear optical properties of media becomes higher.




## 10 Interpolate

perform an interpolation calculation on a defined wavelengths or energy range.

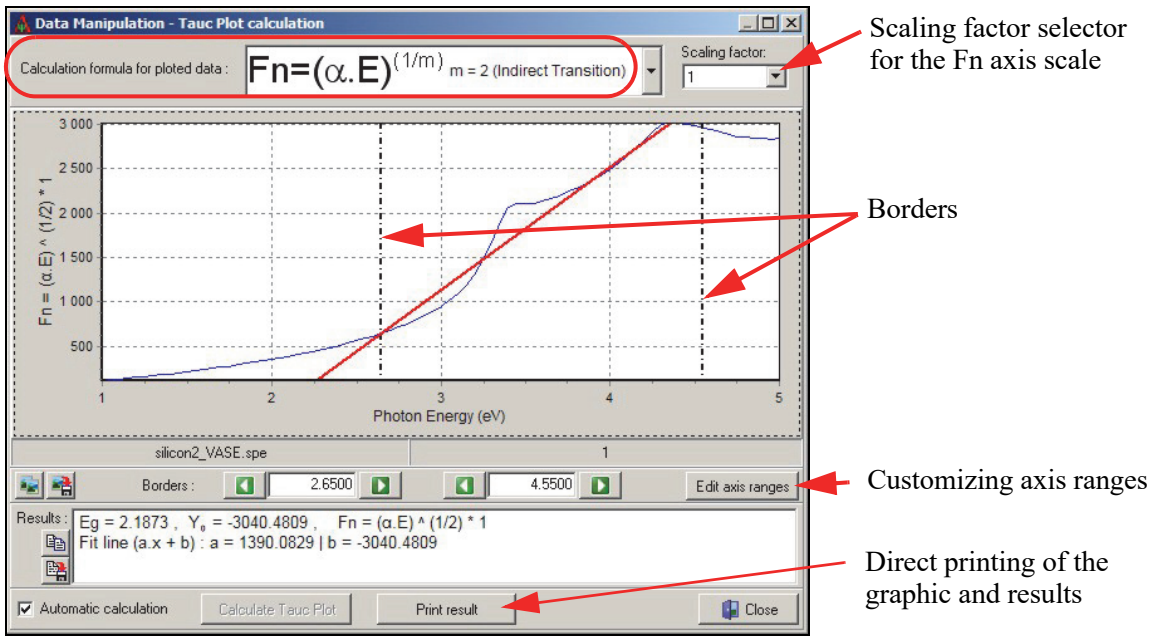



05/20/2016

The interpolation calculation is useful to change the increment of a spectrum opened in the current active graphic window.

## 11 Tauc-Plot

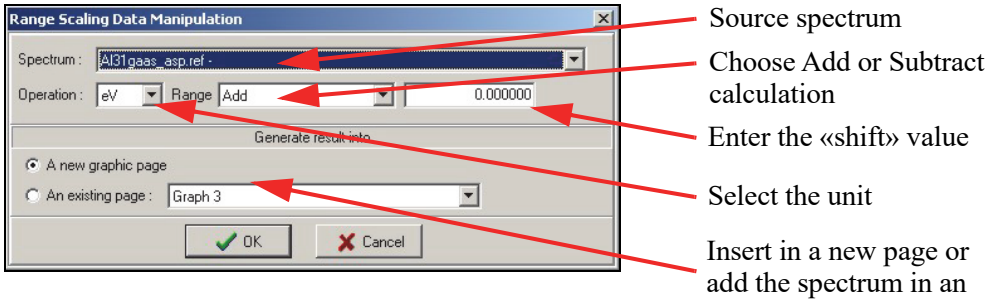
Tauc-Plot feature is a calculation program which determines the Energy gap of a semiconductor material.




**Energy gap definition:** In solid state physics, a band gap, also called an energy gap or bandgap, is an energy range in a solid where no electron states exist. In a graph of the electronic band structure of a solid, the band gap generally refers to the energy difference (in electron volts) between the top of the valence band and the bottom of the conduction band which is found in insulators and semiconductors. It is the amount of energy required to free an outer shell electron from its orbit about the nucleus to become a mobile charge carrier, able to move freely within the solid material.

## 12 Range Scaling

Move spectrum on the X-axis.

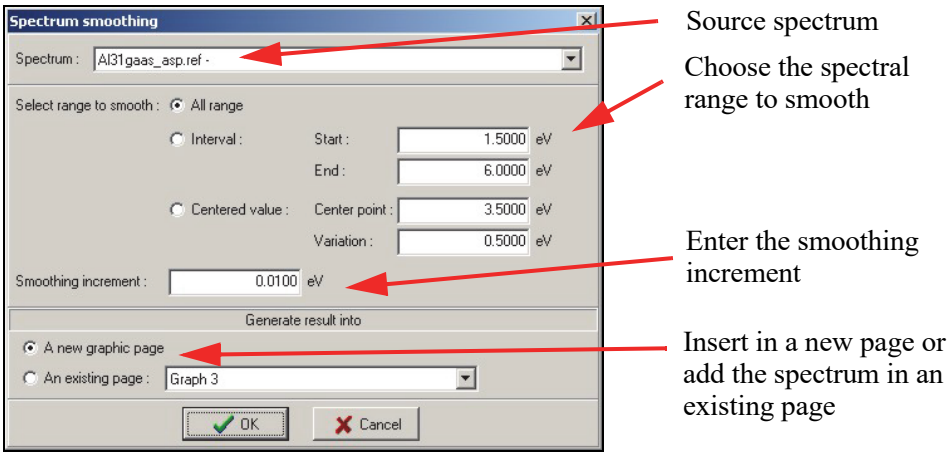



05/20/2016

This feature is useful to compensate for a wavelengths offset introduced by a decalibrated monochromator.

## 13 Smoothing

Perform a smoothing process on a spectrum.



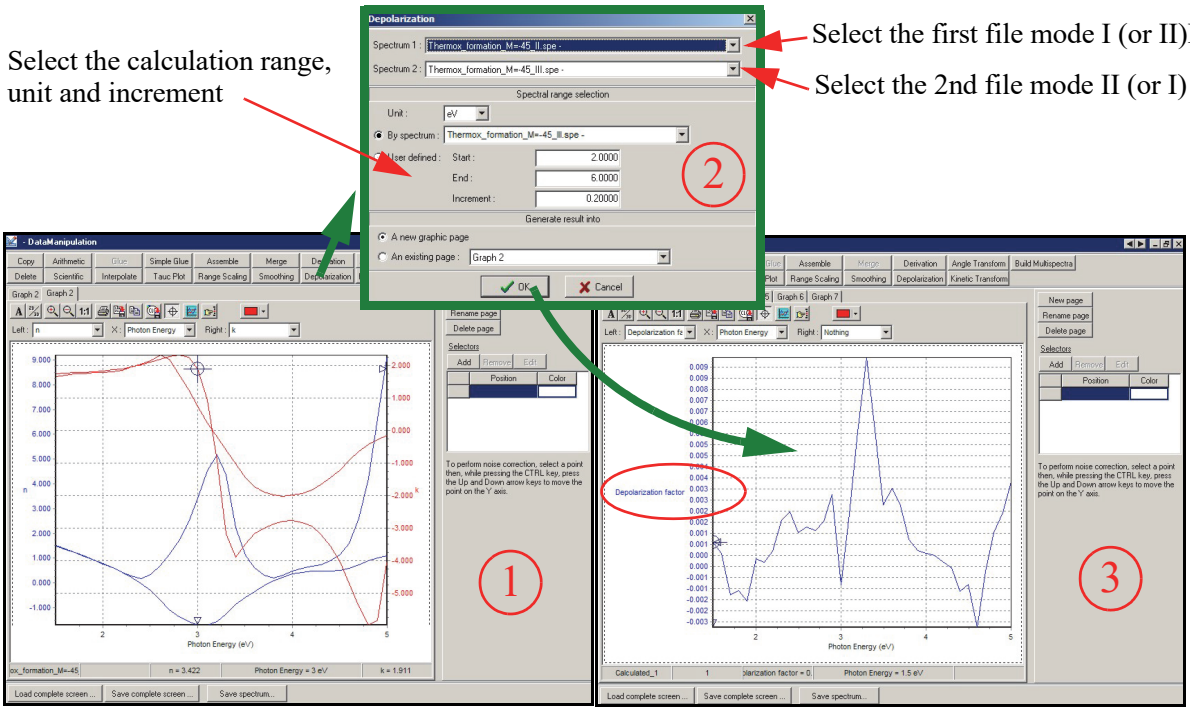

This feature performs a smoothing following the Savitzky-Golay method.

The Savitzky–Golay method essentially performs a local polynomial regression (of degree k) on a series of values (of at least k+1 points which are treated as being equally spaced in the series) to determine the smoothed value for each point. Methods are also provided for calculating the first up to the fifth derivatives.

The main advantage of this approach is that it tends to preserve features of the distribution such as relative maxima, minima and width, which are usually 'flattened' by other adjacent averaging techniques

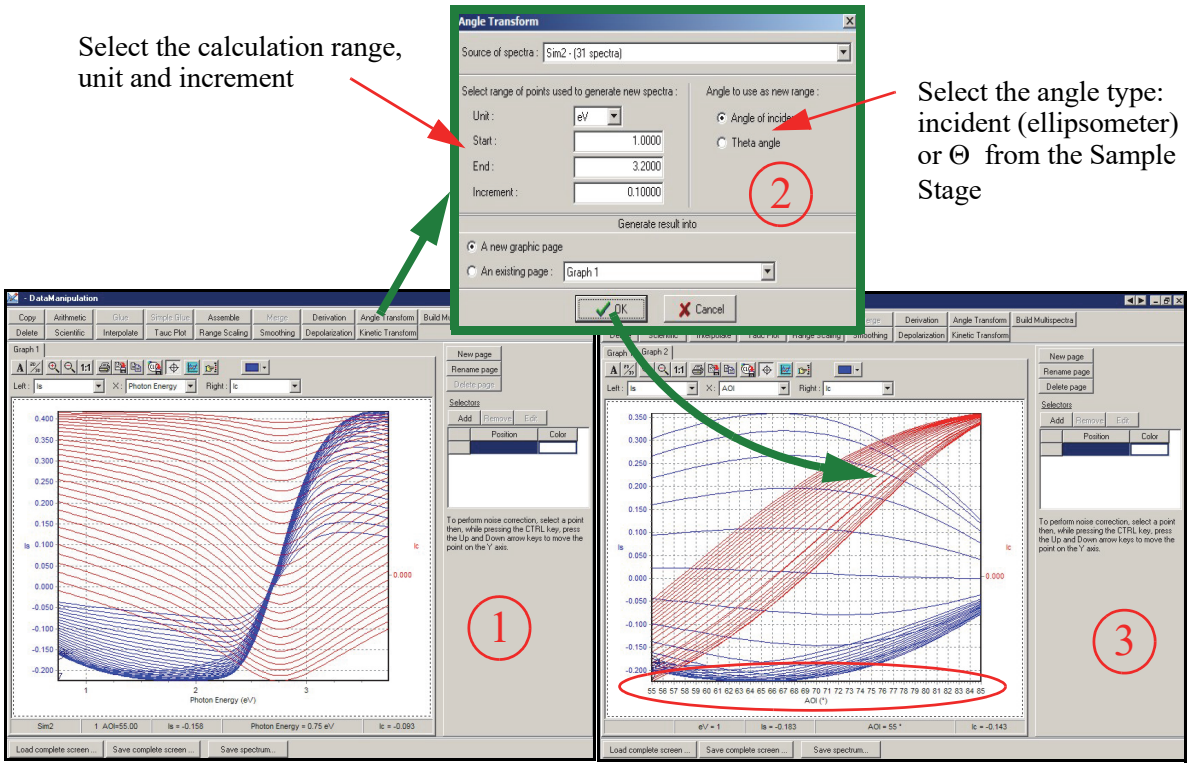
## 14 Depolarization

this feature calculates the depolarization factor from Two acquisition files acquired using Config II and Config III. The figures below show how to use this feature.



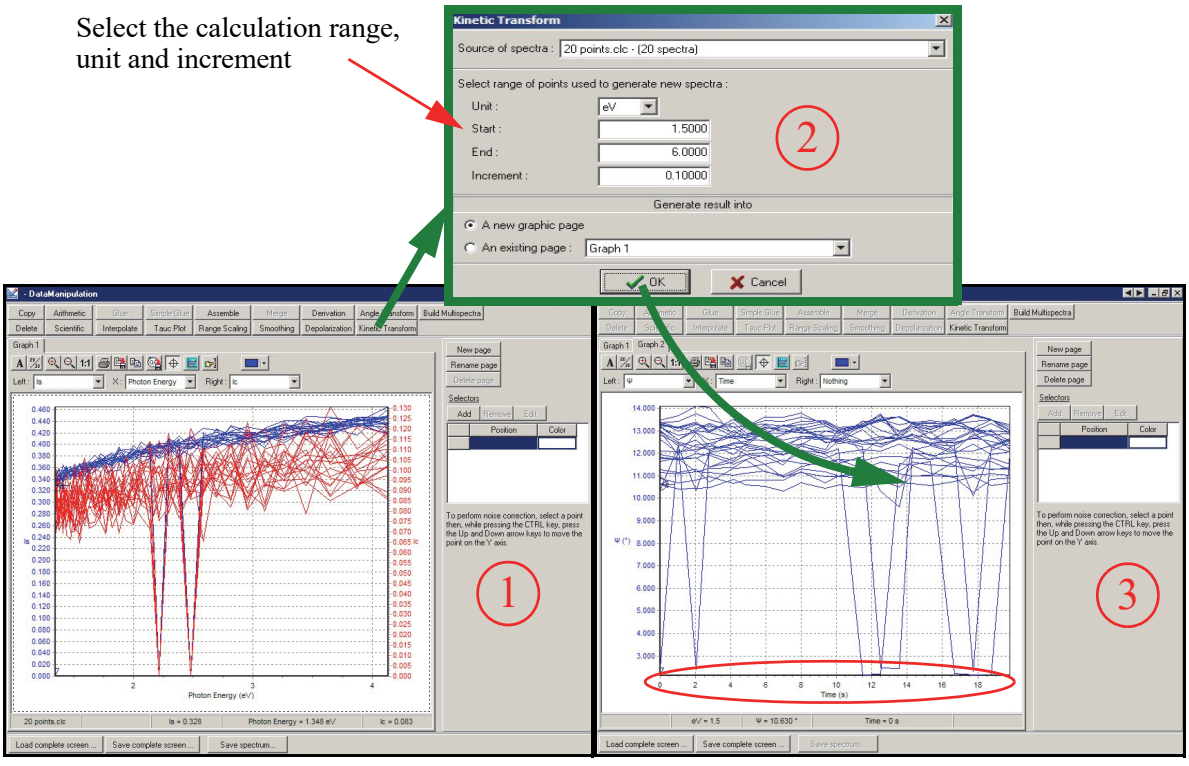

## 15 Angle Transform

the Angle Transform feature allows the user to change the axes representation if a multispectra file has been acquired using the Incident Angle variation or Θ Sample Stage Angle variation. The angle increment can be chosen and thus all the data points will be recalculated and then displayed. The following figure shows an example.



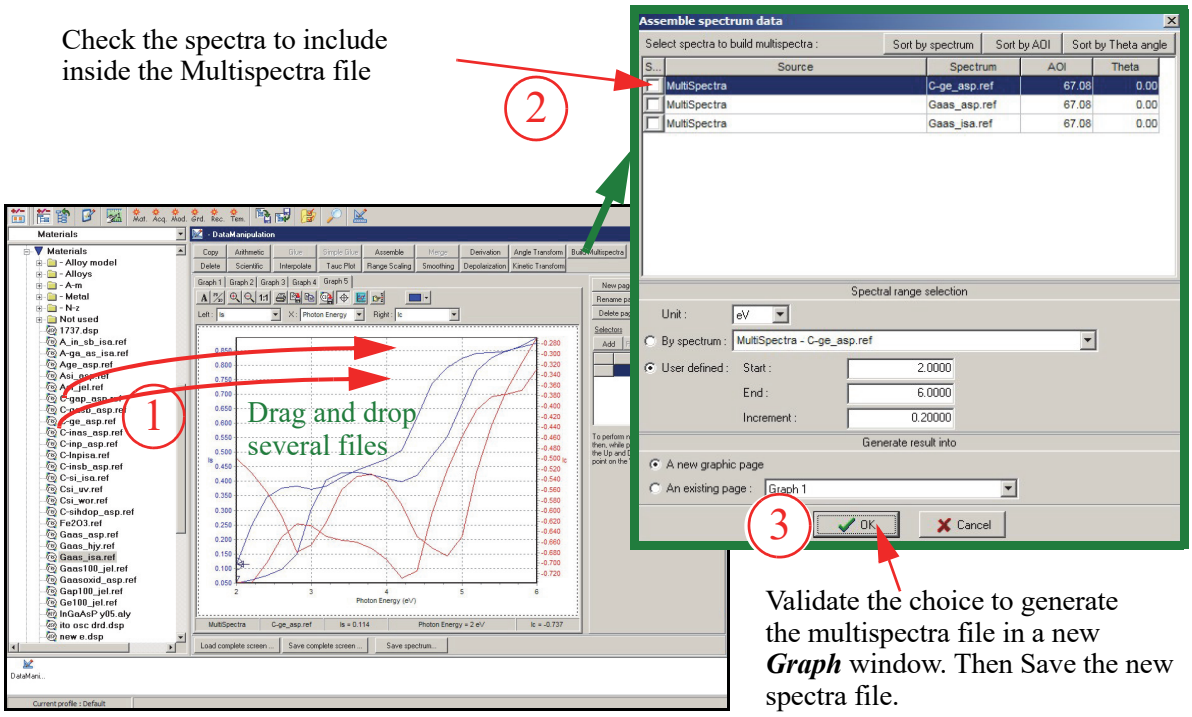

# 16 Kinetic Transform

the Kinetic Transform feature allows the user to display a multispectra file vs. time to a kinetic spectra representation. Once calculated, the Time will be shown on the X-axis and the selected observable on the Y-axis. To perform this calculation, the multispectra must be spectra vs. time. The figure below shows an example.



# 17 Build Multispectra

this useful feature allows the user to create a unique file from several spectra. It can also be used to select just one or several spectra from a long multispectra file.




05/20/2016

# 11 Acquisition and Data Manipulation Scenarios

The ***Acquisition*** and ***Data Manipulation Scenarios*** have been created to generate automated acquisitions with automated calculation processes. This calculation processes are not required but when many acquisitions with different routines are performed, the logical result is processed by calculations to reach a meaningful result.

The Figure 11-1, “Scenarios example diagram”, page 133 shows the logical organization of this feature.

As shown on the diagram, the Acquisition and Manipulation scenarios are two separate function. These scenarios setup is made separately.

How Acquisition and Manipulation scenarios are linked ?

It is very important to know that both scenarios are linked by a very simple relation: The **first acquisition** is automatically send to the **i1** input of the Manipulation scenario, the **second Acquisition** to the **i2** input and so on.

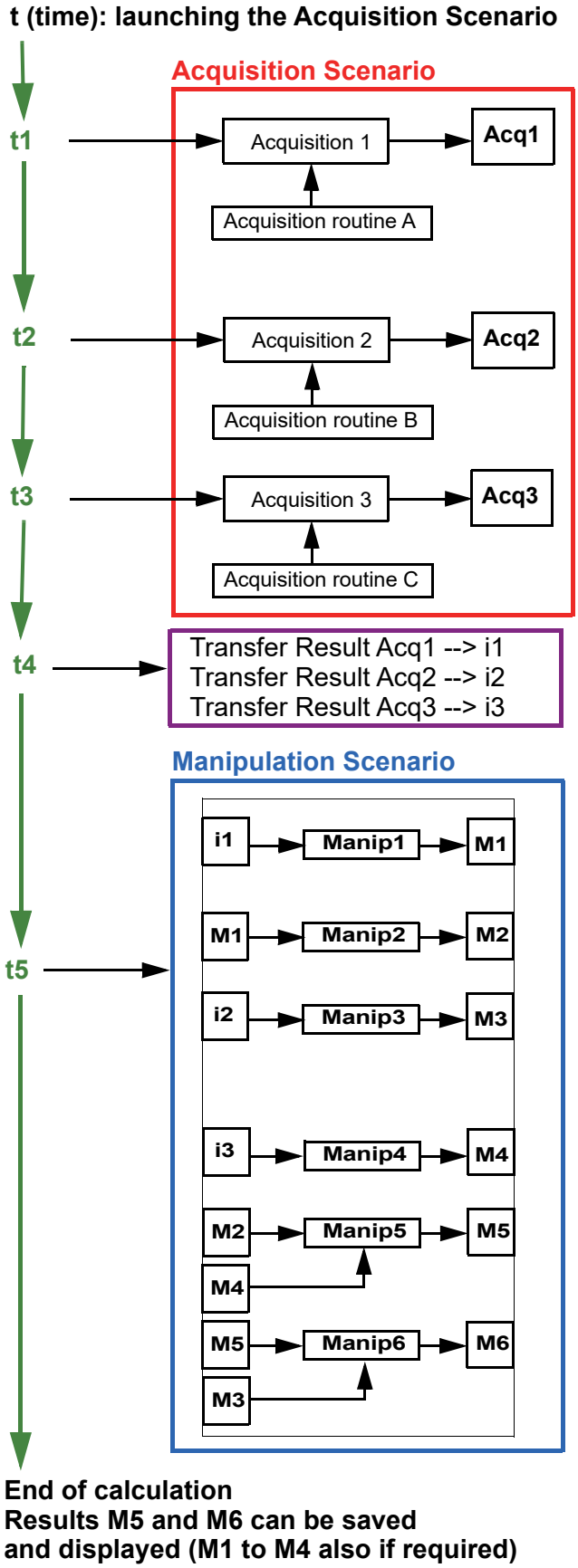



**Figure 11-1 Scenarios example diagram**

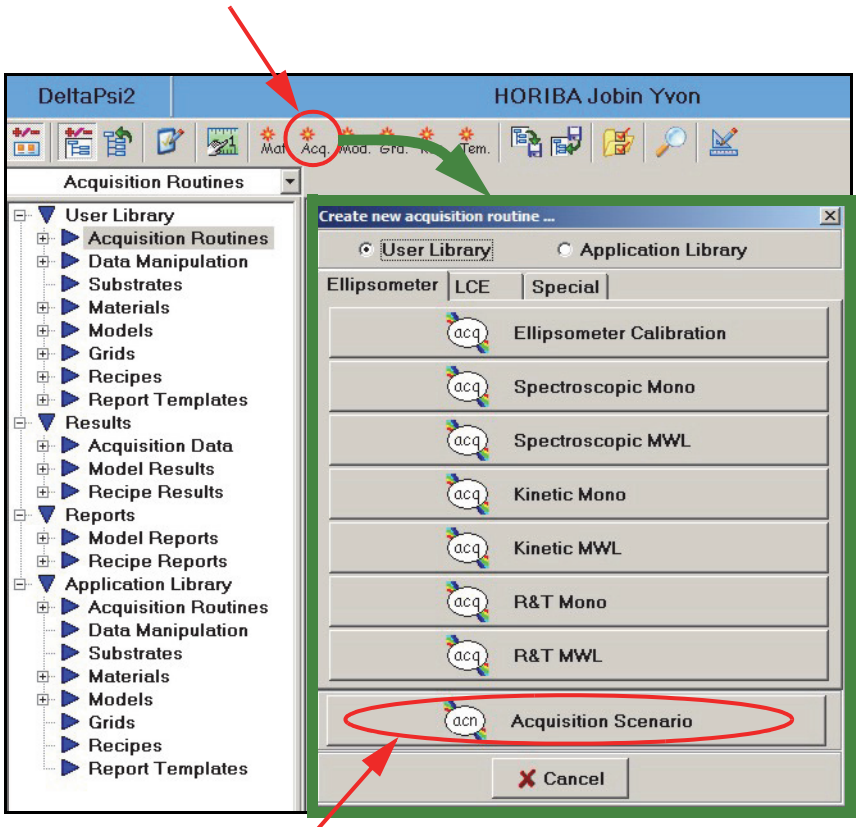
# 11.1 Acquisition Scenario

The Acquisition Scenario has been created to automate several acquisitions using one or several Acquisition routine already prepared. If you have already planified to use it with the Manipulation scenario, you must take into account that the acquired acquisition results will be redirected automatically to the Manipulation Scenario inputs; respectively the first acquisition results to the i1 input, the second acquisition to the i2 input and so on.

## 11.1.1 Accessing

There are two ways for accessing to ***Acquisition Scenario*** window:

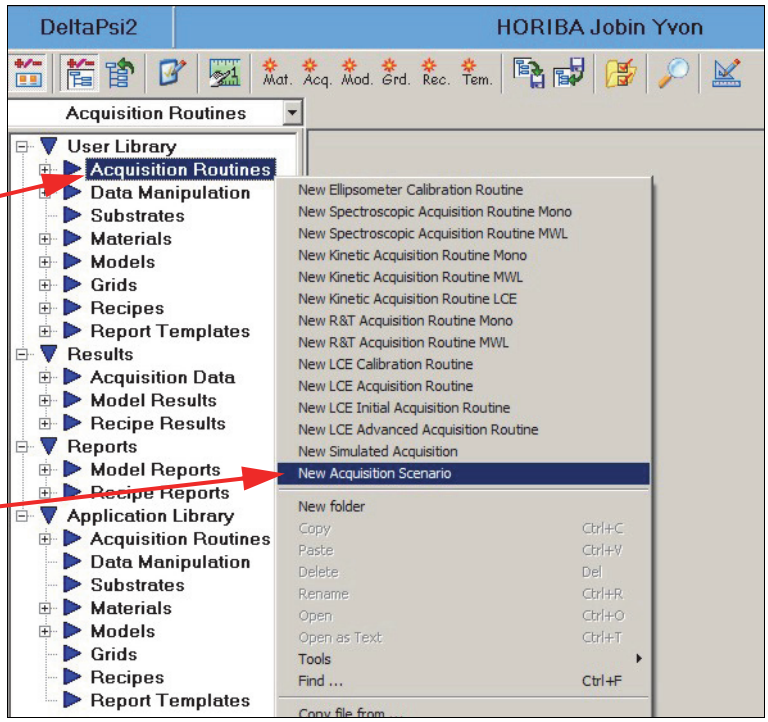
Click on the ***Acq*** icon located on the Main icons bar




Then select the **Acquisition Scenario** choice

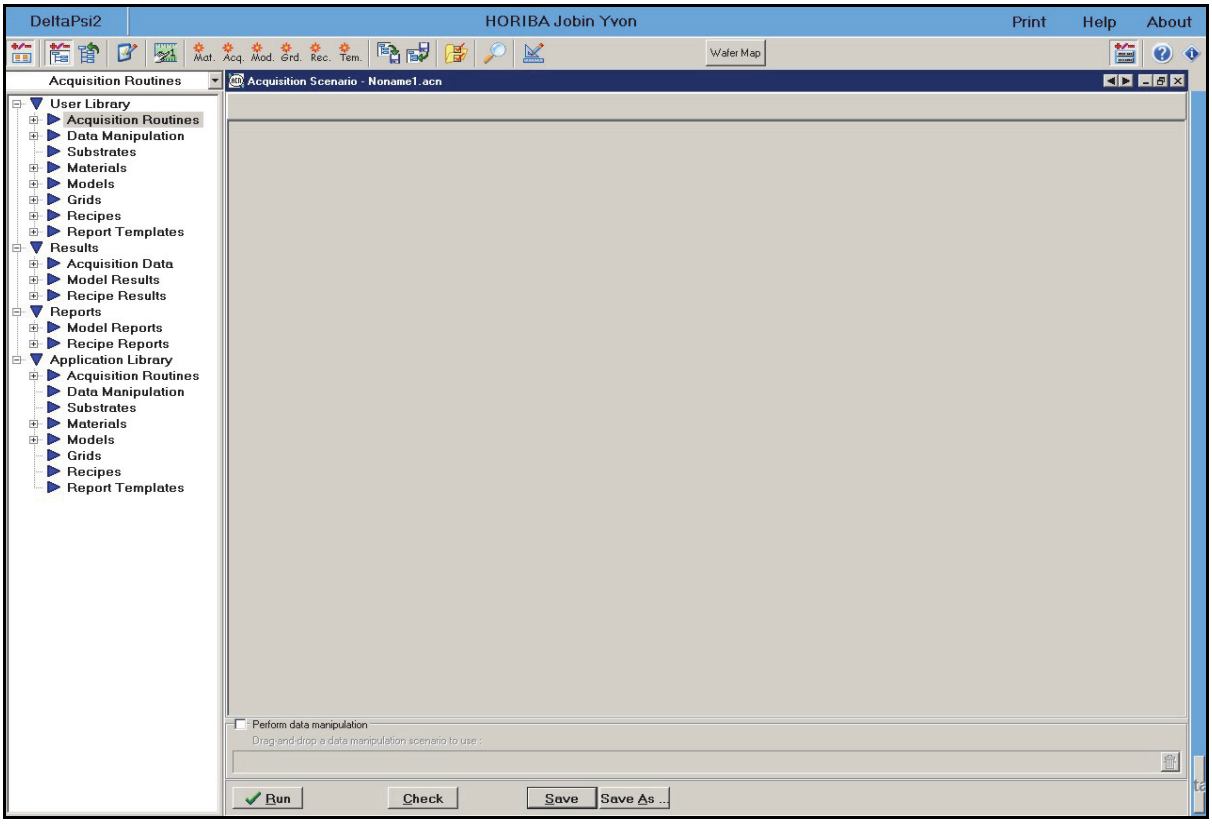
**Right-click** on the ***Acquisition Routines*** choice located in the DP2 TreeView

Select ***New Acquisition Scenario*** choice




### 11.1.2 Description

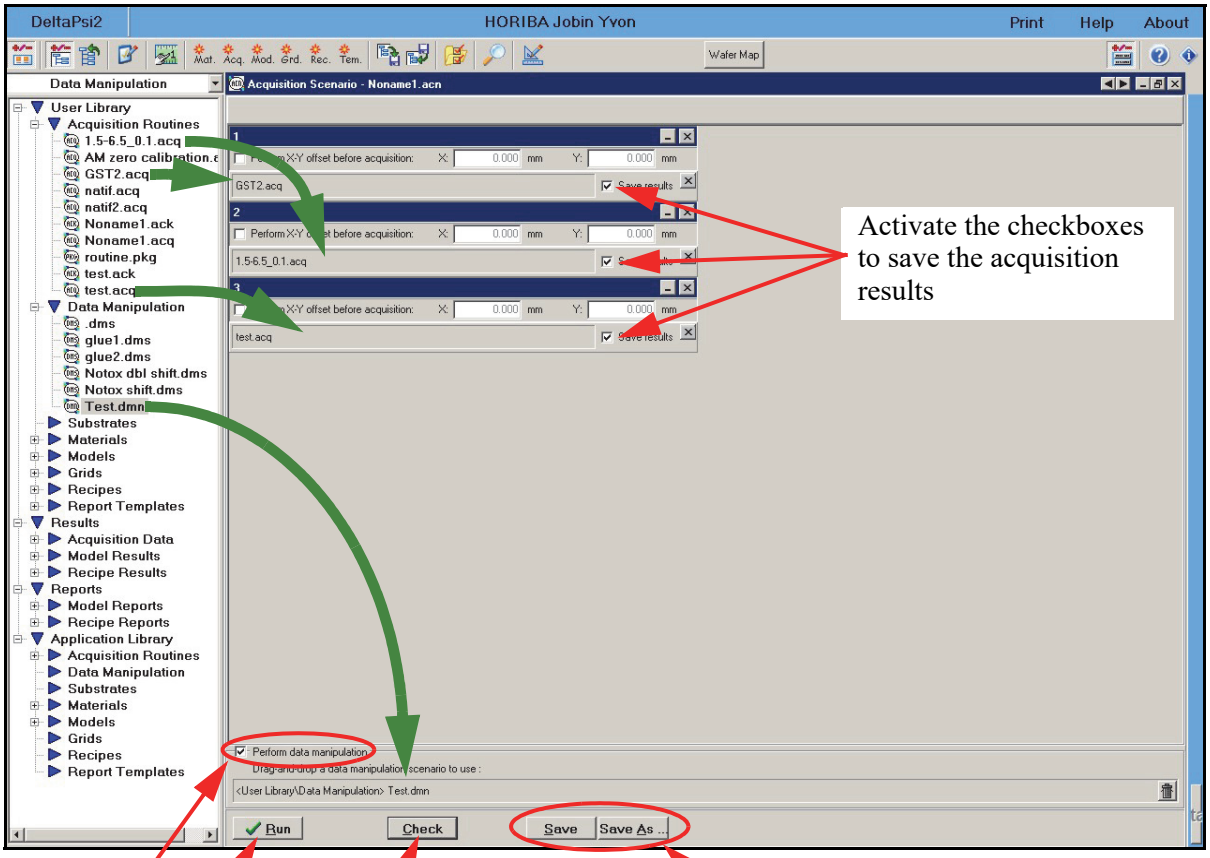
Once selected, the screen shown below will be displayed. The large empty area is ready to receive the acquisition routines.




05/20/2016

Just grab and drop the routines you want to execute to perform the acquisitions.

The screen below shows an example where 3 routines have been dragged and dropped. On the bottom, a ***Manipulation Scenario*** file has been added to link the ***Acquisition Scenario*** to the ***Manipulation Scenario***. In this example, the resulted acquisition 1 will be send to the Input 1 (***i1***) of the ***Manipulation Scenario*** and so on for the others acquisitions results.




## 11.2 Data Manipulation Scenario

The Data Manipulation Scenario is a powerful feature which allows to build a complete processing of acquisition files. We can consider that Data Manipulation Scenario is like a box with inputs and resulted outputs after the manipulations.

Logically, the Manipulation Scenario has been designed to be associated with the Acquisition Scenario. But for test purpose, it is also possible to drag and drop acquisition results files on the input and executes the Data Manipulation Scenario as a stand alone process.

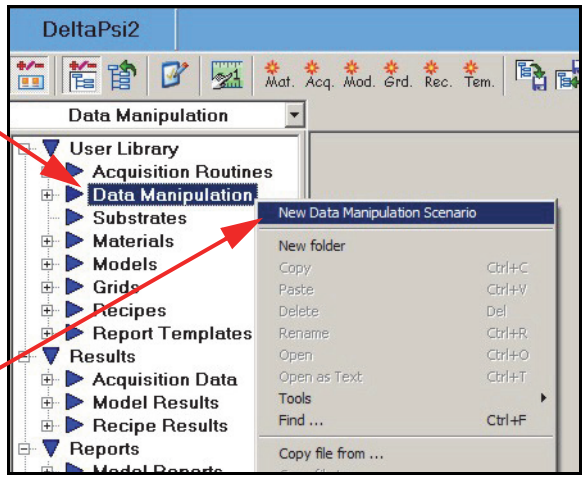
These files which have been dropped on the inputs are only functional when Manipulation Scenario is running as stand alone. That's why, even the inputs show associated files, they will be considered as empty, as long as it is used with an Acquisition Scenario.

### 11.2.1 Accessing

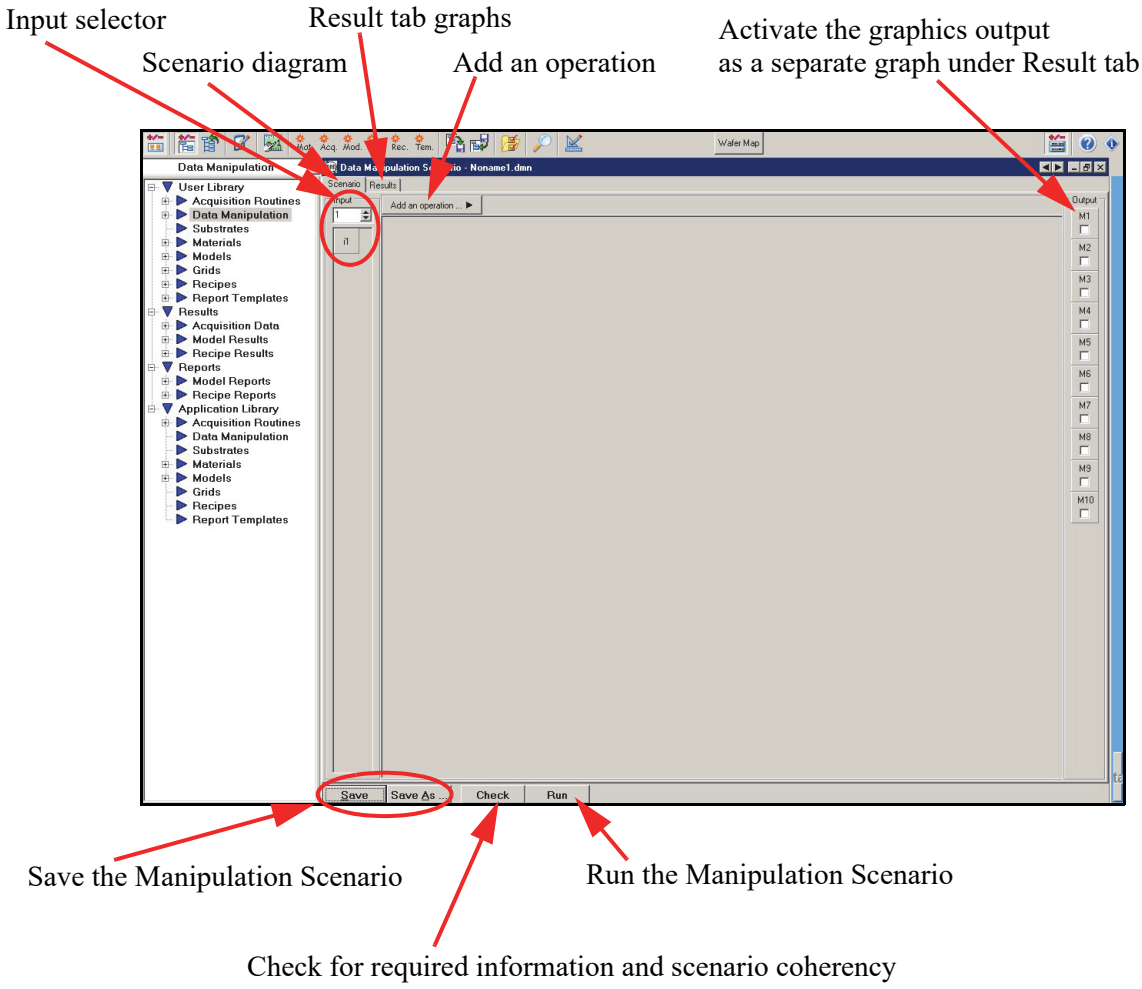
The figure below shows how to access to the ***Manipulation Scenario*** window:

**Right-click** on the ***Data Manipulation*** choice located in the DP2 TreeView

Select ***New Data Manipulation Scenario*** choice




Once selected, the following window will be displayed:




05/20/2016

### 11.2.2 Description

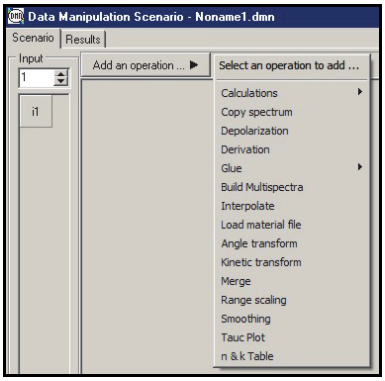
The logic of this powerful feature must be well understood before starting to work with.

The procedure below explains, step by step, how to build a functional diagram.

**Preliminary:**

Do not forget that, in the final step, the Manipulation Scenario will be linked to the Acquisition Scenario through the inputs i1, i2, i3 ...etc. That means that you must have an idea about the acquisition files which will be sent on these inputs and calculations which will be applied to the files..

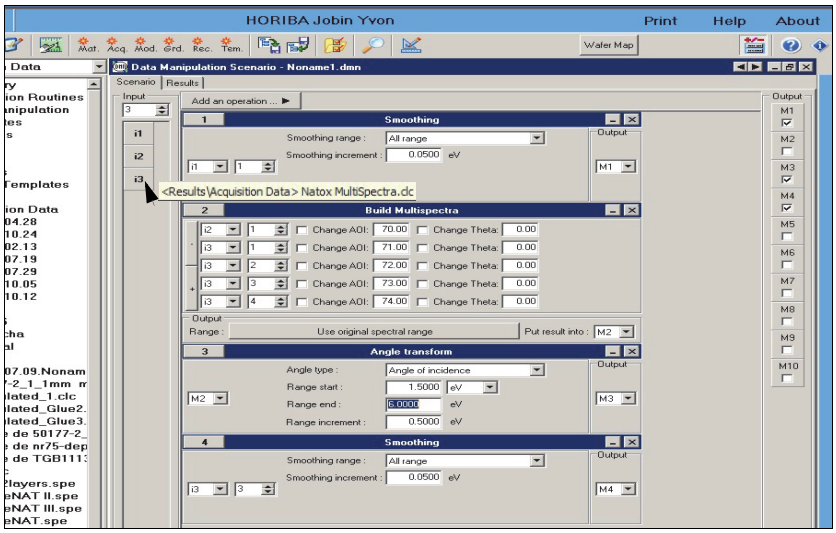
**Procedure**

| | |
|---|---|
| 1 **Input**<br>i1, i2 ...i*x* | Using the ***up/down*** arrows, select the number of input you will need for the entire scenario.<br>If you do not really know, don't worry, you will can add new inputs during the scenario build.<br>These inputs have two functions:<br>**During the scenario build**: you can drag and drop an acquisition results file from the TreeView to an input. This is useful to test the manipulation scenario as a stand alone process.<br>When a file has been dropped on an input, the **i*x*** label becomes bolded. Later, If you want to know the file which has been dropped, just move the cursor mouse above each input and the filename will be displayed.<br>**Linked to an Acquisition scenario**: whatever the files you have dragged and dropped inside the i1, i2 ... boxes during the build, these files will become completely transparent when the Manipulation Scenario is linked with the Acquisition Scenario. In fact, when the Acquisition Scenario is launched with a linked Manipulation Scenario, the first acquired spectrum will be sent to the i1 input, the second acquired spectrum to the i2 input and so on. |
| 2 **Add an operation** | Click on the ***Add an operation*** button to display the list of the available calculations or actions.<br>  |

All of the calculations or actions are those used and described in the Data manipulation feature (see "Data Manipulation", page 117), with the exception of the ***Load Material File*** action.

This action could be necessary to load and insert a file inside the calculation process. This file must be physically available in the TreeView.

The following figure shows a scenario example:




On the above scenario, we have 3 inputs, each one filled with an acquisition file (i1, i2, i3 are bolded). The cursor mouse, above the i3 input shows the filename **<Results\Acquisition Data> Natox MultiSpectra.clc**.

There are 4 operations in the scenario. As shown on the screen, each operation has an input selection (i*x* or M*x*) and an output selection M*x*. Additionally, on some operations, a spectrum can be chosen: this is the case for multispectrum or kinetic file. This spectrum is selected by a number which follows the chronological order. The other parameters are those available for each calculation (see "Data Manipulation", page 117).

Once the output has been selected, there are two choices: keeping the result for a chained only calculation or in addition output it as an independent file. In this case, the M*x* checkboxes located on the right must be checked.

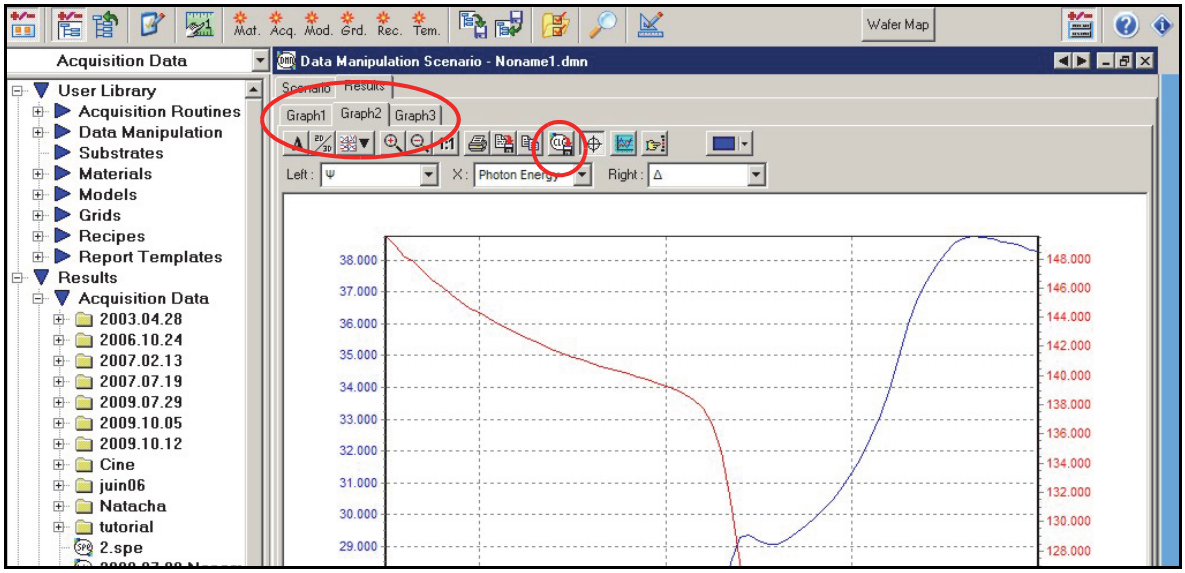
Looking the example above, the scenario will perform the following calculations:

- Smooth the first spectrum on the i1, using all the range with an increment of 0.0500 eV. The results will be sent into M1 which is checked for individual output.
- Build a multispectra file from the first spectrum on the i2 and the four first spectra on the i3. The angle of incidence has been modified. The results will be sent on M2. The individual output has not been selected.

- Transform the multispectra M2 file previously calculated with variable angle of incidence, and display it with the angle of variation unit on the x axis. The results will be sent to M3. The individual output of M3 is selected.
- Smooth the third spectrum of the i3 input on all the range with a 0.0500 eV increment. The result will be sent to M4 and individual output has also been checked.

3 **Mx**

As explained on the example above, each result can be checked to produce an individual result. This result will be displayed as a "graph" in the ***Results*** tab (see page 137).




On this example, three M*x* outputs have been checked. After the calculation, three individual graphs are generated. The resulted graphics screen is standard in DP2.

To save the current graph, click on the clc icon

# 12 Getting Started Session with UVISEL VIP System

The **UVISEL VIP** System includes a High Performance Ellipsometer called UVISEL and a Reflectometer called VIP (**V**isible **I**nspection **P**latform).

The **P**attern **R**ecognition **Station** is permanently activated and can be used for Ellipsometry and/or Reflectometry measurements. The following chapters explain, step by step, a practical example of reflectometry measurements on a patterns sample. This procedure is identical if applied to ellipsometry measurements.

## 12.1 Power on the System

To power ON the UVISEL VIP System, turn the large red knob located on the front of the System to the right, as shown on the figure 12-1, page 141.

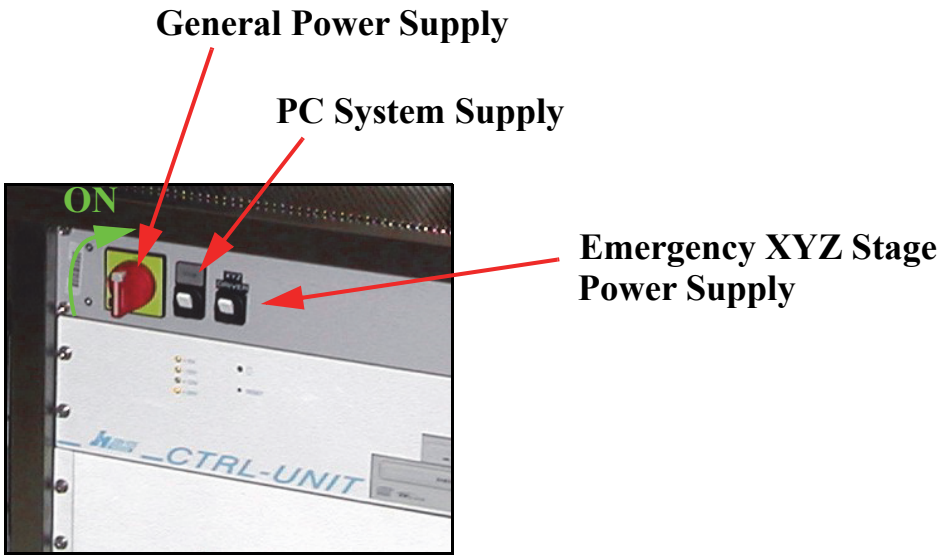



**Figure 12-1 Power supply**

## 12.2 Starting a session

- **System is powered ON.**
- **Double click on the Measurement icon**  **located on the working panel: this action will load DeltaPsi2 and PR Station softwares.**
- **By default, DeltaPsi2 is displayed on the screen. From this step, an Ellipsometry and/or reflectometry measurements can be started.**
- **During the session DeltaPsi or PR Station can be switched using the button located on the right of the screen (see figure below).**

Before running measurements recipes, pre-adjustment phases are essential for a success achievements of the measurement session. There are three pre-adjustment phases:

- **General: sample size, autofocus setup,**
- **Layout creation: dies grid creation,**
- **Measurement points setup.**

## 12.3 Pre-adjustments: general

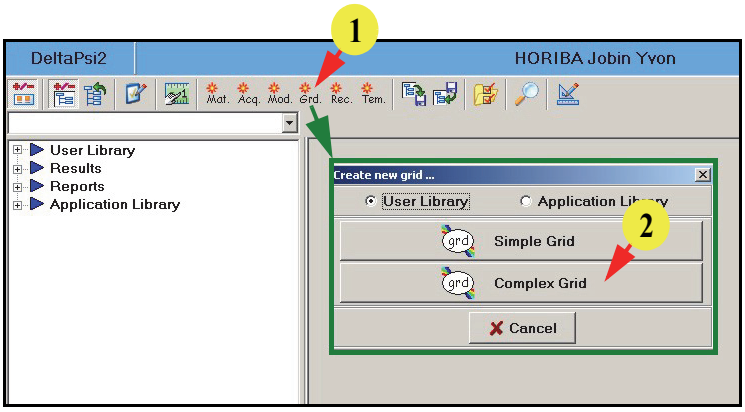
This chapter will describe how to setup a grid and how to adjust the autofocus.

### 1 Starting a session

To start a session, see above "Starting a session" on page 141.

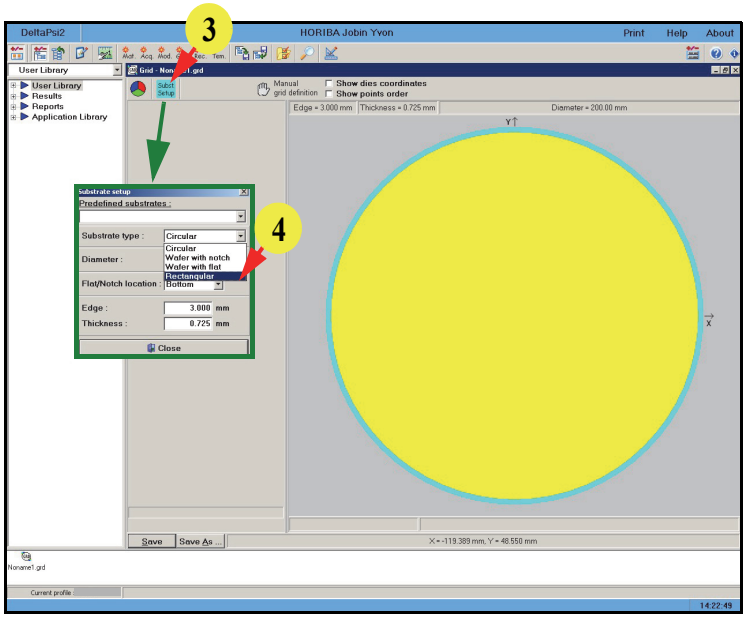
### 2 Choosing a Grid

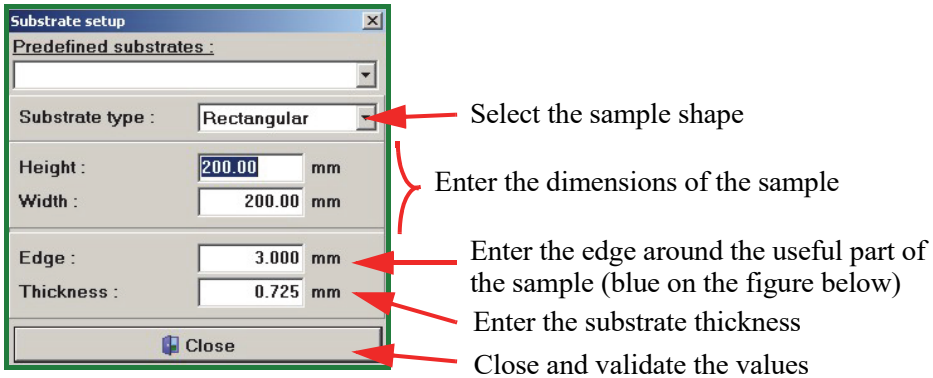
Click on the Grid icon, then choose the Complex Grid option as shown on the figure below. The Grid Editor will be then opened.



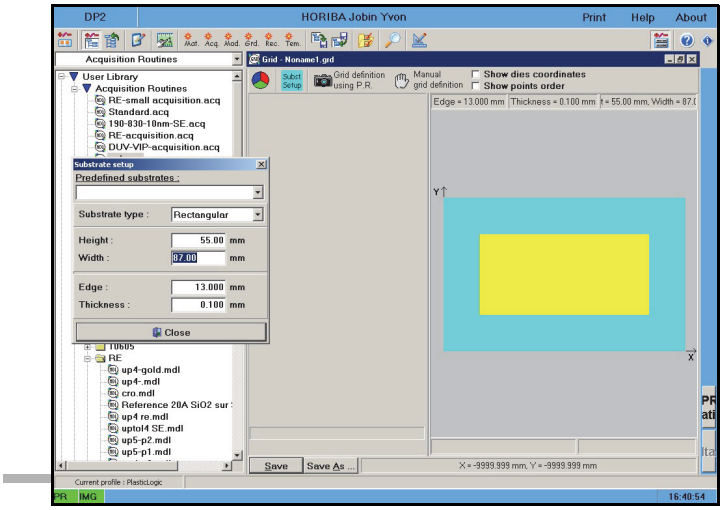

### 3 Setting up the Grid

Click on the Grid Setup icon: a pop-up screen will be opened. Enter the parameters of the Grid. In the case of repetitive patterns, the grid is the entire patterns sample.



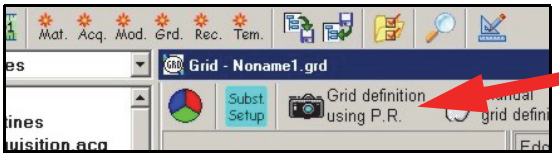

In our example, we have chosen the following parameters:




05/20/2016

# 4 Real Time View

Click on the ***Grid definition using P.R.*** button. This action will transfer the Grid parameters to the Pattern Recognition software.




Transferring Grid definition to PR software

The Real Time image of the patterns sample is then displayed. The following sequence will setup and calibrates the patterns layout and will set the measurement points.

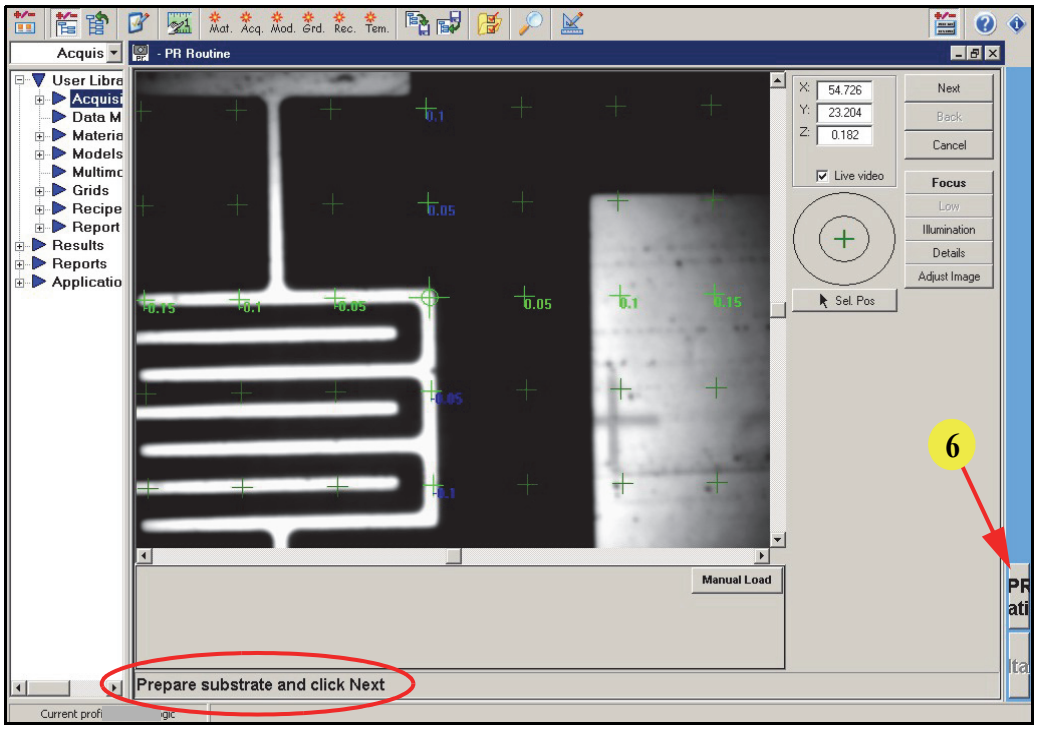



**Figure 12-2 Real Time image**

# 5 Autofocus Adjustment

First and foremost the autofocus setup must be performed. Without this action, the sample could be out of focus.

Click on the PR Station button: see step 6 on Figure 12-2, “Real Time image”, page 144. The Pattern Recognition Station Main screen is then displayed (see figure below).

NOTICE: This procedure can be also performed from DeltaPsi2, but the PR Station screen is more convenient.

This screen is useful to perform the following action:

- **Setup the Z stage height,**
- **Setup manually the pointer either by mouse or keyboard,**

- **Quickly move the stage from one point to another, useful to lay down or remove a sample.**

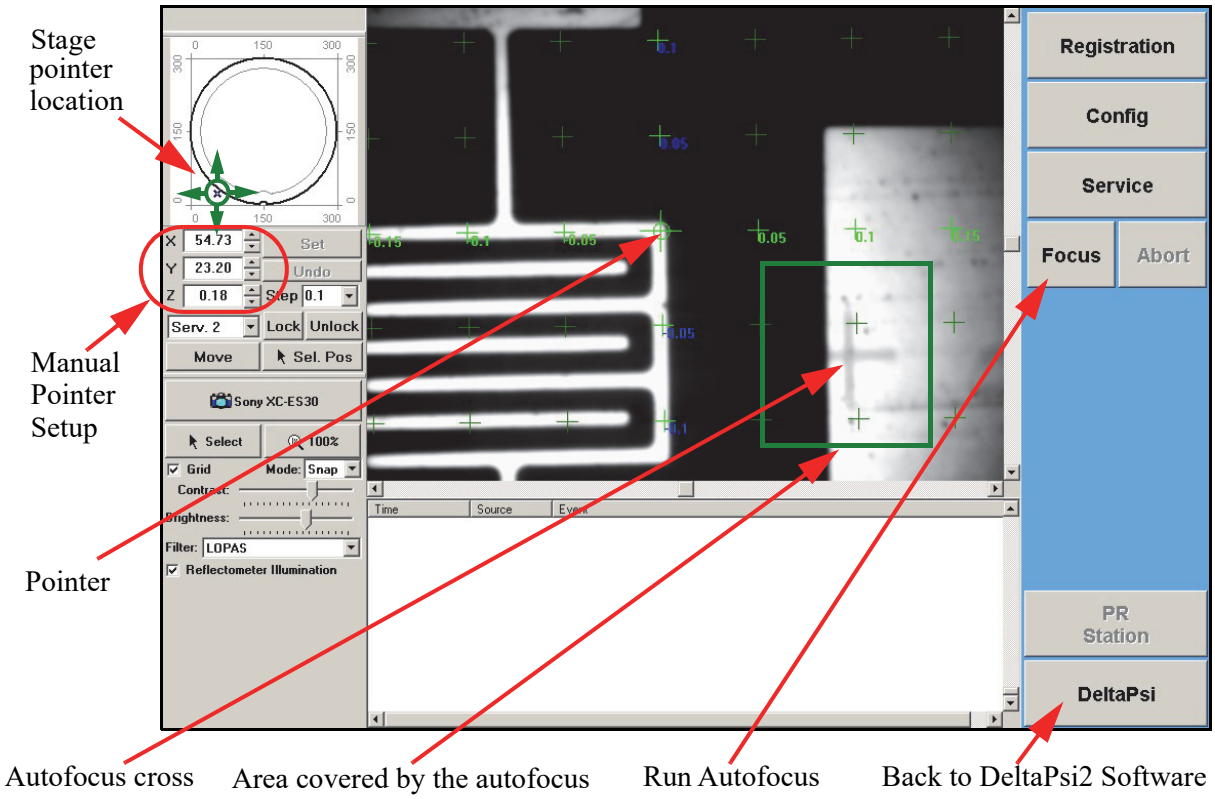



**Figure 12-3 Pattern Recognition Station Main screen**

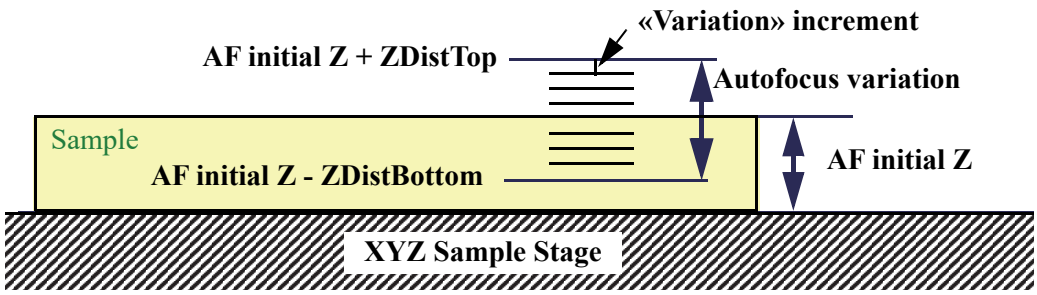
## Z height Setup

Depending on the sample, the initial Z height and the height variation for autofocus must be set. Click on the ***CONFIG*** button then select the ***Auto-Focusing*** tab to open the parameters table. The figure below shows the parameters which are important to verify and change for the best and fast autofocusing operation.

***AF initial Z (mm)***: enter the sample thickness value.

***ZDistTop*** and ***ZDistBottom***: enter the variation value which will be applied around the ***AF initial Z*** value.

***Variation***: enter the autofocus variation increment.




The ***ZDistop*** and ***ZDistBottom*** values must slightly exceed the sample uniformity variation and ***Variation*** increment must be chosen as a compromise between the focus precision (enough for pattern recognition) and focusing speed.

05/20/2016

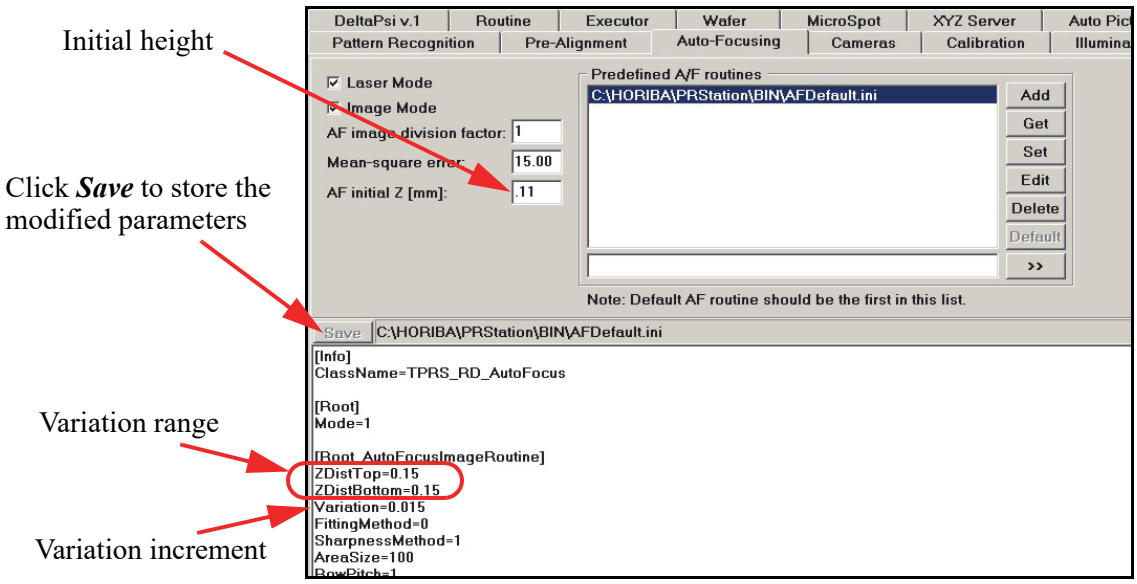



**Figure 12-4 Autofocus parameters location**

### Setup the pointer

Figure 12-3 shows the pointer location over the sample. To move this pointer, click and drag the small cross; once the mouse button is released, the stage moves on the selected position. This feature is useful when a specific location is searched; it can be also used to lay down or remove a sample. To move the stage with a higher precision, enter the target value directly in the X, Y, Z fields.

Click on the ***Save*** button to store the modified parameters.

### How to verify the Autofocus

From the Pattern Recognition Station Main Screen (see figure 12-3), move the pointer with the mouse over the sample and position the dark big Autofocus cross over a clear surface (see figure 12-3). Once done, click on the ***Focus*** button: this will activate the Autofocus feature. Once performed, try several very different locations of the sample: this must be made to insure the sample thickness is within the range of the defined height (see "Z height Setup" on page 145). If necessary, redefine these parameters.

## 12.4 Pre-adjustments: Dies Grid creation

From the figure 12-3, click on the ***DeltaPsi*** button, to switch the screen to DeltaPsi2 Software.

The procedure below will show step by step how to create and memorize a dies Grid.

From the figure 12-2 (just before the Autofocus Adjustment), click on the next button. The following screen will be displayed:

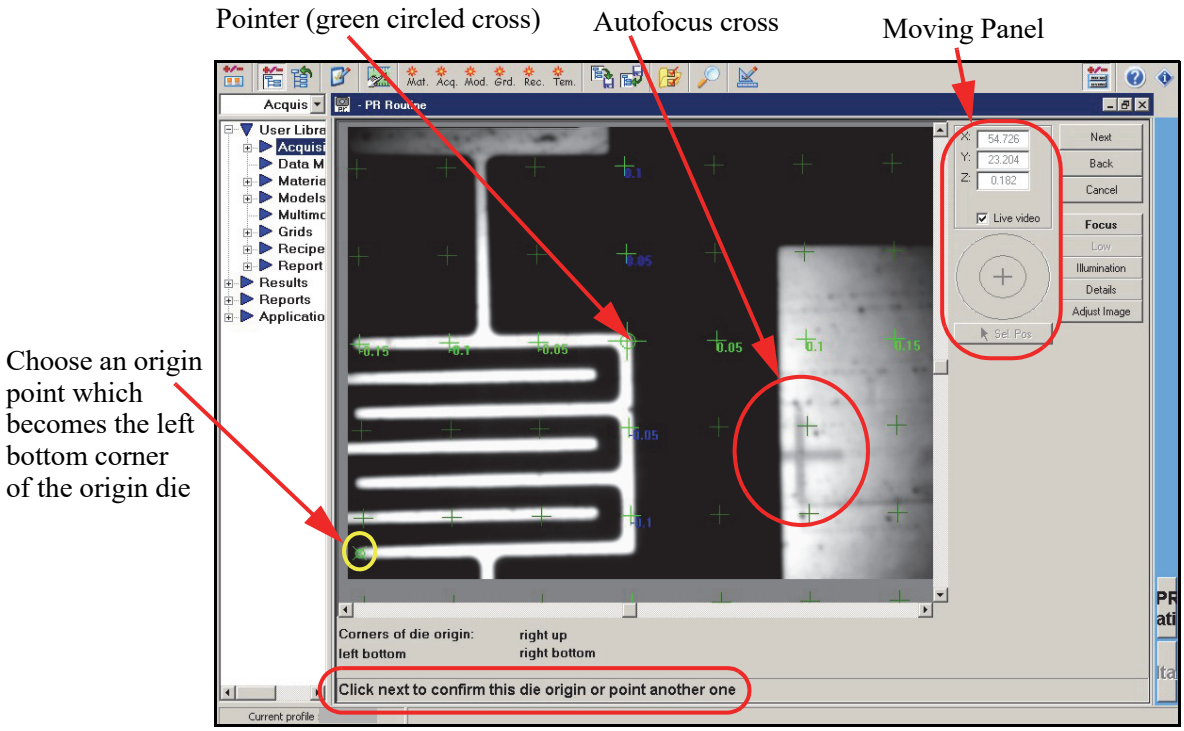
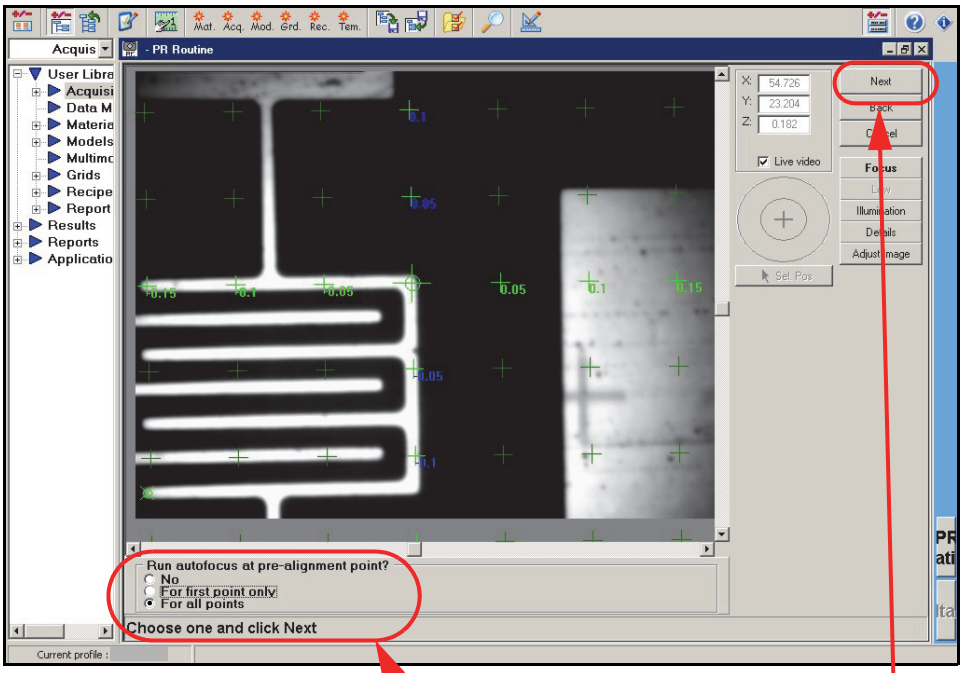



**Figure 12-5 Choosing origin point**

Using the Moving Panel, move the stage and using the mouse click on well distinctive point; see yellow circle on the figure 12-5. At this point, the Autofocus cross must be visible on a clear sample surface. If not, click on another origin point. Once performed, click on the ***Next*** button.

Due to the autofocus principle, the Autofocus cross must be on a reflective surface to be effective.




Select the option: ***for all points*** Then click ***Next***

## Choosing the origin pattern

At this step, an origin pattern must be defined with the following conditions:

- **Choose a shape which is unique in one die, but of course repetitive on the sample. If the dies are not visible on the sample, this is not a problem, just choose a repetitive pattern which will be close to the origin point,**
- **This pattern must be located upper and on the right of the origin point which has been defined previously.**

This pattern will be memorized by the program and will be recognized everywhere on the sample. The selected origin pattern will be the reference origin area from which will be defined the dies Grid. Once performed, click on the Next button.

The figure below shows the current procedure to create the dies Grid (dotted red lines).

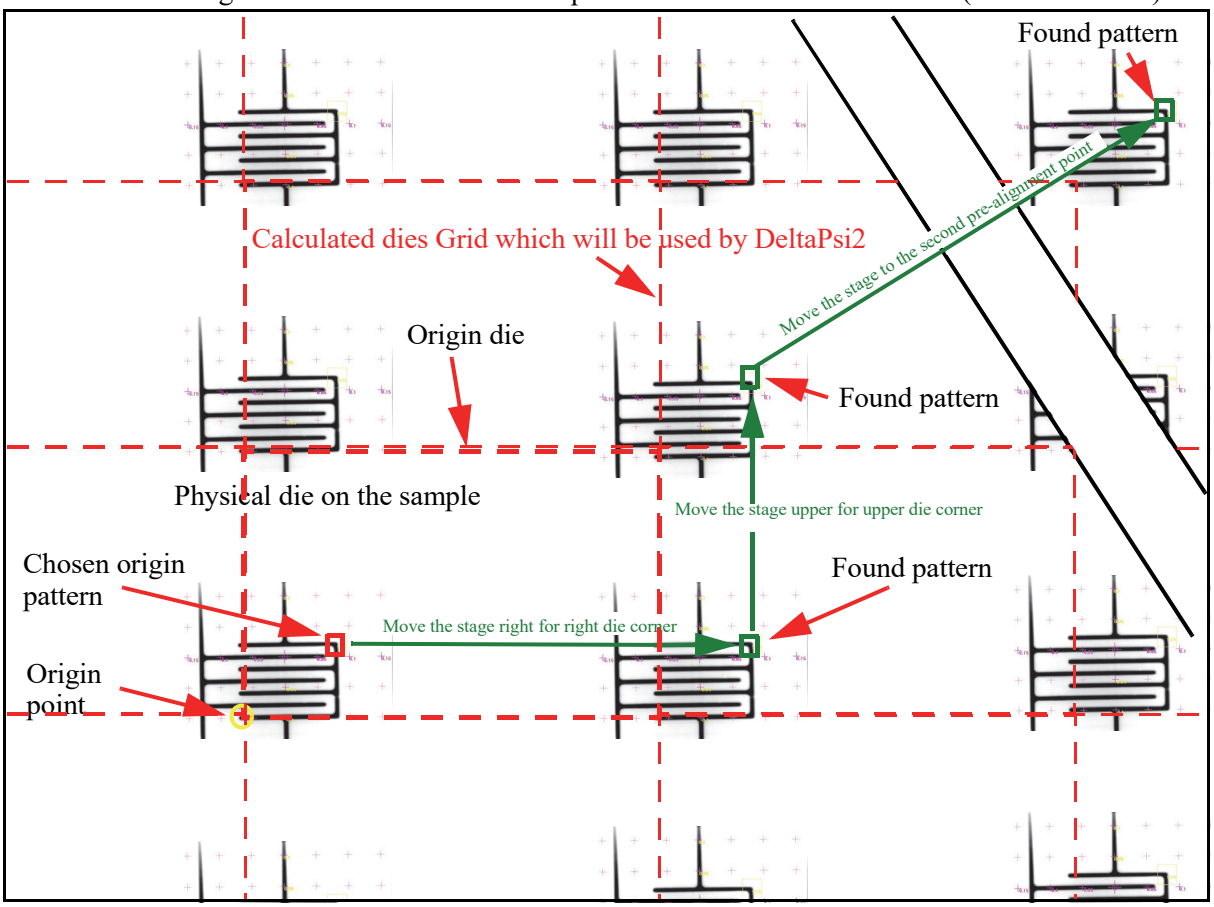
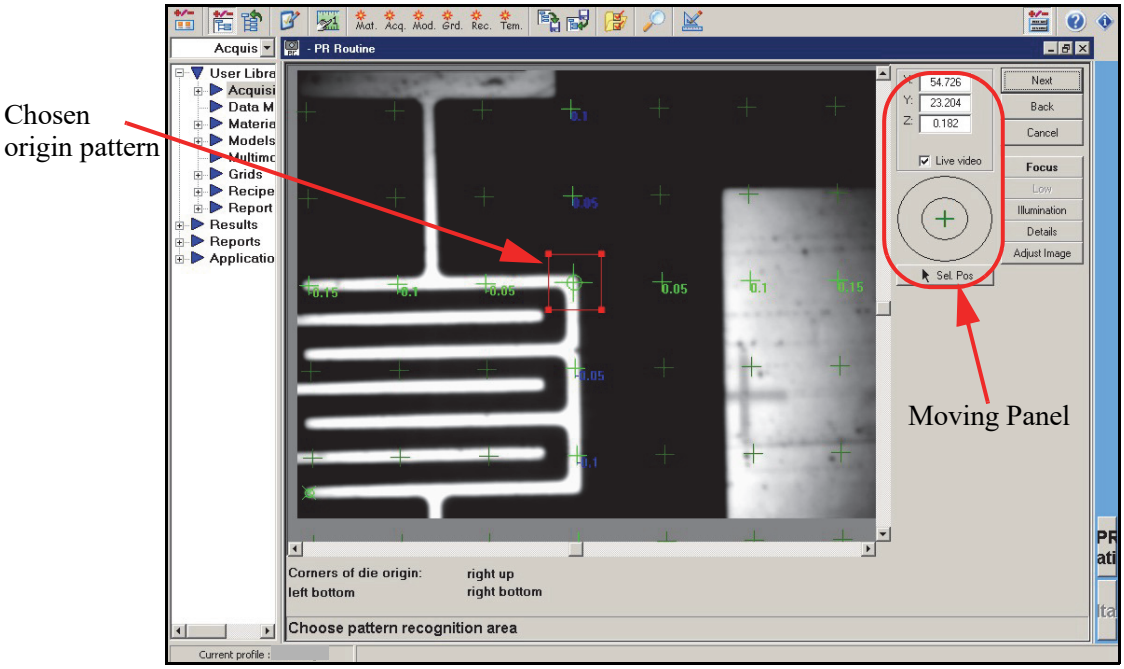



**Figure 12-6 Die size and sample layout calculation**




05/20/2016

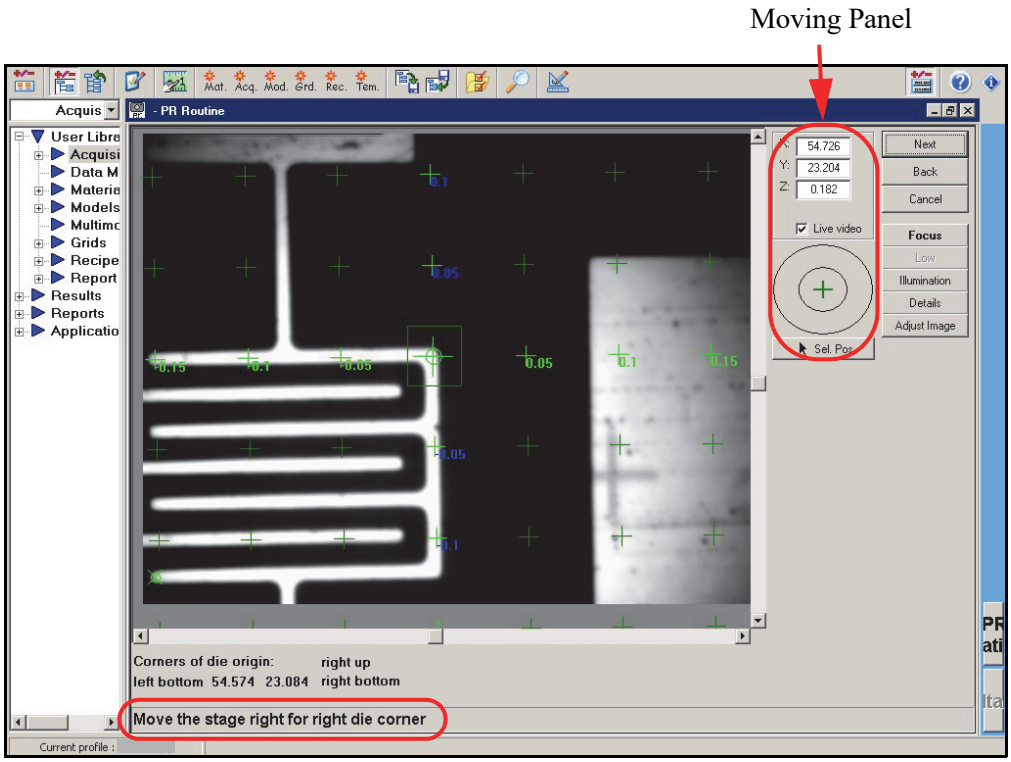



**Figure 12-7 Move the stage to the right die pattern**

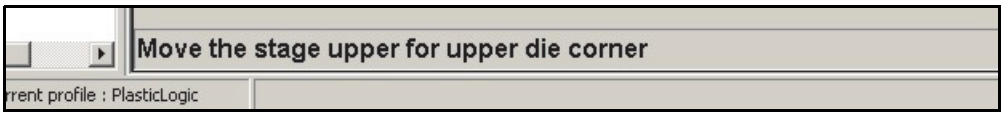
The Software asks the user to move the Stage to the right corner: What does it mean and why to do this?

In fact, step by step, we will program and give useful information to software. The figure 12-6 shows how the software will calculated a «software die» from three adjacent patterns. The 4th pattern, chosen as far as possible from the others, will be used to calculate the XY layout of all the dies.

Finally, knowing the sample size, the die size and the layout orientation, the Software will be able to find the patterns on all dies of the sample.

Back to our figure 12-7: using the Moving Panel, move the stage to the adjacent right die. To perform it faster, add the theoretical die width to the X field value. In our example the current X position is 54.726 mm and the die size is 0.500 mm: the entered value is 55.226 mm. Be sure to see the pattern on the screen. Press the ***Next*** button to validate. Immediately, the Software should detect the pattern and displays in the comments area the percent score. If the score is low, it is recommended to choose another pattern shape. If the detected pattern is fine, click on the ***Next*** button to validate.

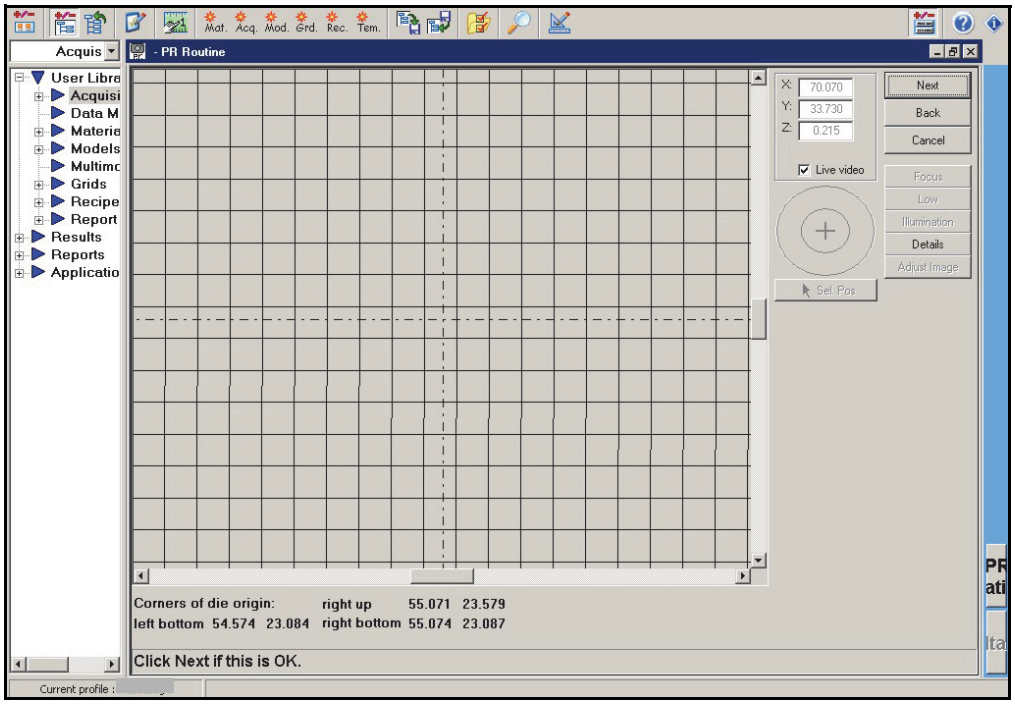
A new screen is displayed. The «comments» ask the user to «***Move the stage upper for upper die corner***». Proceed as for adjacent right pattern described higher. Now, add to the Y field value, the theoretical die height. If the pattern is found, validate by clicking on the ***Next*** button.



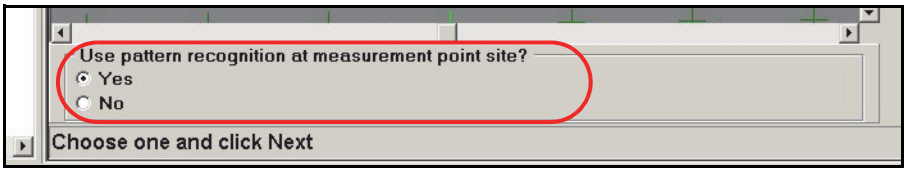

A new screen is then displayed.

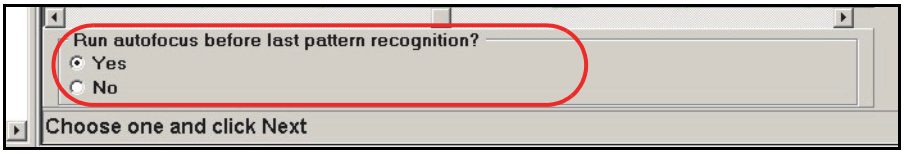

The comments ask the user to «Move the stage to the second pre alignment point». The goal is to find a pattern as far as possible from the first defined pattern. Place the found pattern on the center of the screen then click Next to validate. The pattern should immediately detected and after the validation, the software will calculate the orientation position (θ shift).

Once validated the following screen displays the calculated dies layout.




Click on the ***Next*** button to validate. The same screen with different comments will be displayed. Answer to the questions as shown on the figures below. After each answer, click on the ***Next*** button.




## 12.5 Adjustments: measurement points setup

Next step allows the user to select a new pattern anywhere inside the origin die which has been defined from the previous steps. This new pattern will become the reference position compare to the measurement points which will be selected on the next screen.

The new pattern must be defined on the same «Software die» as the pattern origin (see Figure 12-6, “Die size and sample layout calculation”, page 149). In our example (see figure 12-8), we have defined the new reference pattern over the «origin» pattern.

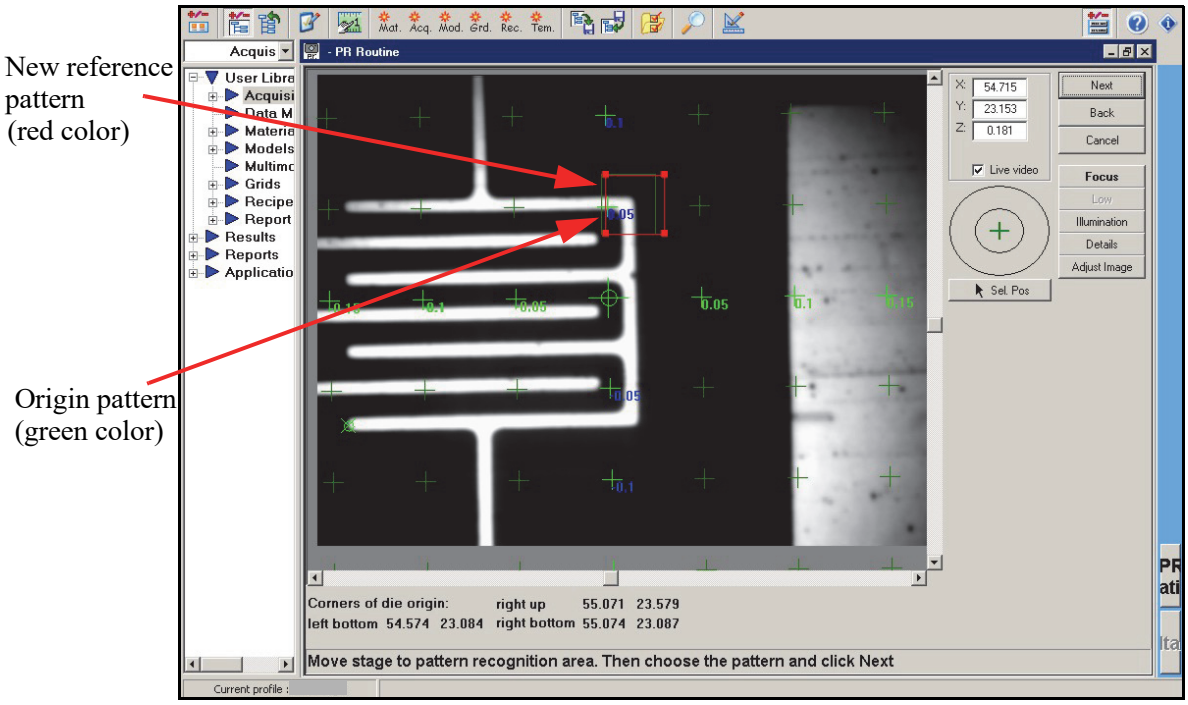



**Figure 12-8 reference pattern for measurement points**

05/20/2016

Once defined, click on the Next button. The following screen will be displayed. This important screen allows the user to define the measurement points. Click as many measurement points as necessary inside the current «Software die» area.

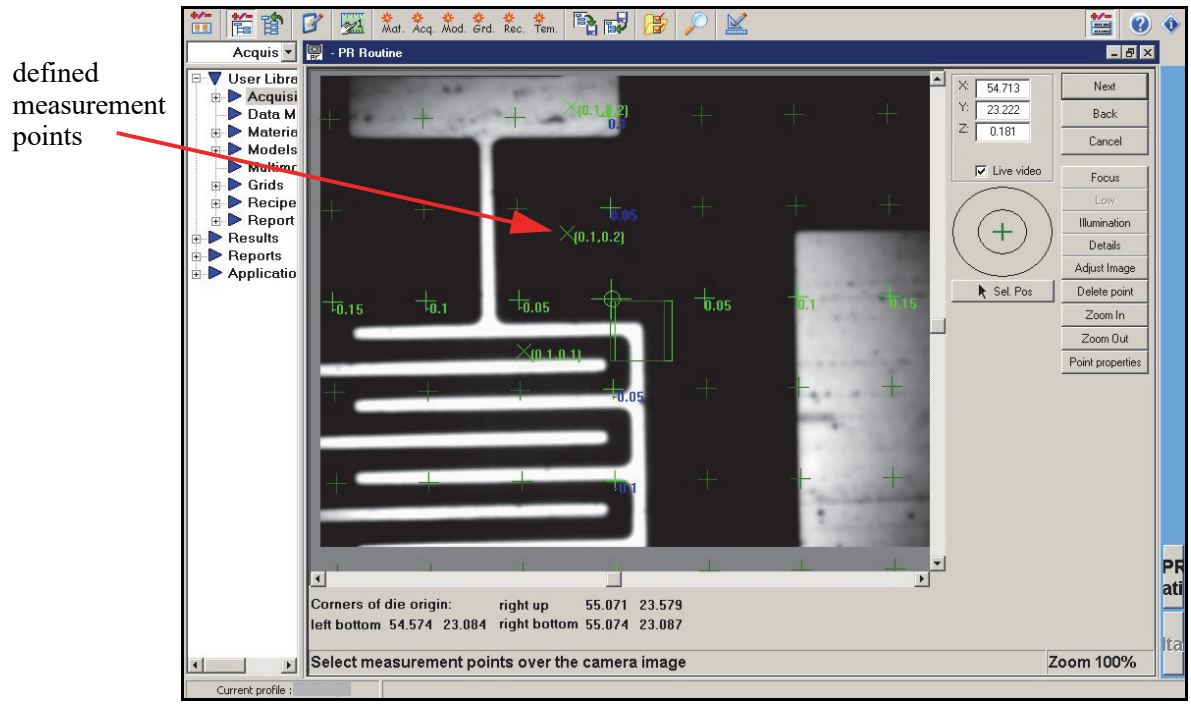



**Figure 12-9 Measurement points**

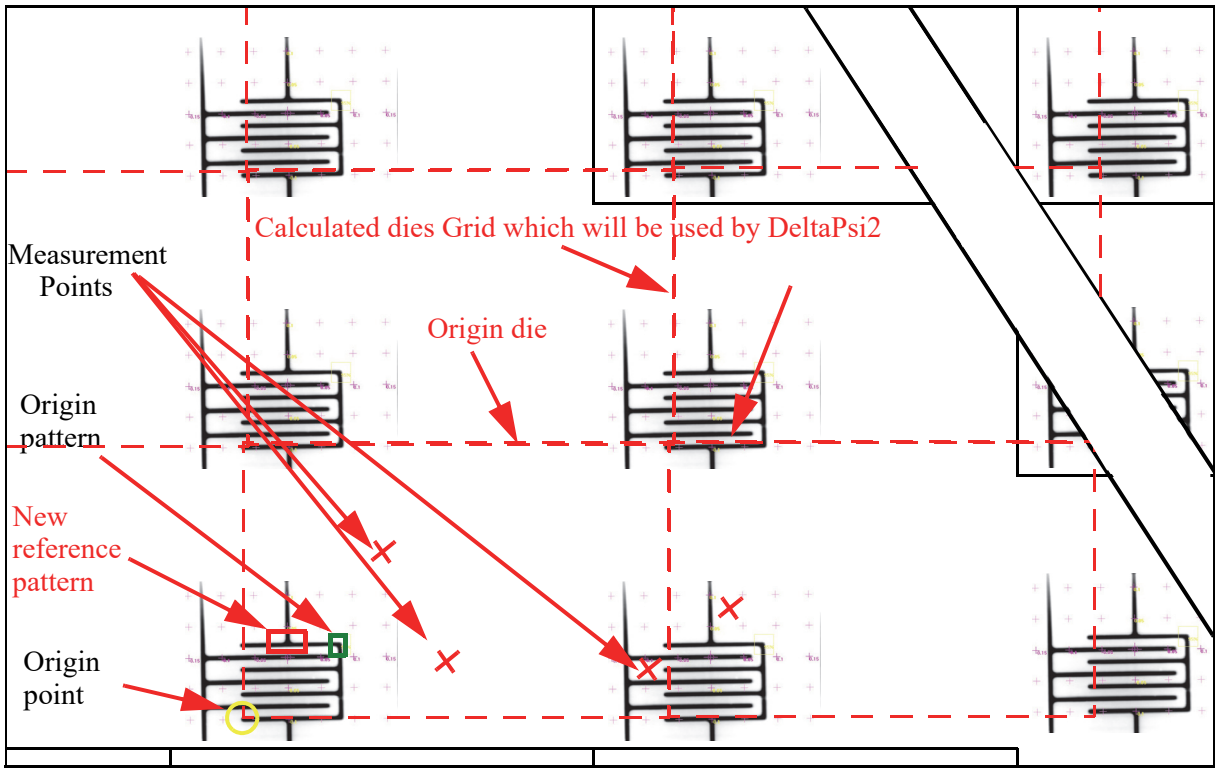
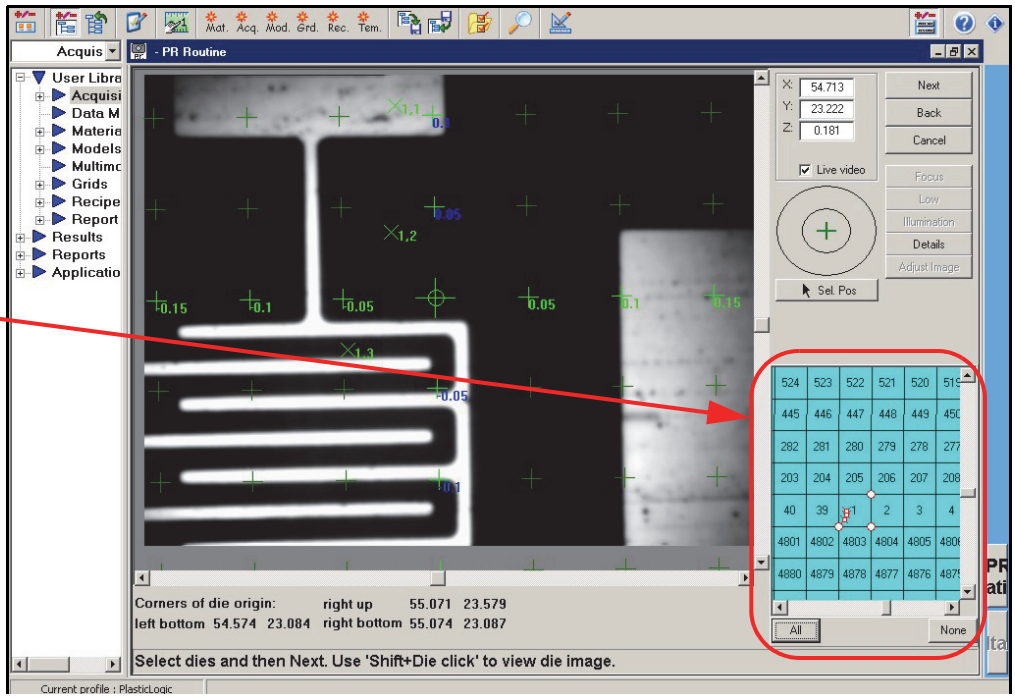



**Figure 12-10** Measurement points layout

05/20/2016

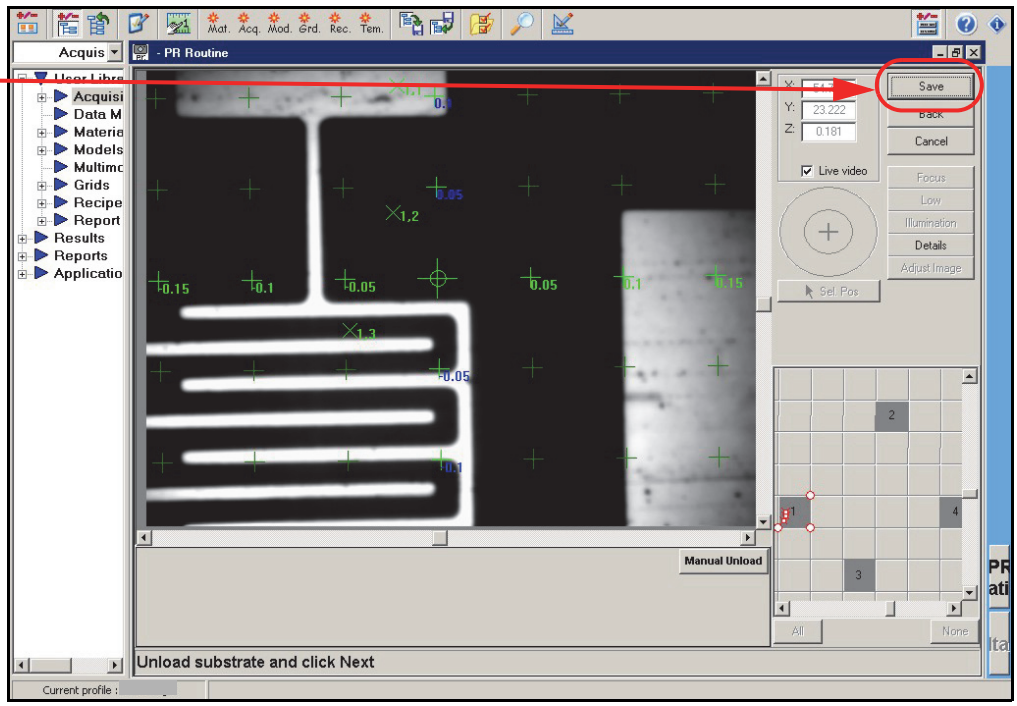
Once the measurements points has been selected, click on the Next button. The following screen is displayed. Using the die selection panel, select the dies (and related measurement points) to include in the measurement process. Click on the ***Next*** button to validate.

Die selection panel



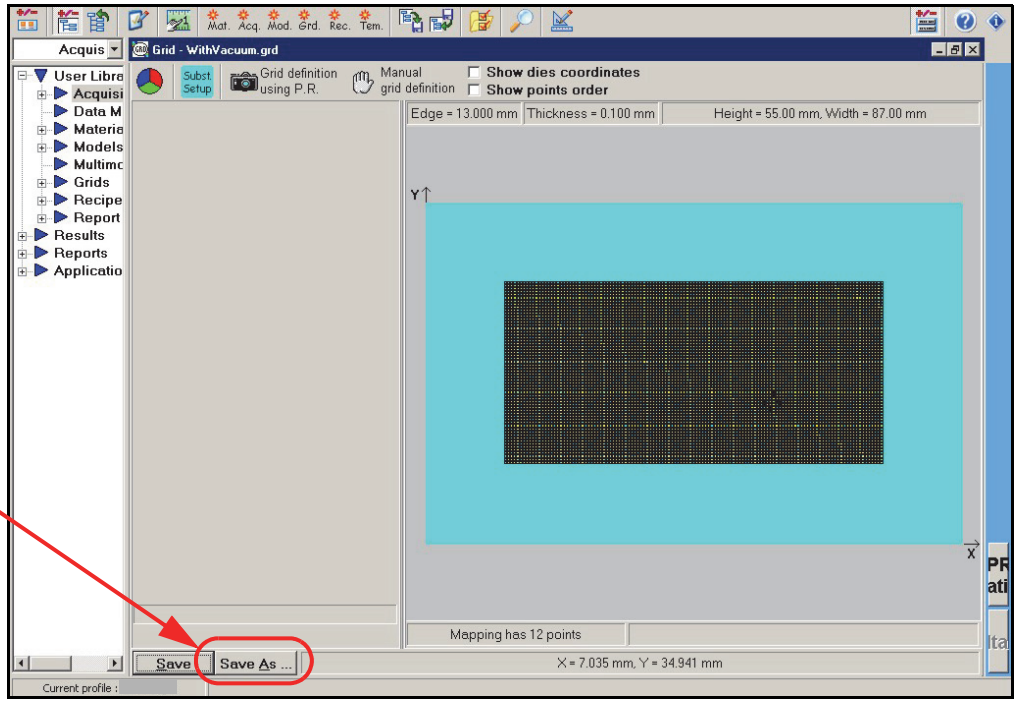

The following next screen is then displayed. The selected dies become dark-greyed. Click on the Save button to validate the PR Station Setup. The sample can be now removed if the measurements Recipe can be run later.

Click Save to validate the Grid



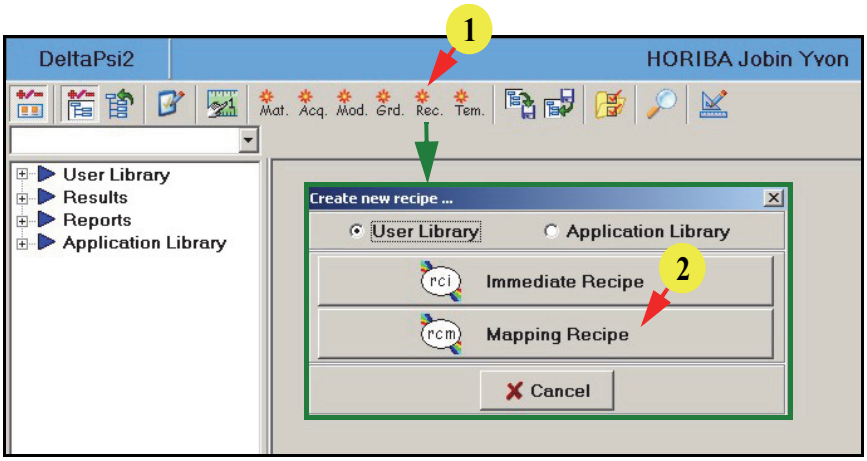

The resulted Grid is then displayed. Click on the Save As button to store the file with a well identified name. Later, this file name will be retrieved to a Recipe.

Click on the ***Save As*** button




## 12.6 Execution: Recipe Creation

The Grid for mapping has been previously created. The Mapping Recipe must be then created to perform measurements on a patterns sample.



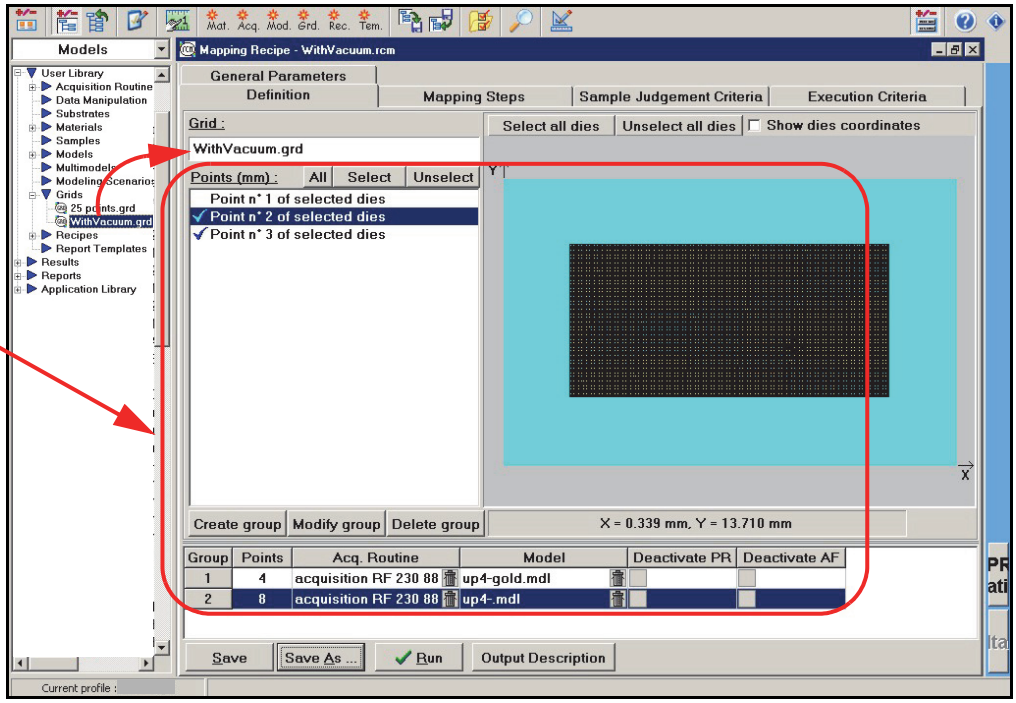

From the TreeView menu, select the Grid file previously saved and drag and drop it in the Grid field (see figure below).

Create as many groups as necessary. Each group can included selected points and can be applied to a specific Acquisition Routine with a specific Model.

Do not check in the last two columns as the system includes a Pattern Recognition and the AutoFocus.

**Further information is described in the chapter "Mapping Recipe using a defined complex grid" on page 267.**

05/20/2016




Once performed, click on the Save As button to store the entire Mapping Recipe in the Recipe folder.

Load the Sample on the Stage and click on the Run button to launch the measurements.

# 13 Modeling

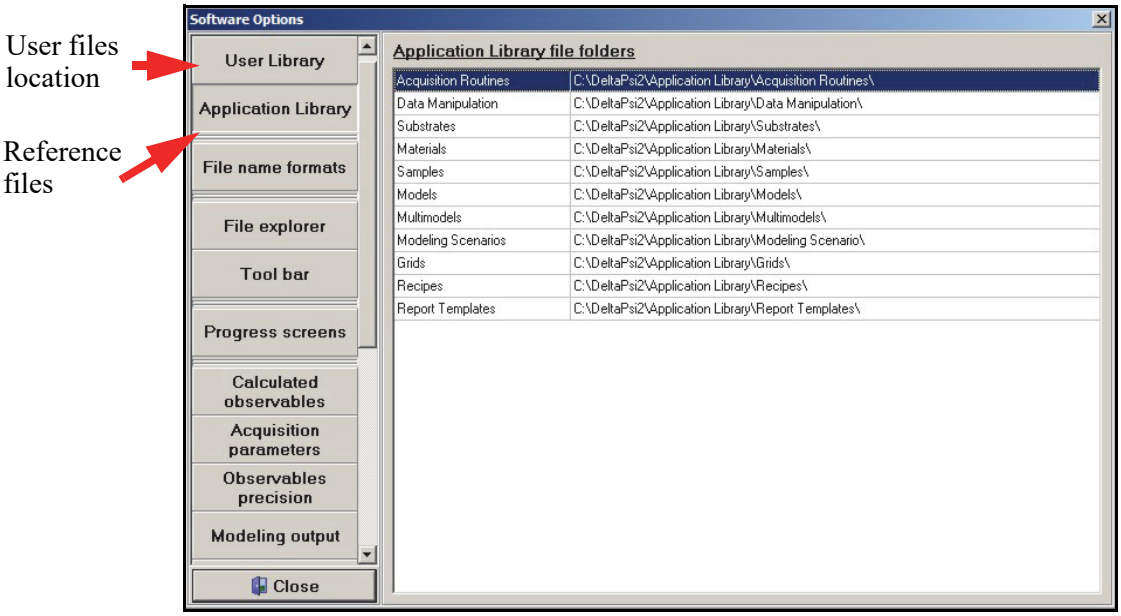
The Modeling feature is the most powerful parts of the DeltaPsi2 Software. The user can build a model then simulate and fit the spectroscopic characteristics of this sample. Simulation is useful if no sample is available and the user wants to verify spectroscopic/optical characteristics.

## 13.1 References and other materials

DeltaPsi2 Software is delivered with many reference and materials files located in the Application Library of the TreeView folder. The ***reference*** files have been built from the Palik reference book or developed by our scientists. The materials files have been developed by our scientists. They are used to built models in DeltaPsi2.

These reference and materials files can not be deleted or modified in the Application Library. Whatever you create in the DeltaPsi2 Software, the results will be saved in the User Library.

From the **Software Options** screen (see "Software Options" on page 14) , it becomes possible to see where the reference and material files are located.



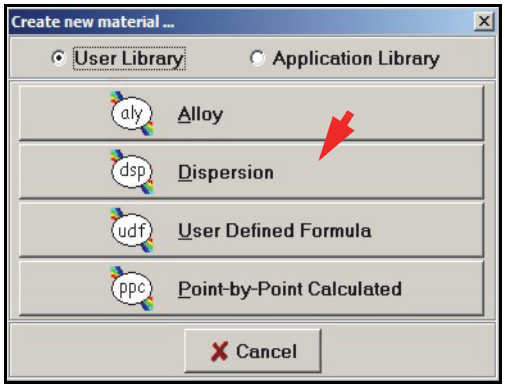

## 13.2 Material Building

### 13.2.1 Launching the build material

Build a material or adapt a material during the Modeling session is an important part to reach good results. The ***Build Material*** feature groups four working screens which allow the user to change some material specifications, the related Dispersion formula (group of formulas) which will be used during the Modeling process. In this chapter we will detail the capabilities range for these materials.

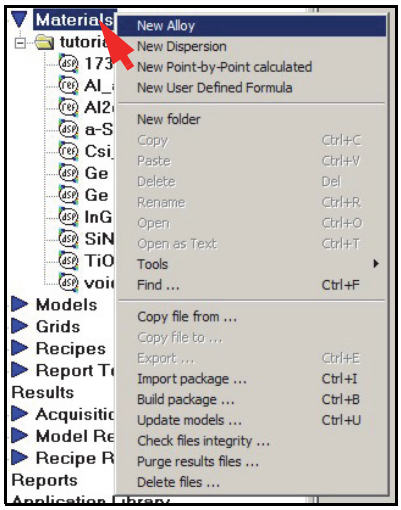
Double-click on the «Mat.» icon from Button Bar


Then select the working material feature from the Selection screen:



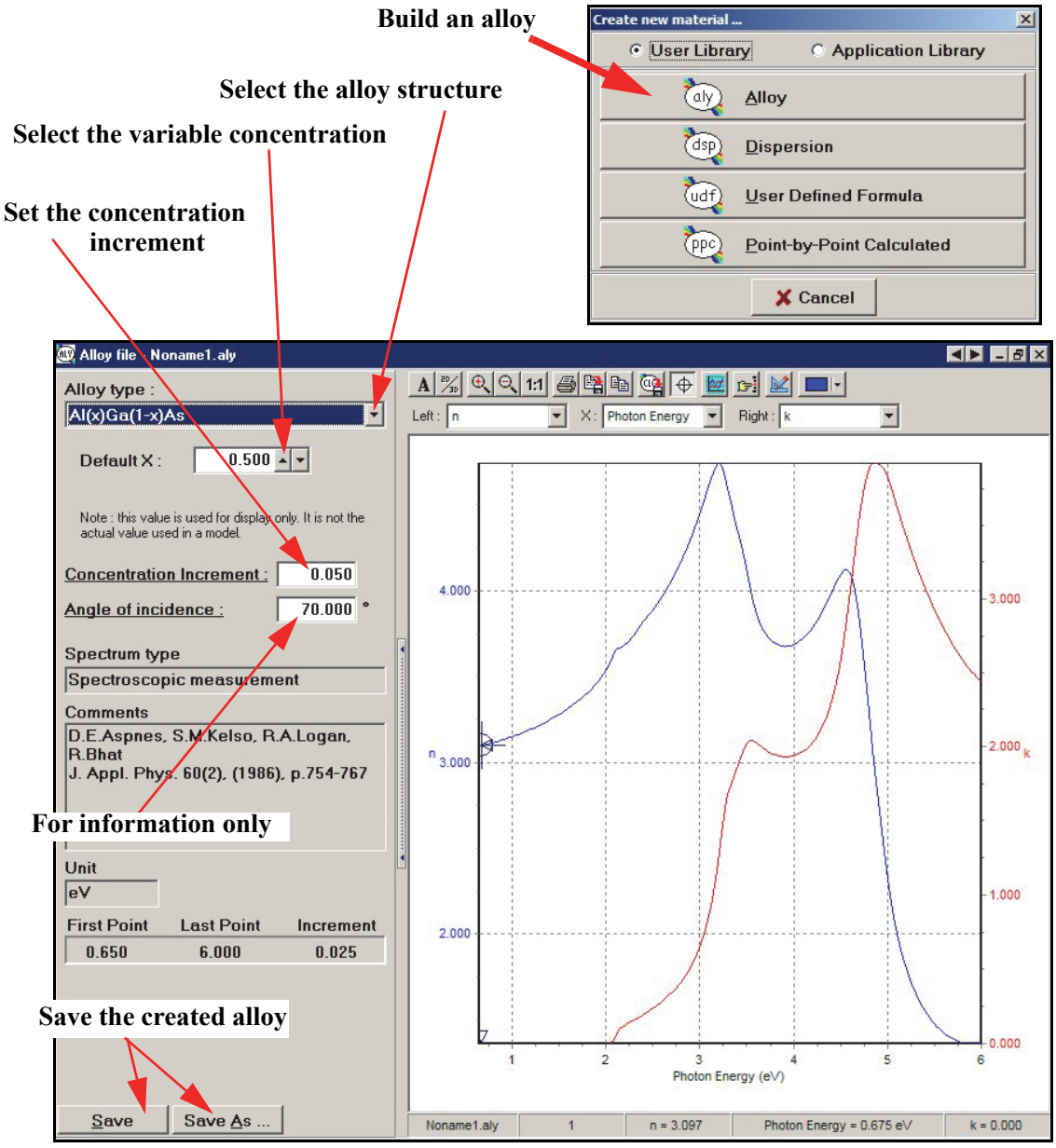

**OR**

Right-click «Materials» from TreeView



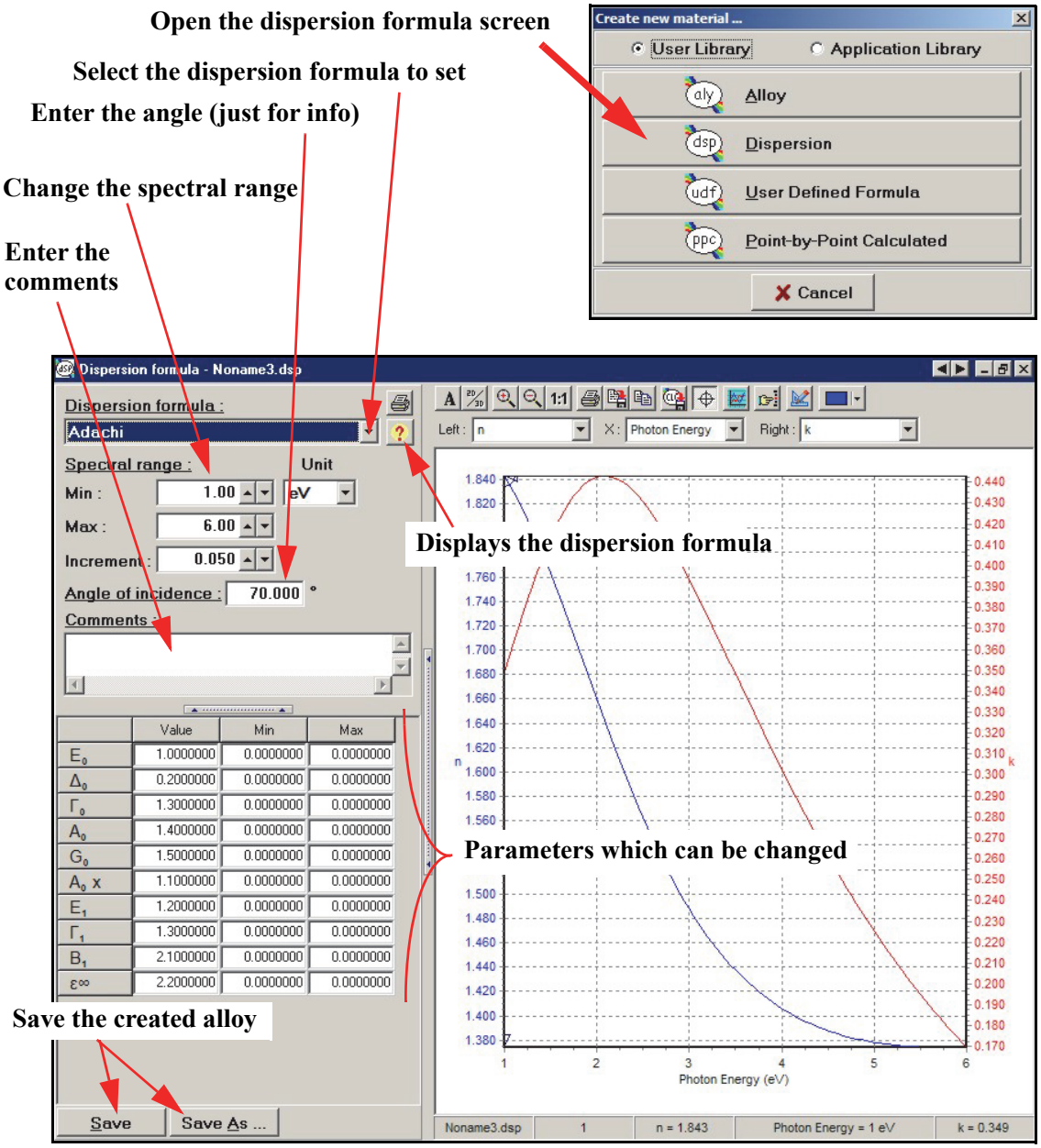

### 13.2.2 Build an alloy

This feature allows the user to build an alloy using a structure preset. Select a structure and set the variable concentration.




### 13.2.3 Set a Dispersion Formula

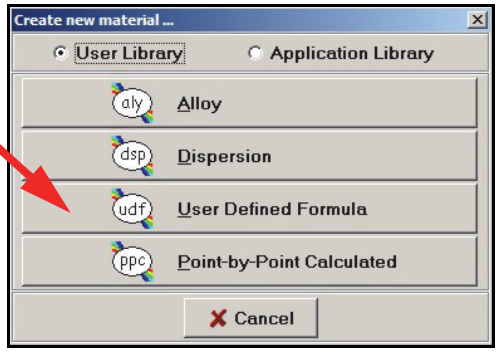
Using this feature, a standard Dispersion Formula can be adapted for a specific spectral range or defined from a defined data points. Moreover, each of the formula parameter can be individually set. The figure below shows the an example of such a screen.



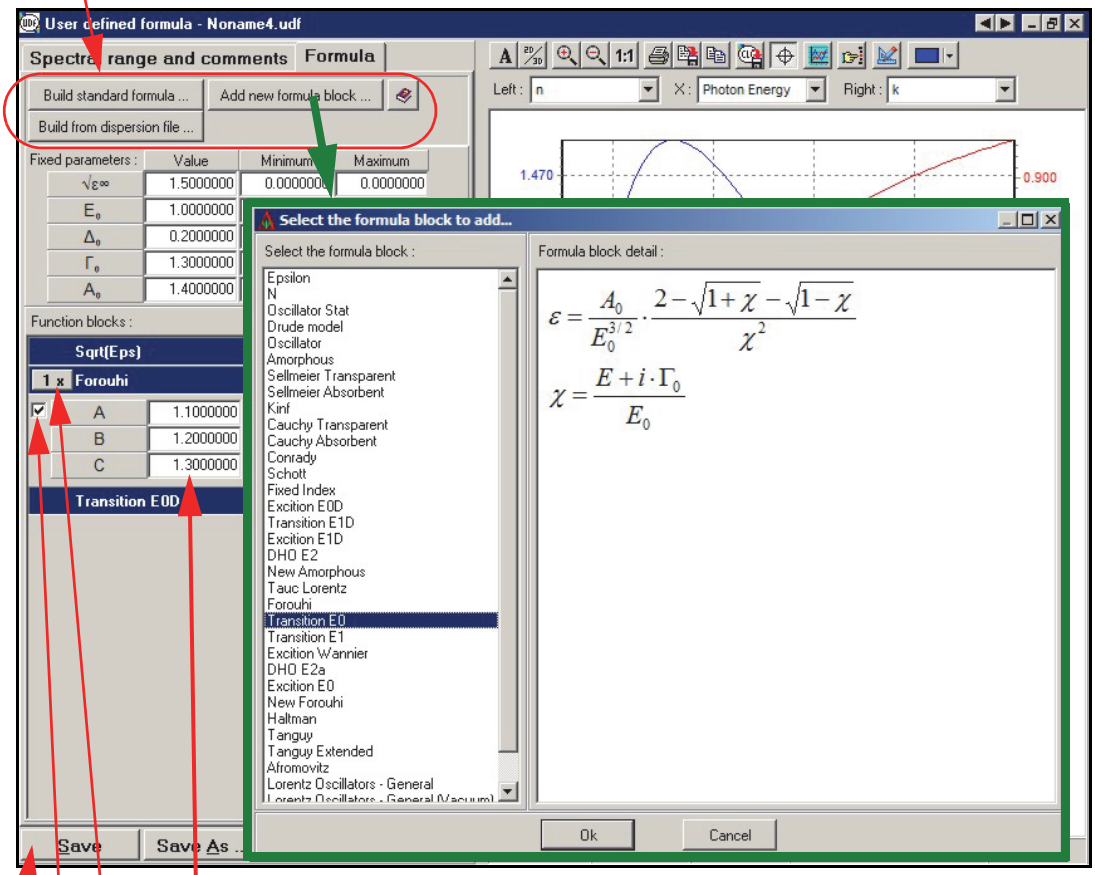

### 13.2.4 User defined formula

This feature offers an additional step to set the Dispersion Formula. It becomes possible to build a formula from many standard formula or by adding a formula block or even from an existing Dispersion Formula located in the .DSP files. The chained formula are not limited by the size.

**Open the User defined Formula screen**




**Choose the type of dispersion to add**




**Parameters can be adjusted from all of the blocks**

**Click on this button to duplicate the dispersion block**

**Check-in/check-out to remove the dispersion block. The dispersion curve is real tim displayed the current status.**

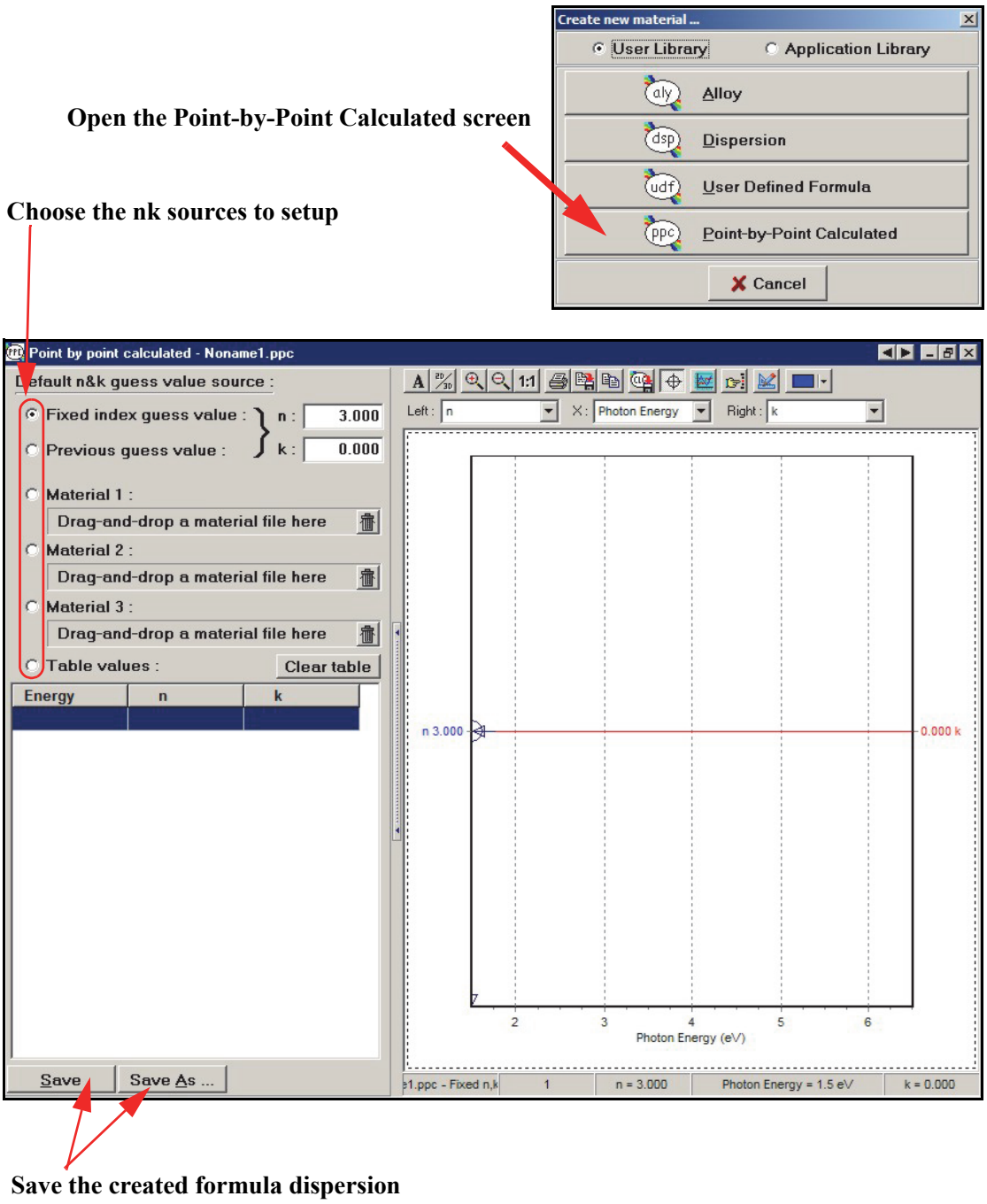
**Save the created formula dispersion**

### 13.2.5 Point by Point Calculated (n, k)

The PPC (Point by Point Calculated) feature must be used to create a material from «scratch» or using minimal properties for n and k parameters. Advanced use and full features can only be reached when the Model structure has been created

05/20/2016

and the session oriented to nk fit. This chapter describes the various ways to use the PPC screen.




**How to use this screen ?**

It is important to understand how to control the screen. In fact some functionalities can be used directly from this screen, some others from the Modeling screen.

The following materials can be created directly from this screen:

- **Fixed index nk:** enter the fixed nk values then click on the Save As button to store the new material.

- **Duplication of an existing Material:** drag and drop a Material from the TreeView to the appropriate field, select this field then save it using a new name.

---

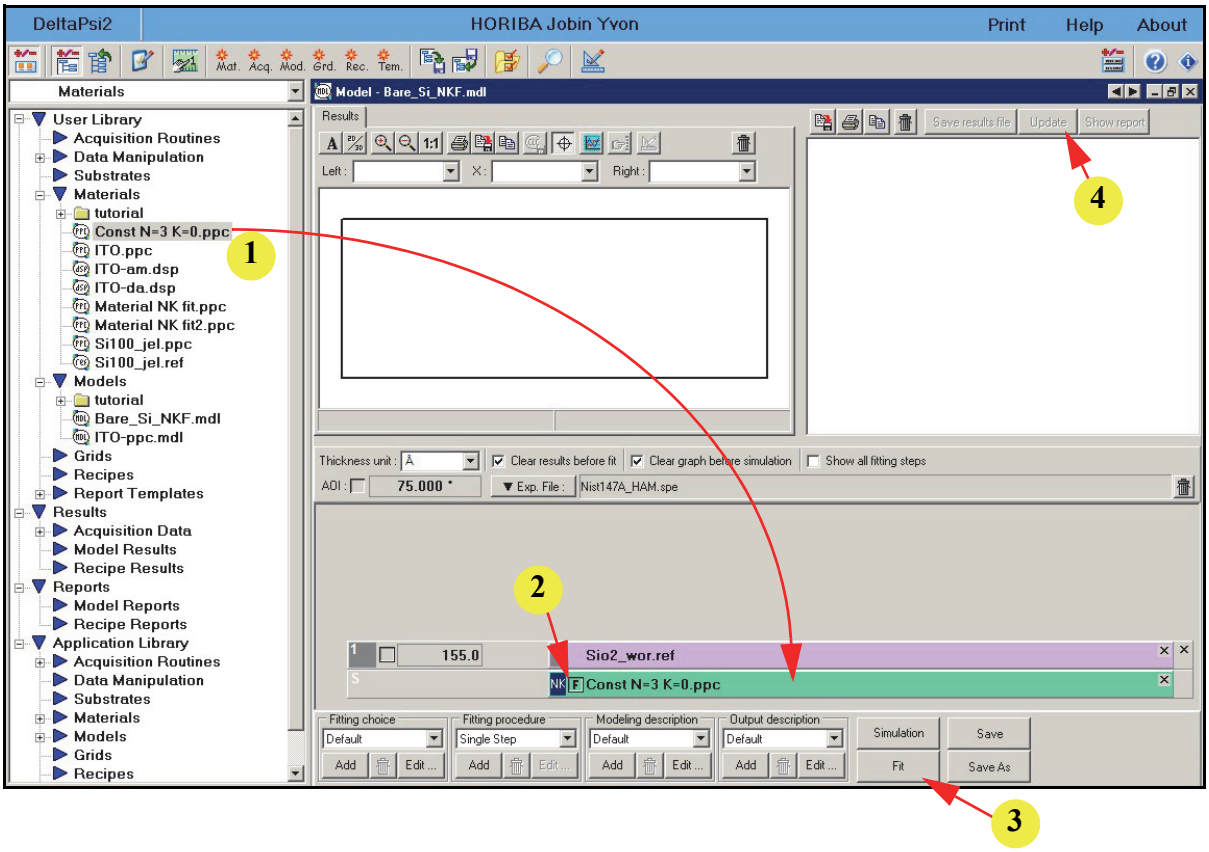
**«nk fit» from the Modeling screen**

From the Modeling screen, the PPC screen is an intermediate interface which allows the user to setup, save and automatically update the material located on a layer, during the «nk fit» process.

## 1 Build a model

For a general build of a Model, please refer to "Model Building" on page 166.

Build a model and insert a .ppc material on a layer. This material will be used for fitting.




## 2 Fitting process

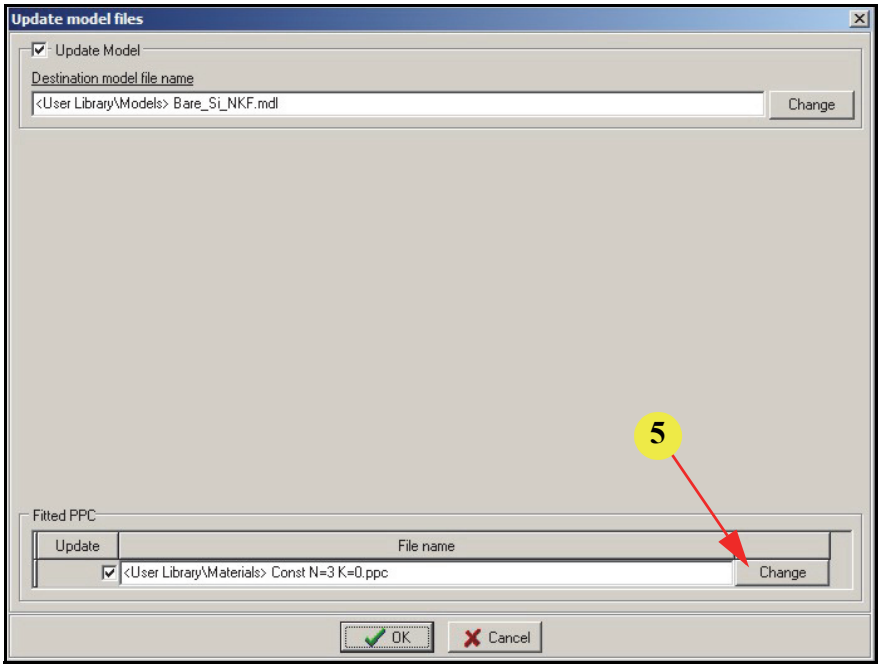
Click on the ***Fit*** button. The fitting process will use the n and k values defined, by default, in the file. In the example above, the file «**Const N=3 K=0.ppc**» has a constant value for n and k, but the fitting process uses these constant values as a guess value for fit research.

Result of the fit is immediately displayed.

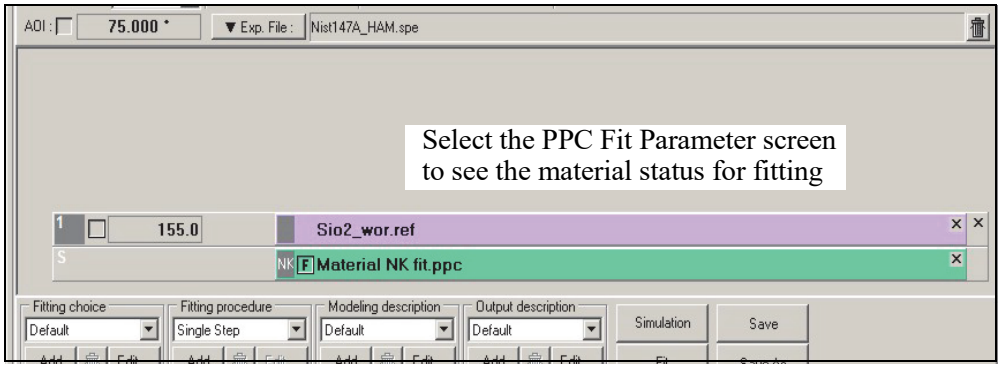
## 3 Updating n&k values

Once the fitting process has been proceeded, the found n&k values can be updated and visualized using the following steps:

- Click on the ***Update*** button 4 , the following screen will be displayed:




- Click on the ***Change*** button 5 and enter a new filename, then save it. The main screen is then displayed, and the new material name is directly updated on the layer.




**IMPORTANT**: at this step, it is important to understand the following

- Clicking on the small square 

will activate or not the fit on this material, but will also display the fitting choice ***PPC Fit Parameters*** screen. This screen shows the current status for this material; it becomes easy to change completely the n&k values.

05/20/2016

- Clicking on the name of the Material 

will only display the **original status** of the material. **Do not use this screen (Point-by-Point calculated) to change the n&k values, they will not be functional if a fit is launched**.

- Once the n&k setup has been performed, a new fit can be launched.

**Choices are «1» or «3»:**
**«3»: each n&k point is averaged with the previous and the next datapoint. The result is used during fitting.**
**«1»: each n&k point is used as it during the fitting.**

**k adjusted means that each k will be used during the fitting.**

**k not adjusted means that each k value will not be used for the fitting but retrieved as it from the selected source.**

**k constant means that only one value will be used over the entire wavelength. In our example, whatever the selected source, used k will be 0.**

**Select the n&k source**

**If the table values has been selected, each n&k value can be changed**

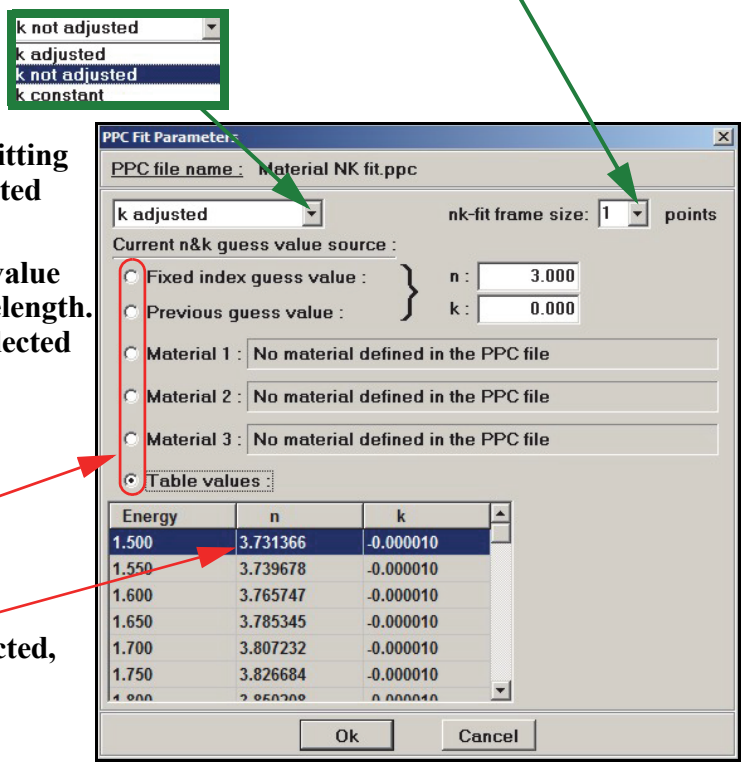



**Figure 13-1 PPC Fit Parameters screen**

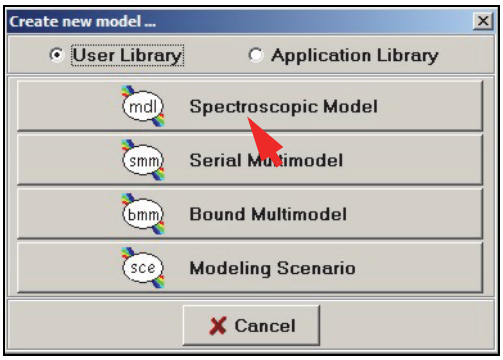
# 13.3 Model Building

This chapter will explain how to build a model and how to adjust parameters.

## 13.3.1 Launching the building screen

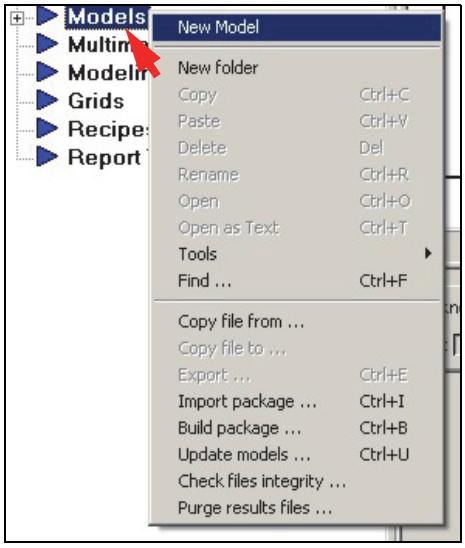
Double-click on the «Mod» icon from Button Bar


Then select the «Spectroscopic Model» type from the Selection screen:



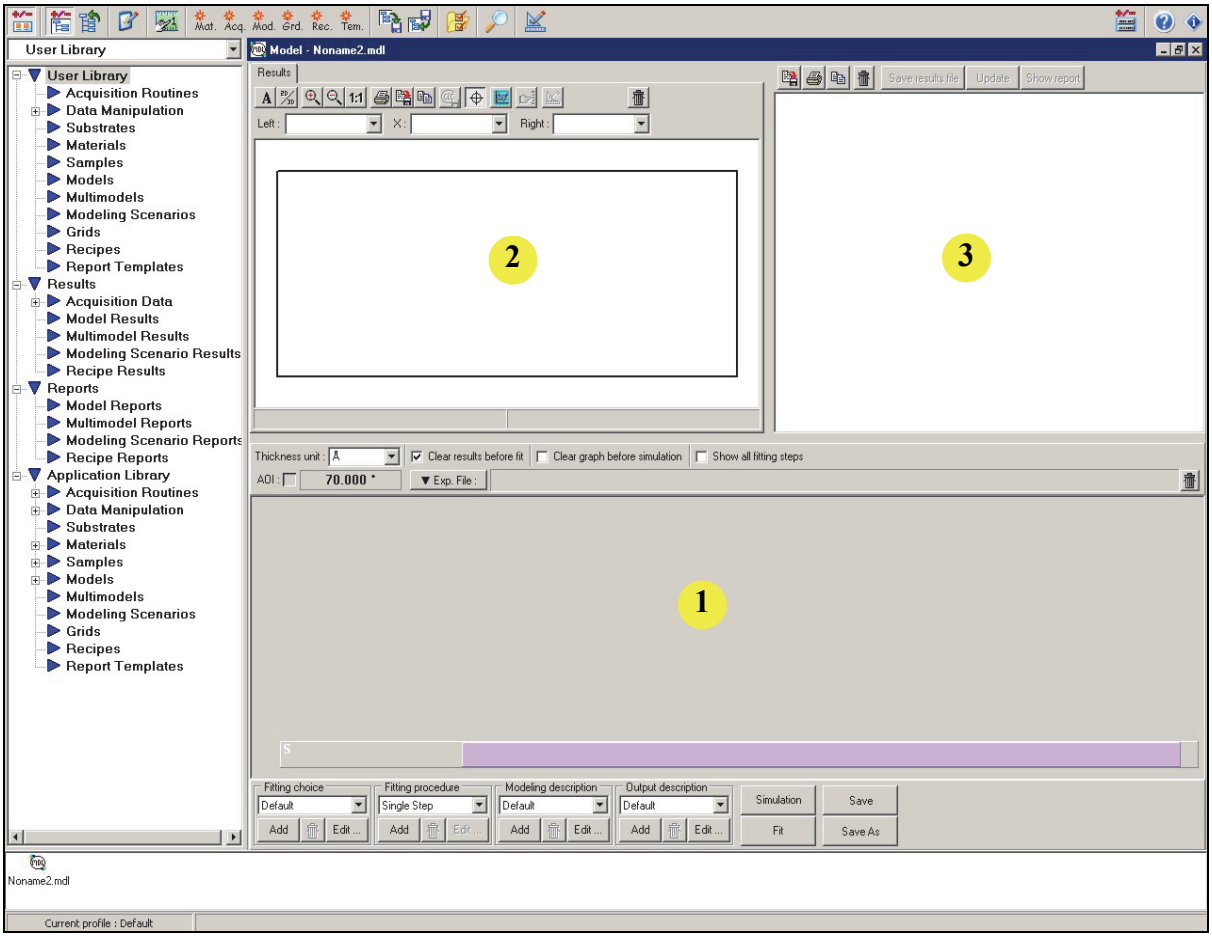

**OR**

Right-click «Models» from TreeView




05/20/2016

**The following general blank screen is displayed:**




**1 Sample Structure area**

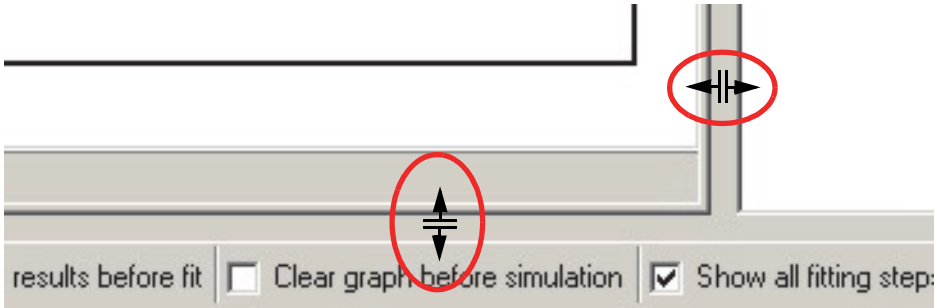
**2 Results Graphics display area**

**3 Fitting Results area**

**Figure 13-2 Modeling screen layout**

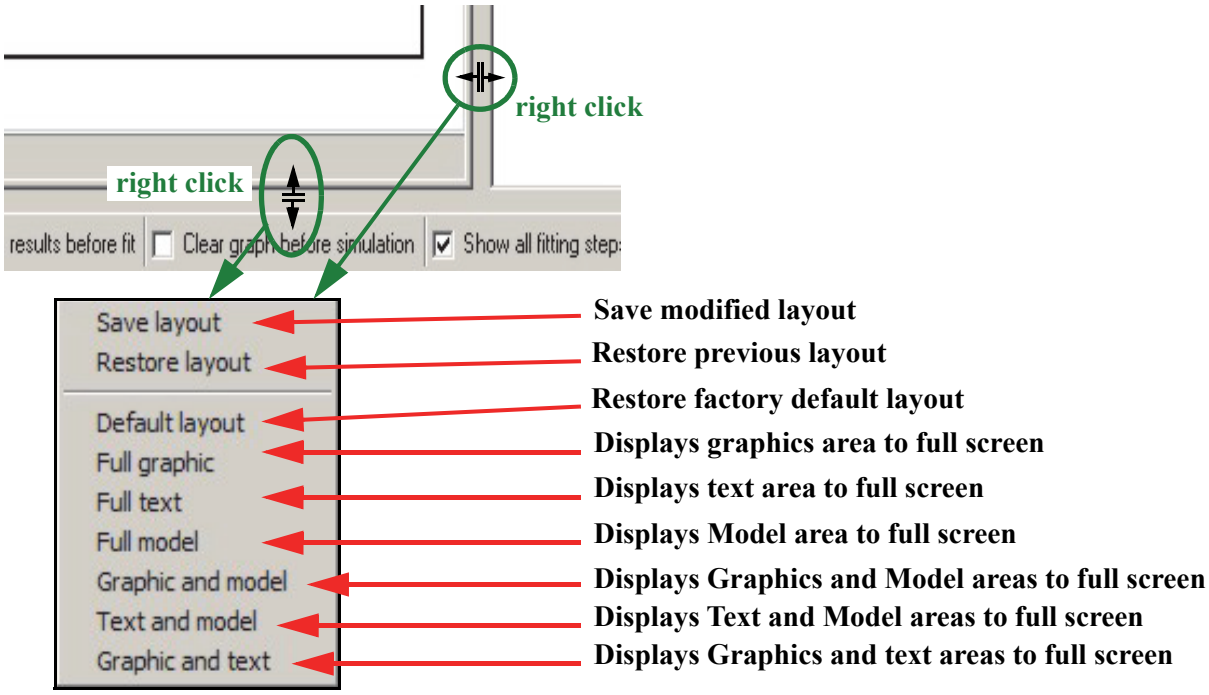
### 13.3.2 Screen Layout Setup

The three area shown above can be setup to match your needs. Just move the mouse cursor over the window frame This management is summarized in the figure below:



- **Extended features**

Right-click on Model windows limit to activate additional features




### 13.3.3 Tools to build a model

Read carefully this tutorial. It contains all the commands and shortcuts which will help you to create a model.

Before any explanations, move the windows frame to adapt the layout. The ***Sample Structure*** area is totally open.

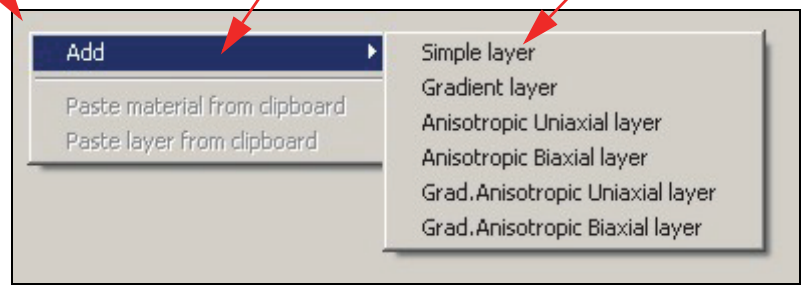
---

**Default blank model area**

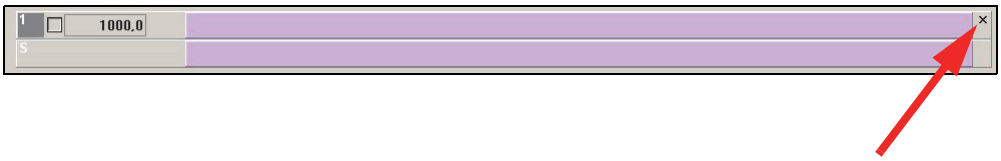
By default, a non defined substrate is displayed and a Sample and a Model default filenames are created.

**How to add a layer ?**

**1. Right-click on the blank area** **2. Select «Add»** **3. Select the layer type**

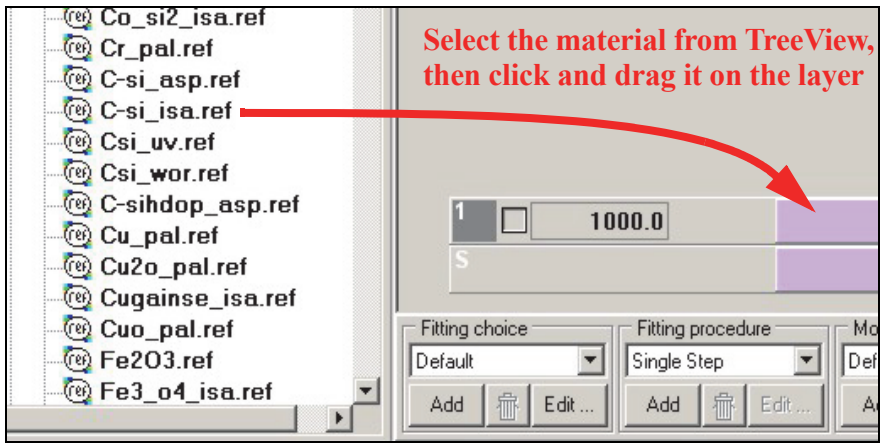



**Result:**




To remove a layer, click on the little cross located on the right of the layer

**How to add a material to a layer ?**




This action can be repeated as many times as necessary with a limit of three materials per layer.

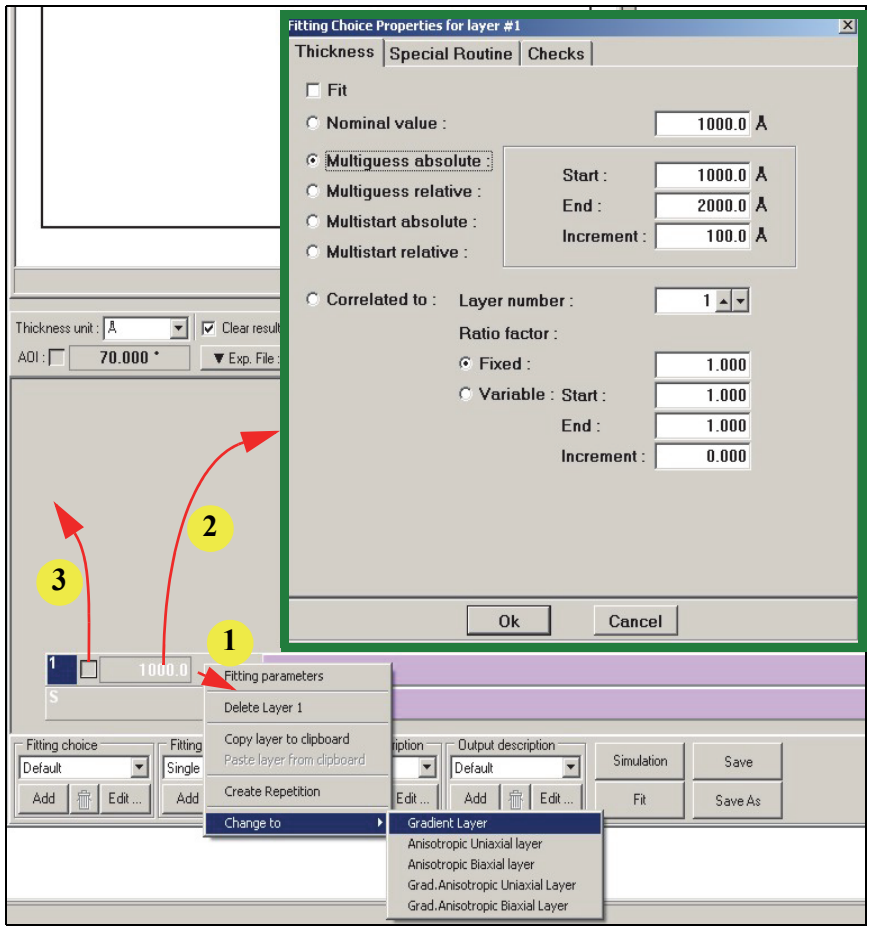
### Manipulation on a layer

1 Right-click to duplicate, delete or change the layer properties

2 Click on the thickness field to display the ***Fitting Choice Properties*** Screen

3 Click on the toggle button to activate this layer to the fitting process

NOTICE: the same parameter is located on the top of the ***Fitting Choice Properties*** Screen




### Advanced Manipulations on Layer

The figure below shows an example of a complex materials structure.

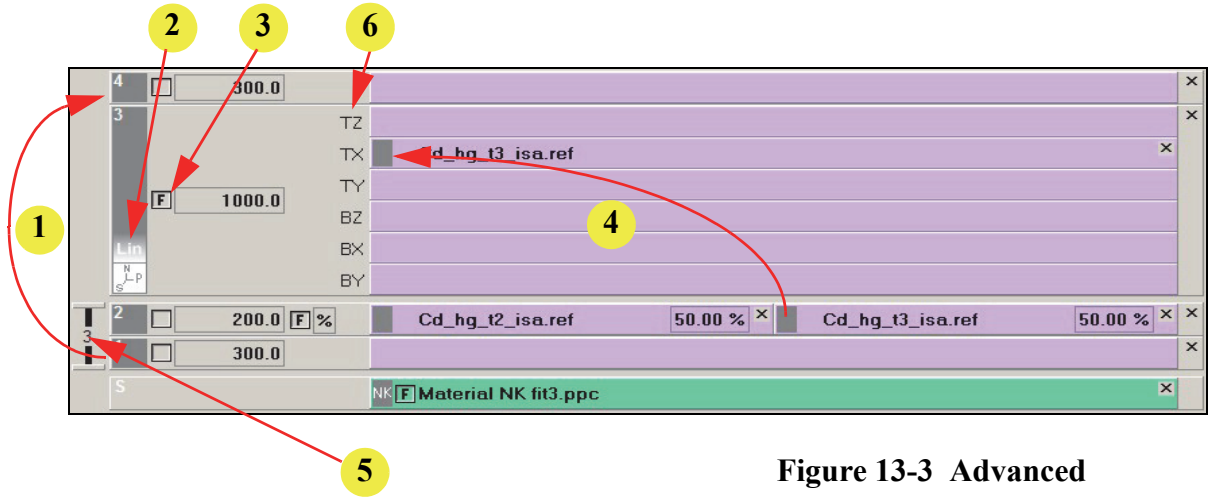



**Figure 13-3 Advanced**

1 To copy a layer on the top or between two created layers, click on the layer number, then drag above the other layer numbers and release the mouse. The new layer will be automatically inserted.

2 **Layer information:**

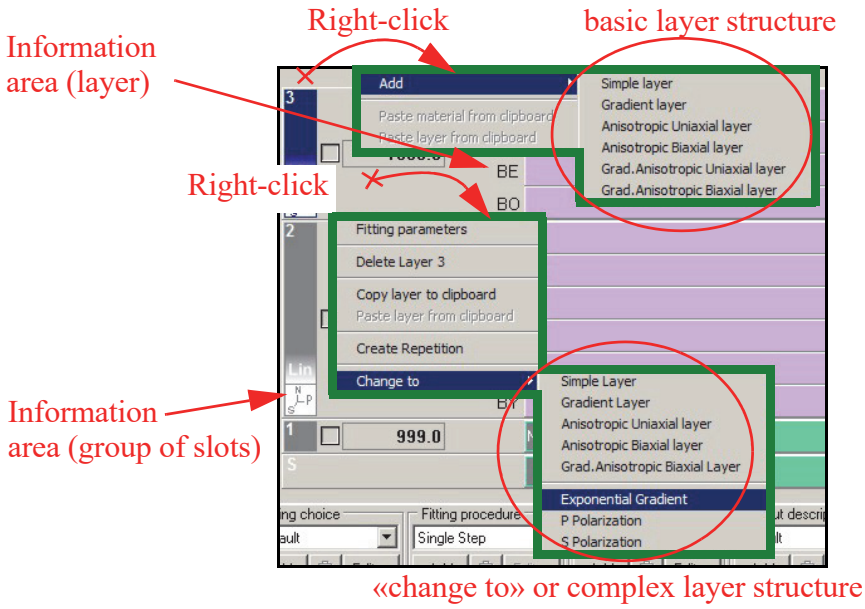
When a layer is created, a specific information is displayed on the layer diagram. The figure below explain how to create and setup a layer. To create a

6 layer, two steps could be necessary:

1. Right-click on the area outside a layer: choose one of the six layer structures,

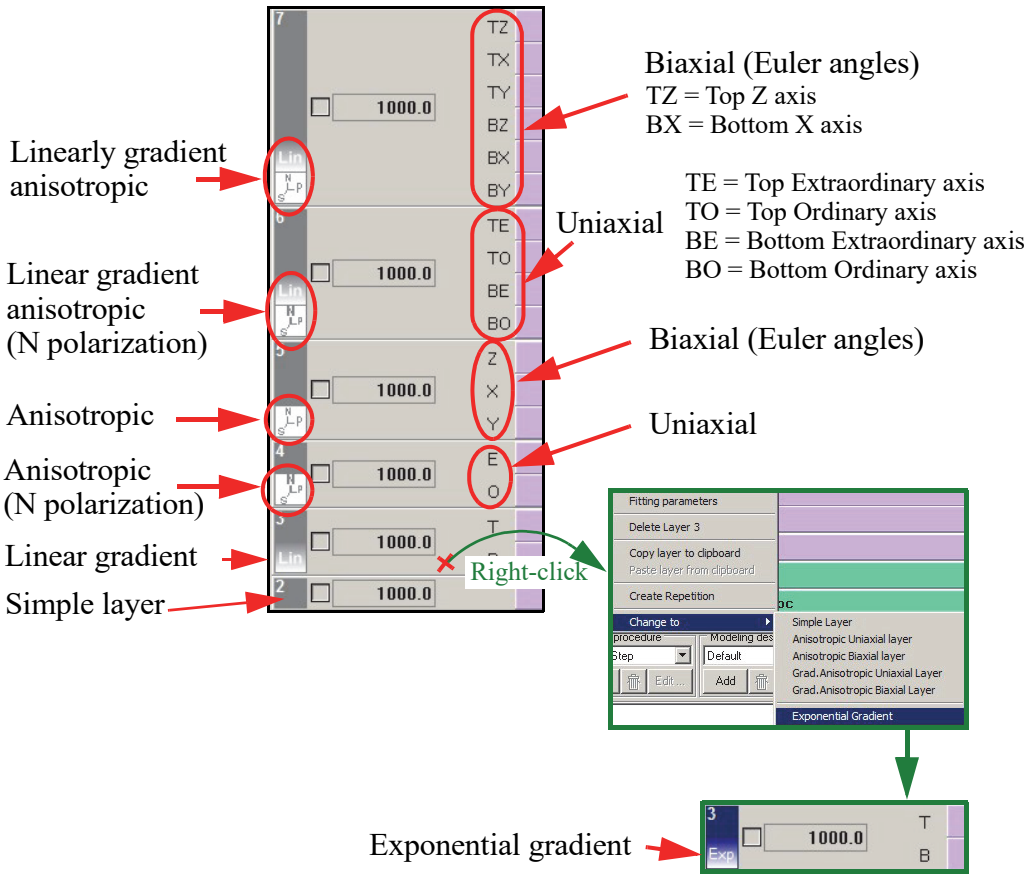
2. Right-click on the area before the layer: the basic layer structure can be changed or a more complex structure can be chosen: exponential gradient or P, S polarization. N polarization is default.

Each of the choices modifies the two information areas located near to the layer.




05/20/2016

The information areas are listed below:




Each time a specific parameter is chosen, the Fitting parameter menu adds the new variables related to this new parameter.

**3** Toggle button which activates (***F*** letter) the Fitting process on the selected layer.

**4** To copy or move a material from a layer to another, proceeds as follow:

**To copy**: Click on the square of the material (see figure 13-3, page 170) and drag to the layer.

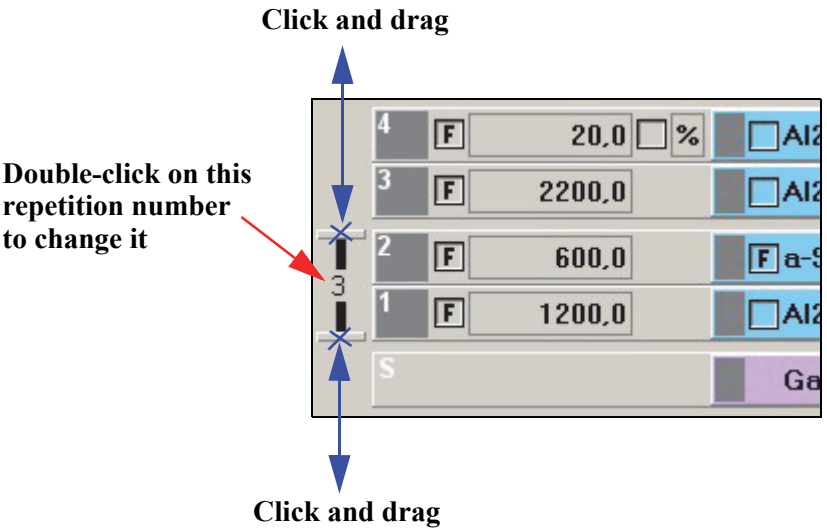
**To move**: Press the Ctrl key and click on the square of the material, then drag to the layer.

05/20/2016

5 Layer(s) repetition

Some structures require one or several layers repetition. Using DeltaPsi2 this function can be applied fast and easy.

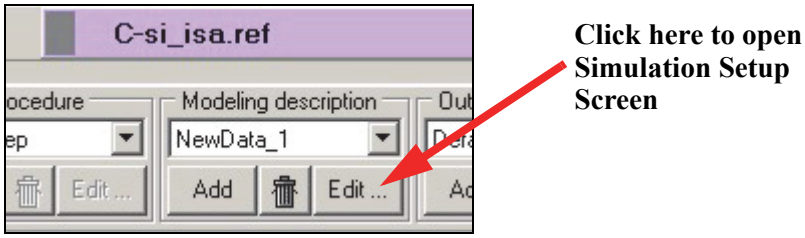
Right-click on the layer number and select ***Create repetition***. The default double repetition is immediately created. Then, using the mouse click and drag, it becomes easy to extend the repeated layer to other layers. To set the number of repetition, double click on the already selected number (default is 2) and enter the new number.




## 13.4 Simulation

Simulation is useful to verify spectroscopic/optical characteristics of a built structure. The user can verify if a layer thickness has an effect.

- Before performing a Simulation, parameters for variations must be set. Click on the ***Edit..*** button located in the ***Modeling description*** screen section (see figure below).




05/20/2016

## Step 1: *Modeling conditions* screen

The ***Modeling Conditions*** tab is useful to set the spectral range used by the fitting/simulation process.

- **Spectral Range** 1

By default, the used Spectral Range will be those defined in the experimental file; it will also use the same acquisition points.

**Experimental file limited spectral range**

Select the spectral range unit, then enter the new limits of the experimental file. The original acquisition data will be conserved.

**User defined spectral range**

Select the new spectral range limits and set the increment. That means the original acquisition data will be replaced by new calculated data (by interpolation).

**Display specific observables after fit or simulation** 2

Specific observables can be displayed on the curve after a fit or a simulation. Check-in the feature then select the observables to display.

- **Measurement Mode** 3

The Measurement Mode choices are directly related to the acquired parameters and then the used Ellipsometers. Take care to always select the right mode according to your Ellipsometer:

***Ellipsometric*** = for UVISEL
***Polarimetric*** = for MM-16, Auto SE
***Merge*** = in case of Merge acquisition

**Ellipsometry configuration**

The Modulator and Analyzer angles can be retrieved from the Experimental configuration file (default) or set manually. Check-in the desired choice.

- **Spectral Data** 4

Each Data Acquisition file can contain several Multispectra files. In fact, the file structure in DP2 includes the Multispectra capabilities. These Multispectra files could be created or generated from a standard acquisition, a kinetic acquisition, a V.A.S.E. (**V**ariable **A**ngle **S**pectroscopic **E**llipsometry) acquisition or an acquisition scenario.

In this section, the Multispectra Index means the Index number located in the acquisition file if this file includes several Multispectra acquisitions. By default Index 1 is selected. The Spectrum Index is the index of a spectrum inside a Multispectra acquisition. See detailed explanation “How to assign an index number” on page 212.

By definition, the Angle of incidence V.A.S.E. and Theta (Θ) V.A.S.E. are Multispectra files: the choice is only the selection of the Multispectra V.A.S.E. acquisition index. When on these V.A.S.E. Multispectra are selected, V.A.S.E. fitting will be proceed.

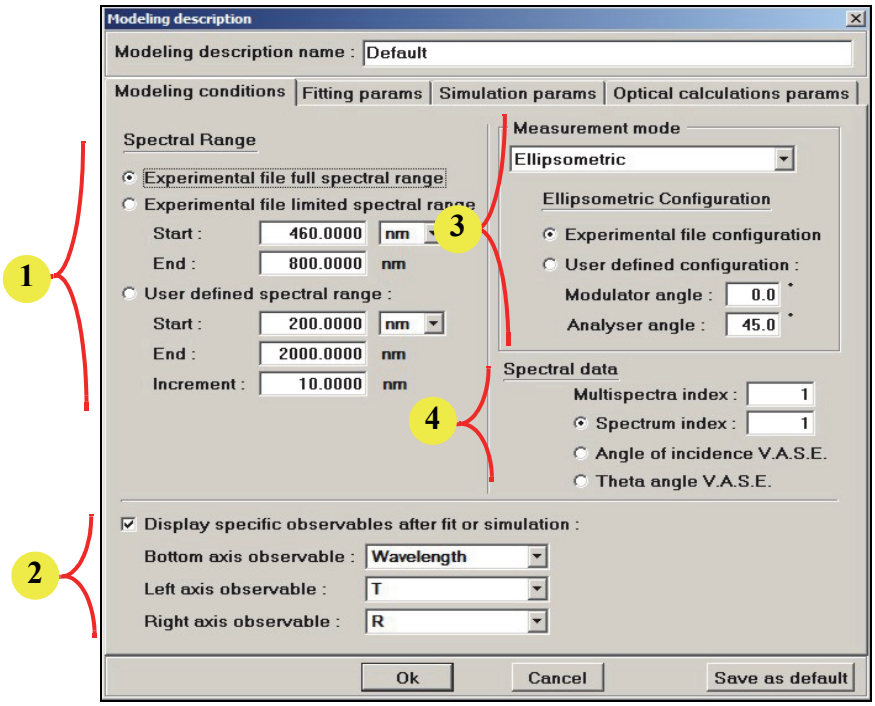



**Figure 13-4 Modeling conditions setup**

## Step 2: *Simulation parameters* screen

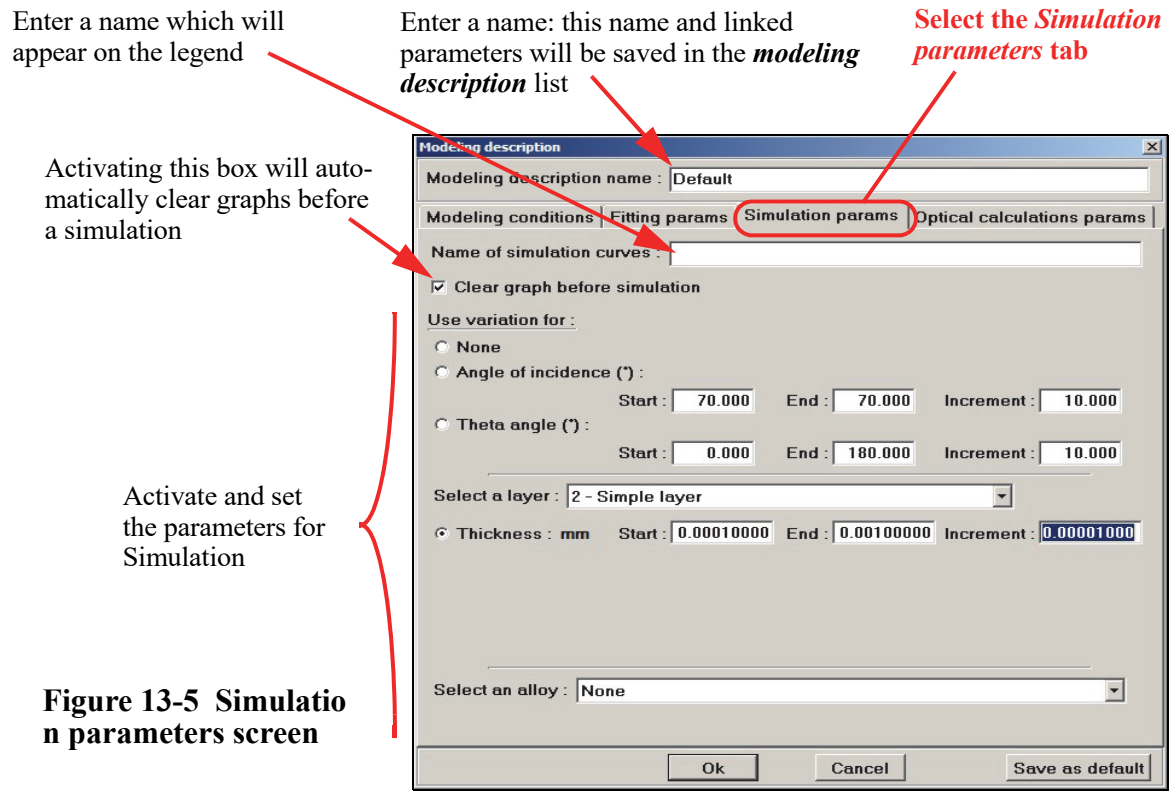
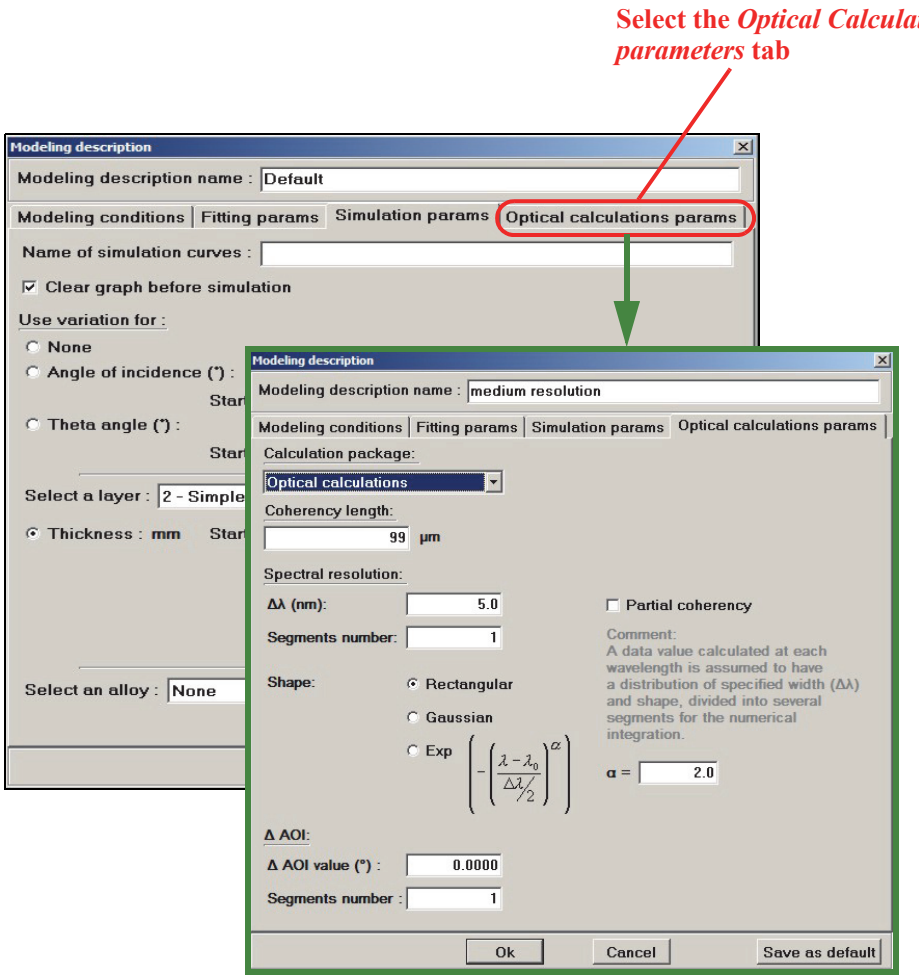



**Figure 13-5 Simulation parameters screen**

- Figure 13-5 shows the parameters which can be varied during a Simulation. The parameters displayed on this screen are related to the current model.
  - ***Simulation curves name***: enter a name related to the current simulation. This name will be displayed on the graph as a legend if the «legend visible» parameter has been checked from the ***Graph Setup*** screen (  icon, see "Graph Setup" on page 35).
  - The variation can be performed on the ***Incidence Angle*** (Modulator/Sample/Analyzer), the ***Thickness*** or ***Volume Fraction*** of a selected layer. The options are added according to the model layers.
  - On the bottom of the screen, the ***Save as default*** button saves the parameters under the ***Default*** description name. Clicking ***OK*** button validates the parameters screen and saves them under the entered ***Modeling description name***.

## Step 3: Optical Calculations Parameters




The parameters described in this screen are used to improve precision in the direction of real curves profile using calculation.

As a rule, the «Calculation package» should always be set to «Optical calculations».

- **Coherency Length and Spectral Resolution**

**What is it?**

**Coherency Length**: The coherence length, usually applied to light waves, gives the distance over which a wave can interfere with itself. For example, imagine a source of light being split into two beams. The beams are then taken on different paths and later recombined. If the difference between the path lengths is less than the coherence length then the beams will interfere with each other and an interference pattern can be observed. If the difference is greater than the coherence length then no interference will be observed.

From this it can be seen that the concept of coherence length is extremely important in holography as it is the process of splitting and recombining a beam to produce and interference pattern that makes a hologram. This is why laser light is usually used in holography, as lasers have extremely long coherence lengths, in the order of a few metres is possible. Compare this with the few centimeters possible with a sodium lamp.

The coherence length depends on the purity of the light source. A laser emitting at a single wavelength λ, will have a very narrow spread in wavelengths, Δλ. Traditional light

sources, such as sodium lamps, whilst only emitting "one" frequency will have a larger spread in wavelength, Δλ, than the laser.

Formally, coherence length is defined as

$$L = \lambda^2 / n\Delta\lambda$$

where λ and Δλ are defined as above and n is the refractive index of the medium in which the wave is travelling.

Keep the 99µm value shown on the figure above.

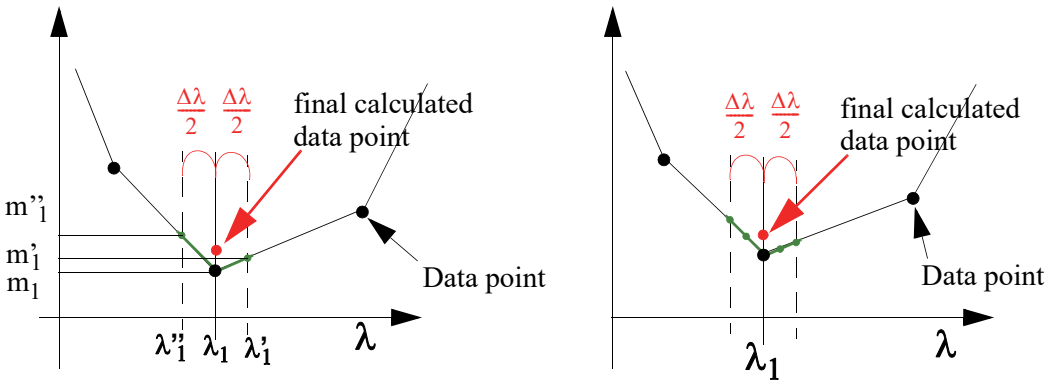
**Spectral Resolution**: The spectral resolution of an optic instrument is a measure of its power to resolve features in the electromagnetic spectrum. It is usually defined by:

$$R = \frac{\lambda}{\Delta\lambda}$$

where Δλ is the smallest difference in wavelengths that can be distinguished, at a wavelength of λ.

The spectral resolution is an instrumental specification. To follow as closer as possible the real situation, two parameters have been added:

- **Setting Δλ**: enter the Monochromator resolution for the working wavelengths range. For UVISEL family keep the displayed 5nm.
- **Segments number**: If equal to 1, the calculation around each data point will not be activated. By entering several segments, each data point will be integrated over ±Δλ/2, using the chosen segments value. Figure below shows how the calculation is performed. The next paragraph will show real DP2 results.




Segment value=2 Segment value=4

**DeltaPsi2 practical examples**: DP2 manages very simply the calculations through the sample. In order to clarify the use of this feature, we can determine the following several cases:

- **Example 1: Effect of spectral resolution**

A sample is composed of one $SiO_x$ layers of 4µm on a silicon substrate. We have set a wavelength range of 450-830 nm with a 1nm increment. Generally, if a layer thickness is

within 0-5µm, the segment value of 10 is a good choice. You can try several values to reach the best $\chi 2$. Figure 1 shows the simulation result of such a Model.

**Example**: this figure shows two simulations: lower graphs have been processed without integration (Segment number=1), upper graphs with an integration and segments number of 10. Thus upper graphs better reflect the reality.

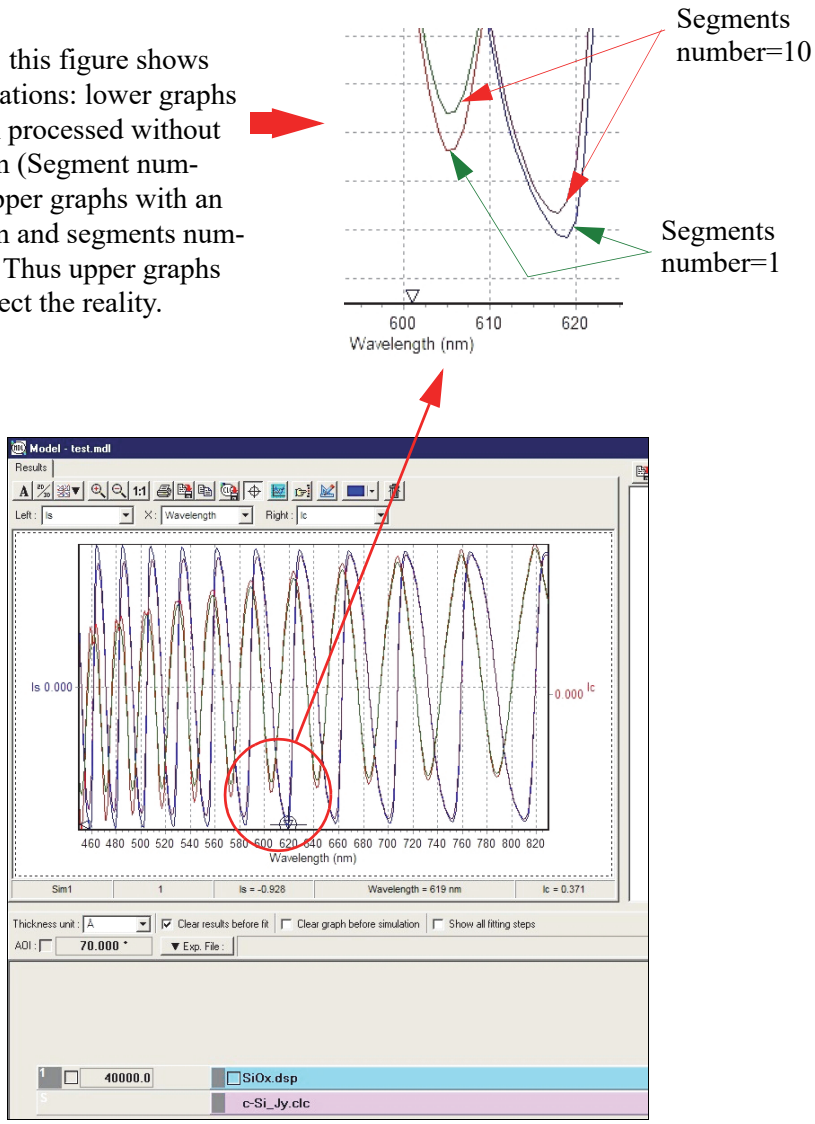



**Figure 13-6 Spectral Resolution effect**

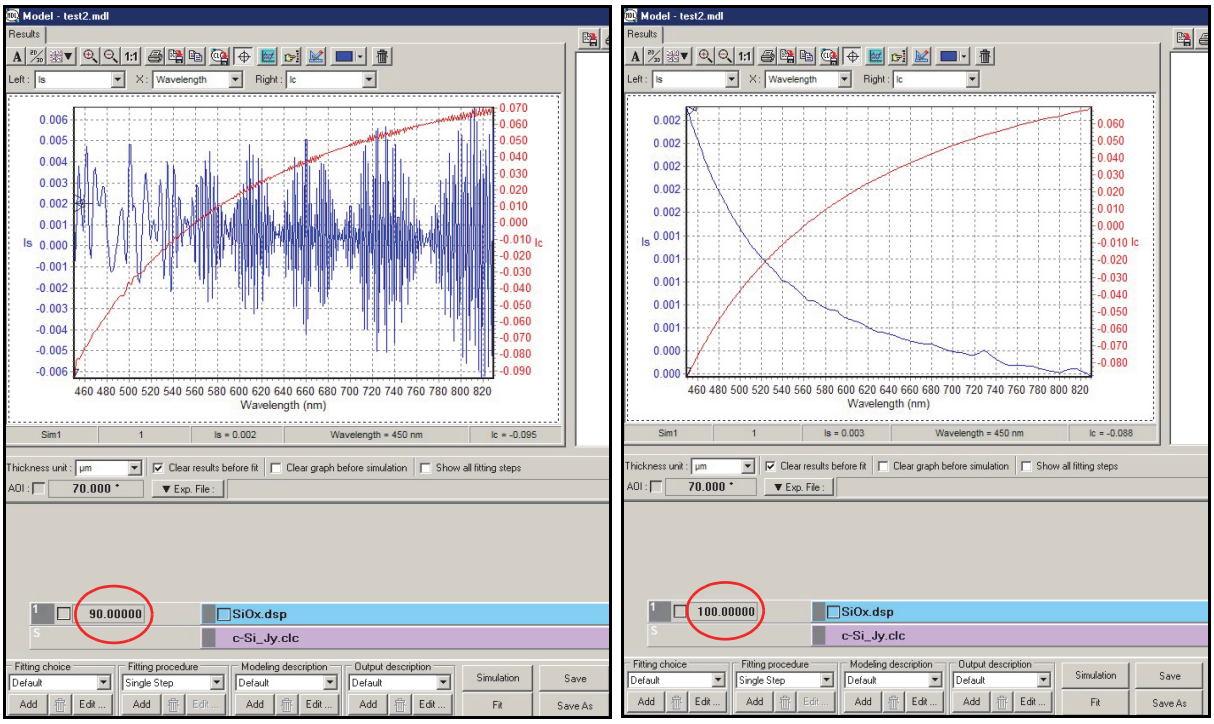
**Setting the *Shape*:** We have previously seen that each data point will be integrated on a $\pm\Delta\lambda/2$ on several segments. The integrated results can be then smoothed using specific shapes: ***Rectangle*** means that the smoothed action will not be effective between the integrated calculated points on $\pm\Delta\lambda/2$. If ***Gaussian*** shape is chosen, a Gaussian smoothed profile will be activated. The user have also the choice to use the displayed formula with the $\alpha$ variable. If $\alpha$ is equal to 2, the shape is Gaussian. By lowering the $\alpha$ value, the smoothing will be more effective.

- **Example 2: Effect of Coherency length**

In the case of thick layers, the interferences over the spectral range of the instrument are too numerous and then too close to be resolved by the monochromator limited resolution. Thus these layers must be treated differently in the calculations to be compatible with what is measured by the instrument. Usually these kind of layers above 100 µ (apart from polymer sheets) are substrates (silicon, glass...) and are not thin films anymore with thickness to be fitted. This is why coherency length value has been set to 99µm. If a layer is found to

be 80µm, a good idea will be to change the coherency value to 70µm and verify the obtained result.

In this example, the Spectral Resolution parameters will be set as shown on the Example 1 with a Segments number equal to 1.




**90µm $SiO_x$ layer** **100µm $SiO_x$ layer**

**Figure 13-7 Example2: Effect of coherency length**

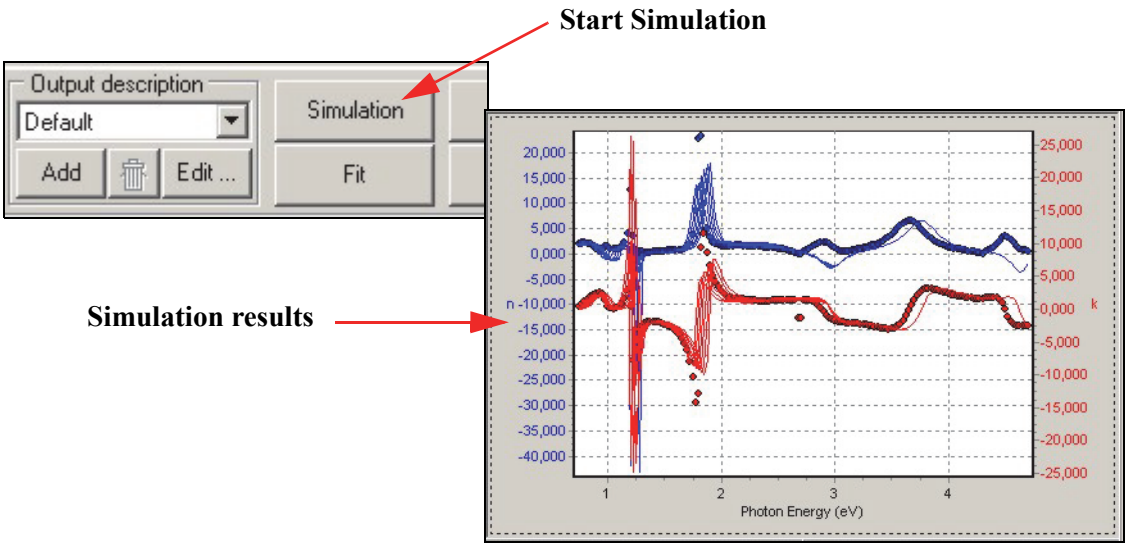
- **Δ AOI Settings**

  As for the wavelengths, the integration can be performed on each V.A.S.E. data point on a Δ AOI using several ***Segments***.

05/20/2016

## Launching Simulation

- To launch the Simulation, validate the parameters screen then click on the *Simulation* button. A fraction of second later, graphics area displays the resulted Simulation graphs.



# 13.5 Fitting

**What is the fitting process?**

The fitting process is a process which adjusts a theoretic sample (previously built model) to an experimental data sample (real measure). The user can define the setup of the parameters which must be taken into account by the process. The data fitting is performed using the iterative fitting algorithm. It is designed to minimize the value of the $\chi 2$-residue parameter by adjusting the sample fitting parameters. $\chi 2$ Parameter is used to describe how close the generated data matches the experimental ones.

When the **Model** structure is displayed, the user can access to various options that control and perform the data fitting process. To start the fitting process the following items are assumed to exist:

- A set of selected experimental data, which is to be fit;
- A model, which is used to generate data for comparison with the experimental data;
- A set of fit parameters, which will be varied by the fitting algorithm to minimize the difference between experimental and generated data.

**Clear Results before fit**: checking this option, located in the main Model screen, will erase the previous fitting result text. If not checked, each fitting result text will be added.

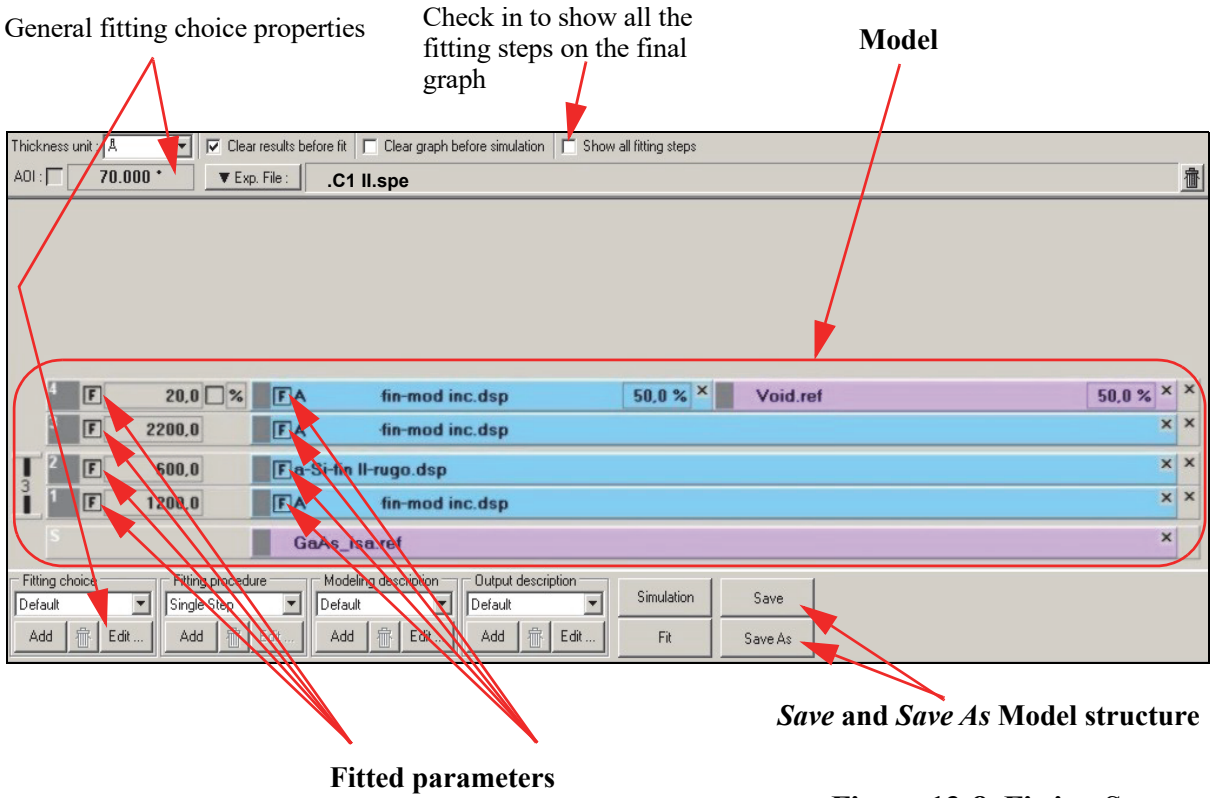


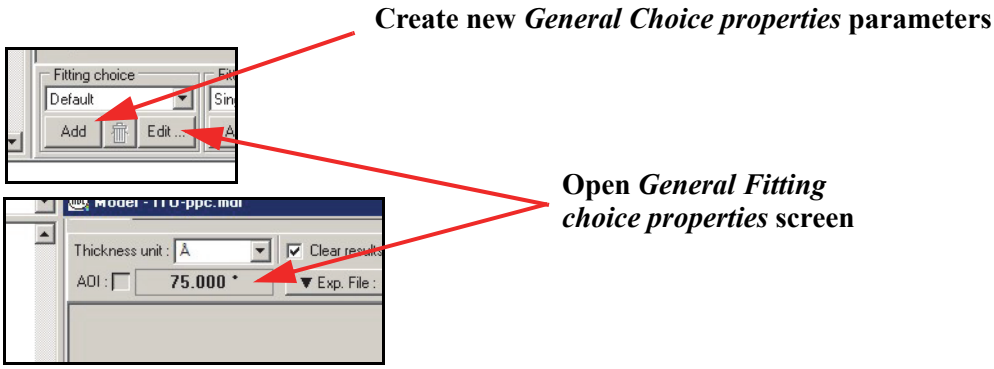

**Figure 13-8 Fitting Setup**

## 13.5.1 General Fitting Choice properties Setup

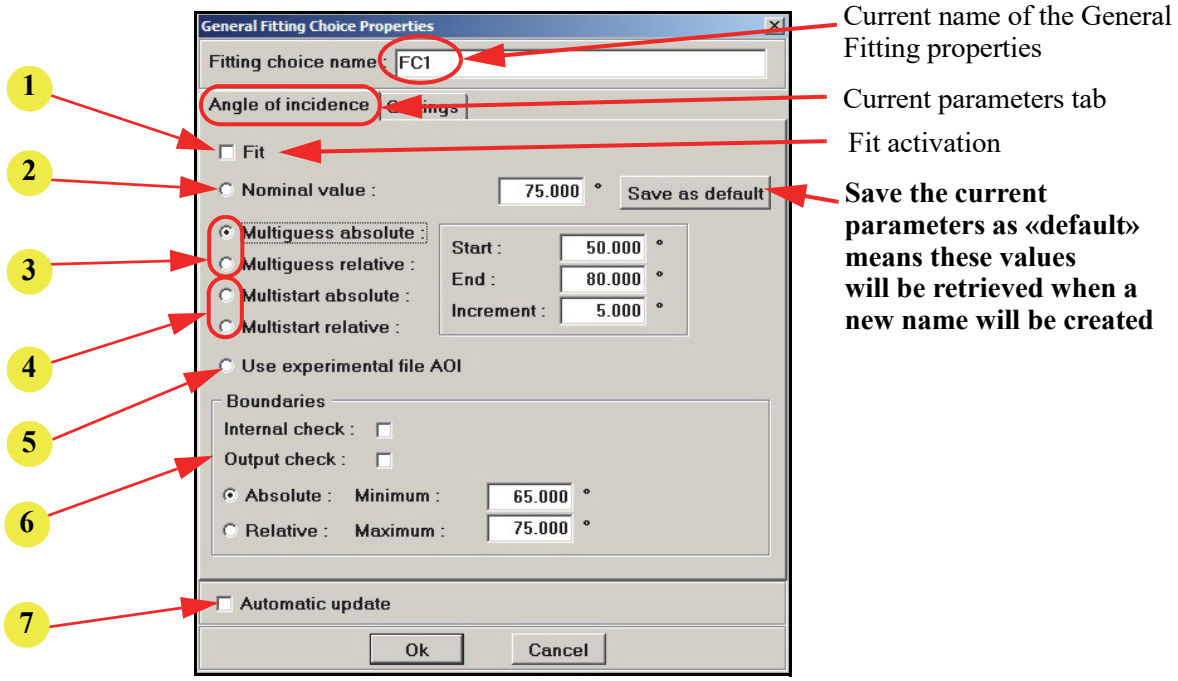
The first parameters to set concern the General Fitting Choice Properties. Please follow the following procedure:

- Create a new name by clicking on the *Add* button located on *Fitting choice* section of the Model screen.

- Open the General Fitting Choice properties screen by clicking from the following locations:



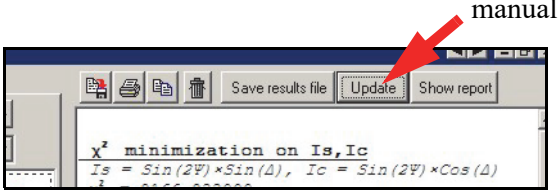

- The following is then displayed:




- **Angle of Incidence (AOI) fitting setup:**

| | |
|---|---|
| **Fit** 1 | Check the box to activate the fitting process on the current parameter. |

| | |
|---|---|
| **Nominal Value**<br>2 | Enter or select the nominal value of the current AOI (Angle of Incidence) parameter. |
| **Multiguess**<br>3 | The ***Multiguess*** choices allow the user to perform a fitting process with different initial/end values of AOI (angle of incidence). These initial/end values (and related increment) will be used as a plain algorithmic values to find the right AOI. The retained fitted parameter is directly related to the better $\chi^2$.<br>Choose the guess range with care as the Fitting process will scan all of the possibilities and it could drastically increase the process time. |
| **Multistart**<br>4 | When a fit is launched with the ***Multistart*** option, the process will adjust all of the checked-for-fit parameters for every defined and fixed ***Multistart*** values. The retained value is directly related to the better $\chi^2$.<br>In other words, all of the checked-for-fit values will be fitted for a fixed value of ***Multistart*** (this value will not be fitted).<br>The retained thickness is directly related to the better $\chi^2$. |
| **Use experimental file AOI**<br>5 | Use this option to retrieve the AOI from the experimental file. |
| **Boundaries**<br>6 | **Internal check**: the internal check limits are directly imported to the Marquart formula to perform the optimization,<br>**Output check**: the Output check will remove the results which exceed the limits after the optimization.<br>Enter the limits if one of the check has been chosen |

| | |
|---|---|
| **Automatic Update:** 7 | If checked before the fitting process, the values of the Model will be updated. To prevent any loss of the entered values and if the sample has not been previously saved, a screen will ask for a new sample and model file name.<br>The same operation can be performed manually by clicking on the ***Update*** button which is displayed above the results screen after a fitting process. See below the location of the manual ***Update***.<br>manual update<br>  |

- **Grating properties setup**

  This feature is used to characterized grating samples. Thus, it becomes possible to determine grating shapes, profile models, microelectronics (process control), metrological characterization of samples and overlay simulations.

  The following figure shows the general diagram for characterization of gratings:

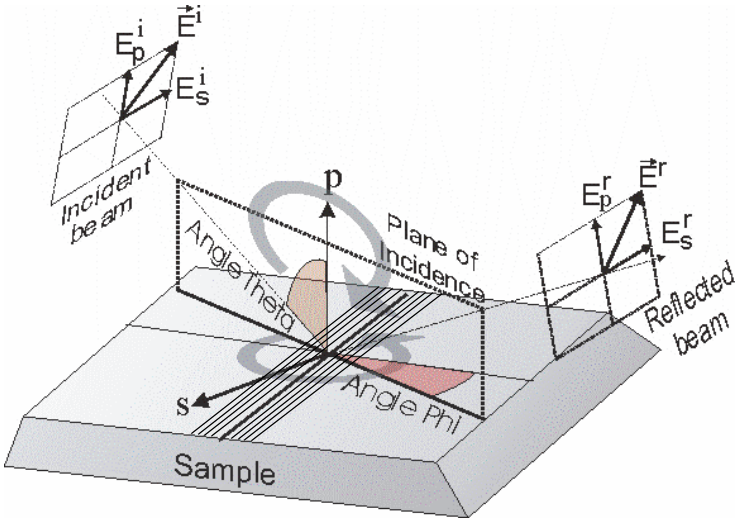
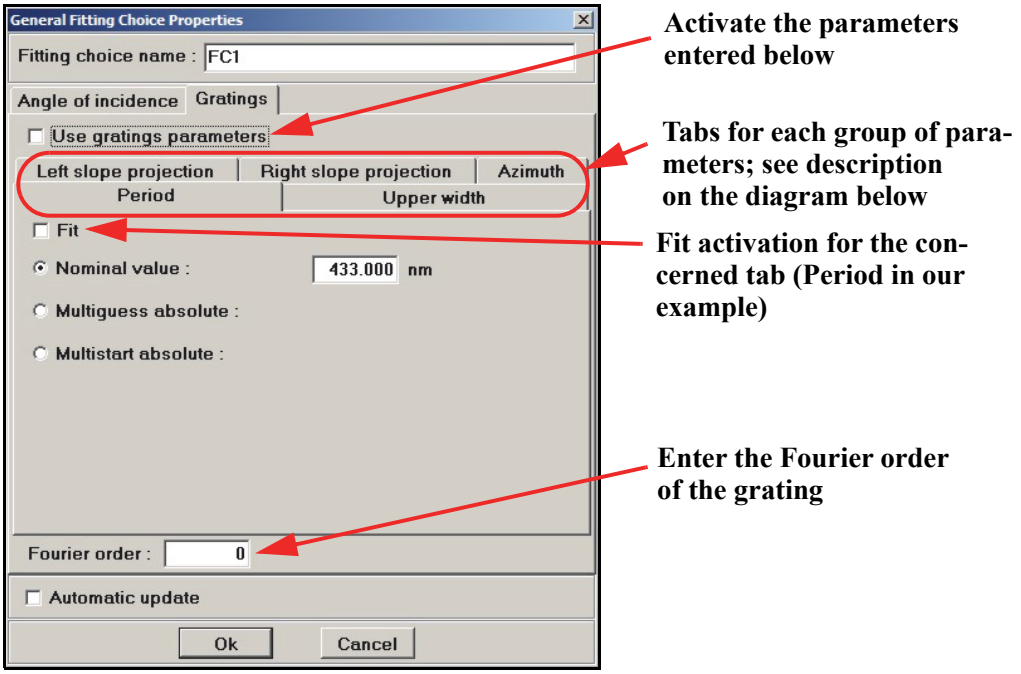



**Figure 13-9 Grating characterization diagram**

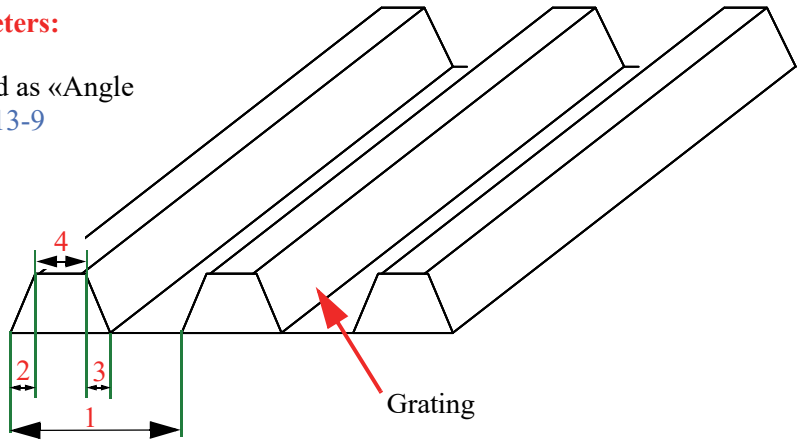
The Grating tab of the ***General Fitting Choice properties*** shows the parameters to enter for a fitting process.




**Grating parameters:**

Azimuth is the angle defined as «Angle Phi» on the diagram figure 13-9

1 = Period
2 = Left slope projection
3 = Right slope projection
4 = Upper width




### 13.5.2 Specific Fitting Choice properties Setup

Specific means that properties for fitting can be setup for a specific component which can be adapted. This component can be a layer or a slot of a layer (if a layer is a material gradient or with an anisotropic profile), the substrate layer, a material of a layer.

---

**Parameters Setting**

Figure 13-8 shows a general Model structure which is ready to be fitted. From this screen, all of the parameters can be easily setup. These parameters can be accessed with a click on the elements of this screen.

The square boxes on the screen show the elements which can be fitted. The table below lists all of the existing screens and how to display them.

| How to ... | Accessing from |
|---|---|
| Setup general fitting conditions (see figure 13-8) | Click on the ***Edit..*** button located on the bottom part of the main screen (***Fitting choice*** section) |
| Fit on a layer thickness | Click on the layer value |
| Adjust fitting parameters for each layer (excepting .ref files) | Right-click on the layer and select «***fitting parameters***» |
| Adjust dispersion fitting parameters (if available) | Right-click on the ***Material*** to open the ***fitting parameters*** and/or ***Dispersion parameters*** screen<br>The parameters are also opened if square box is in the activation phase. |
| Modify dispersion parameters (if available) | Click on the material name |
| Fit on the material volume fraction | Click on the **%** sign |

*05/20/2016*

# 13.6 General fitting conditions: Modeling Description

- Before performing a Fitting, general parameters for variations must be set. Click on the ***Edit..*** button located in the *Modeling Description* screen section (see figure below)


The following steps should be completed:

## Step 1: *Modeling conditions* screen

The ***Modeling Conditions*** tab is useful to set the spectral range used by the fitting process.

- **Spectral Range** 1

By default, the used Spectral Range will be those defined in the experimental file; it will also use the same acquisition points.

**Experimental file limited spectral range**

Select the spectral range unit, then enter the new limits of the experimental file. The original acquisition data will be conserved.

**User defined spectral range**

Select the new spectral range limits and set the increment. That means the original acquisition data will be replaced by new calculated data (by interpolation).

**Display specific observables after fit or simulation** 2

Specific observables can be displayed on the curve after a fit or a simulation. Check-in the feature then select the observables to display.

- **Measurement Mode** 3

The Measurement Mode choices are directly related to the acquired parameters and then the used Ellipsometers. Take care to always select the right mode according to your Ellipsometer:

***Ellipsometric*** = for UVISEL
***Polarimetric*** = for MM-16, Auto SE
***Merge*** = in case of Merge acquisition

**Ellipsometry configuration**

The Modulator and Analyzer angles can be retrieved from the Experimental configuration file (default) or set manually. Check-in the desired choice.

- **Spectral Data** 4

Each Data Acquisition file can contain several Multispectra files. In fact, the file structure in DP2 includes the Multispectra capabilities. These Multispectra files could be created or generated from a standard acquisition, a kinetic acquisition, a V.A.S.E. (**V**ariable **A**ngle **S**pectroscopic **E**llipsometry) acquisition or an acquisition scenario.

In this section, the Multispectra Index means the Index number located in the acquisition file if this file includes several Multispectra acquisitions. By default Index 1 is selected. The Spectrum Index is the index of a spectrum inside a Multispectra acquisition. See detailed explanation "How to assign an index number" on page 212.

By definition, the Angle of incidence V.A.S.E. and Theta (Θ) V.A.S.E. are Multispectra files: the choice is only the selection of the Multispectra V.A.S.E. acquisition index. When on these V.A.S.E. Multispectra are selected, V.A.S.E. fitting will be proceed.

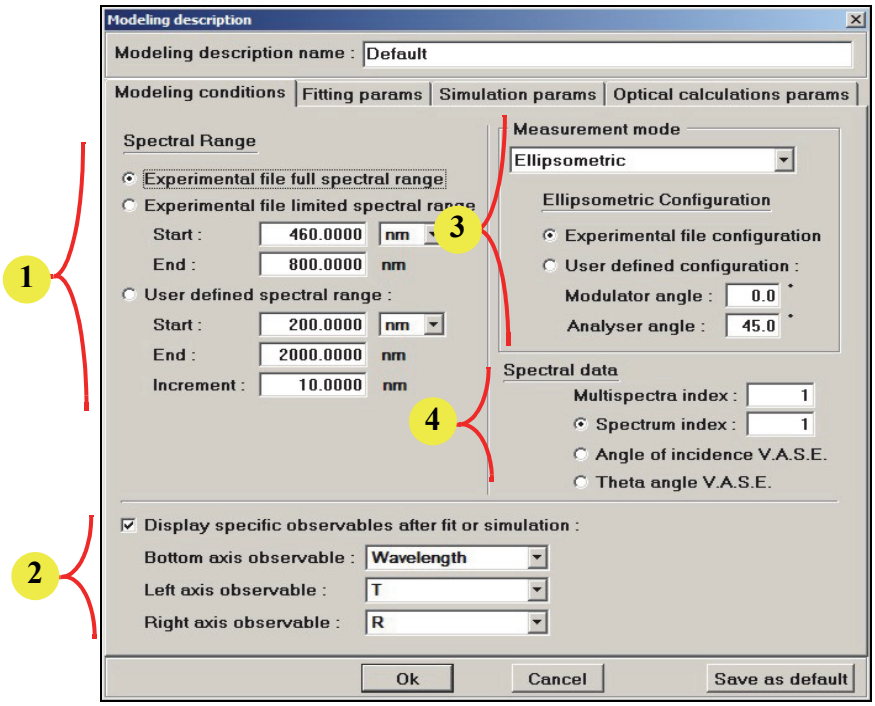



**Figure 13-10 Modeling conditions setup**

## Step 2: *Simulation parameters* screen

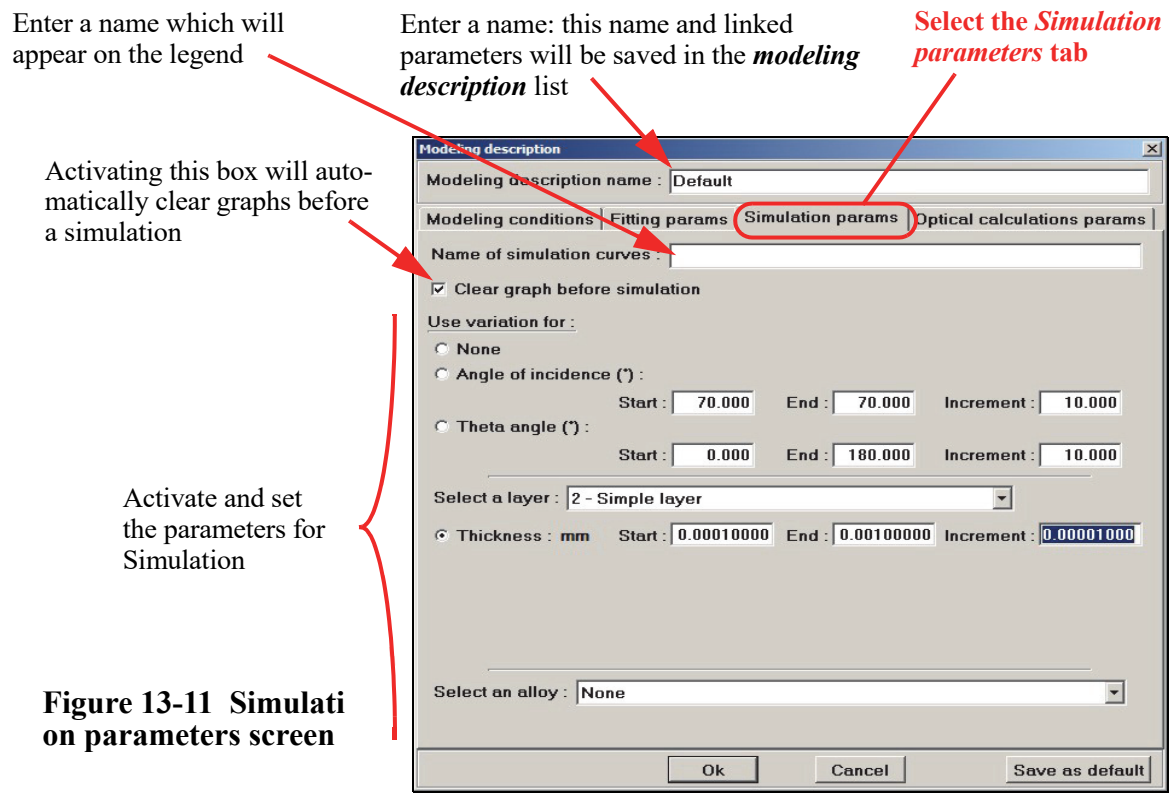
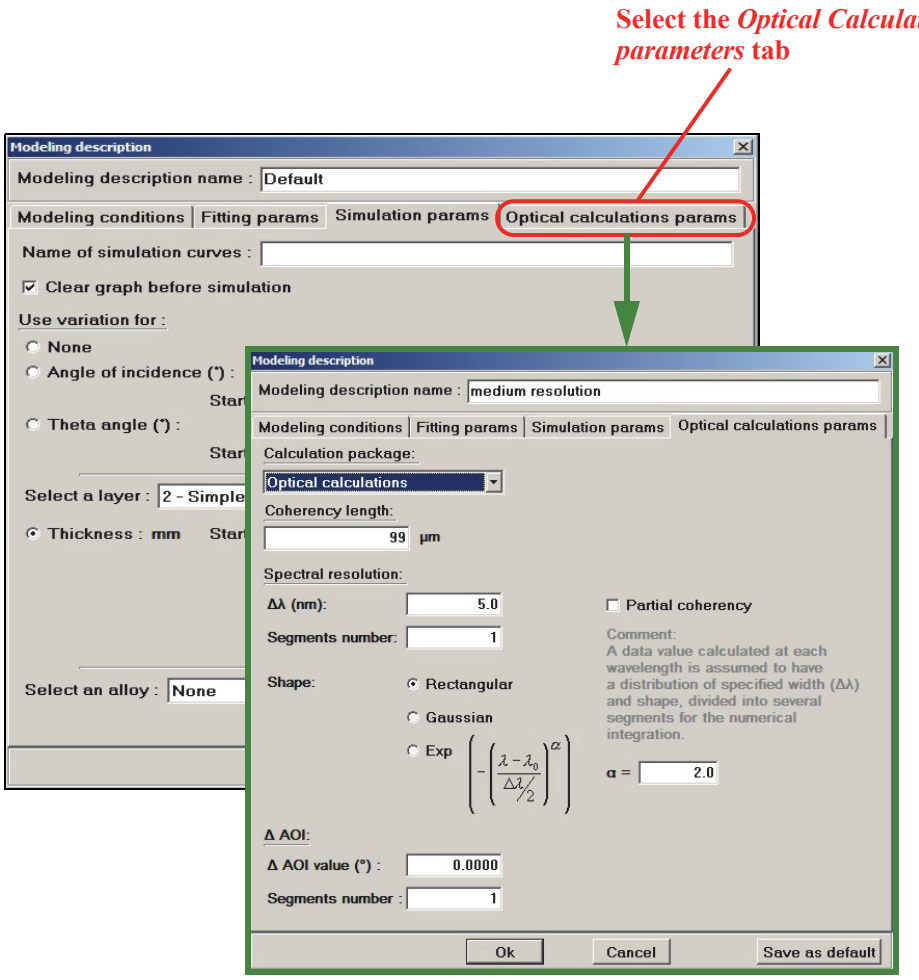



**Figure 13-11 Simulation parameters screen**

- Figure 13-5 shows the parameters which can be varied during a Simulation. The parameters displayed on this screen are related to the current model.
  - ***Simulation curves name***: enter a name related to the current simulation. This name will be displayed on the graph as a legend if the «legend visible» parameter has been checked from the ***Graph Setup*** screen (  icon, see "Graph Setup" on page 35).
  - The variation can be performed on the ***Incidence Angle*** (Modulator/Sample/Analyzer), the ***Thickness*** or ***Volume Fraction*** of a selected layer. The options are added according to the model layers.
  - On the bottom of the screen, the ***Save as default*** button saves the parameters under the ***Default*** description name. Clicking ***OK*** button validates the parameters screen and saves them under the entered ***Modeling description name***.

## Step 3: Optical Calculations Parameters




The parameters described in this screen are used to improve precision in the direction of real curves profile using calculation.

As a rule, the «Calculation package» should always be set to «Optical calculations».

- **Coherency Length and Spectral Resolution**

**What is it?**

**Coherency Length**: The coherence length, usually applied to light waves, gives the distance over which a wave can interfere with itself. For example, imagine a source of light being split into two beams. The beams are then taken on different paths and later recombined. If the difference between the path lengths is less than the coherence length then the beams will interfere with each other and an interference pattern can be observed. If the difference is greater than the coherence length then no interference will be observed.

From this it can be seen that the concept of coherence length is extremely important in holography as it is the process of splitting and recombining a beam to produce and interference pattern that makes a hologram. This is why laser light is usually used in holography, as lasers have extremely long coherence lengths, in the order of a few metres is possible. Compare this with the few centimeters possible with a sodium lamp.

The coherence length depends on the purity of the light source. A laser emitting at a single wavelength λ, will have a very narrow spread in wavelengths, Δλ. Traditional light sources, such as sodium lamps, whilst only emitting "one" frequency will have a larger spread in wavelength, Δλ, than the laser.

Formally, coherence length is defined as

$$L = \lambda^2 / n\Delta\lambda$$

where λ and Δλ are defined as above and n is the refractive index of the medium in which the wave is travelling.

Keep the 99µm value shown on the figure above.

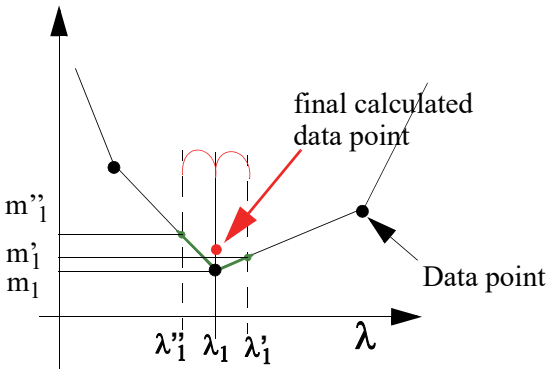
**Spectral Resolution**: The spectral resolution of an optic instrument is a measure of its power to resolve features in the electromagnetic spectrum. It is usually defined by:

$$R = \frac{\lambda}{\Delta\lambda}$$

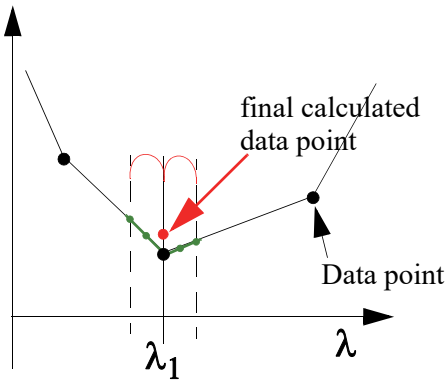
where Δλ is the smallest difference in wavelengths that can be distinguished, at a wavelength of λ.

The spectral resolution is an instrumental specification. To follow as closer as possible the real situation, two parameters have been added:

- **Setting Δλ**: enter the Monochromator resolution for the working wavelengths range. For UVISEL family keep the displayed 5nm.
- **Segments number**: If equal to 1, the calculation around each data point will not be activated. By entering several segments, each data point will be integrated over ±Δλ/2, using the chosen segments value. Figure below shows how the calculation is performed. The next paragraph will show real DP2 results.




Segment value=2




Segment value=4

**DeltaPsi2 practical examples**: DP2 manages very simply the calculations through the sample. In order to clarify the use of this feature, we can determine the following several cases:

- **Example 1: Effect of spectral resolution**

A sample is composed of one $SiO_x$ layers of 4µm on a silicon substrate. We have set a wavelength range of 450-830 nm with a 1nm increment. Generally, if a layer thickness is

within 0-5µm, the segment value of 10 is a good choice. You can try several values to reach the best $\chi 2$. Figure 1 shows the simulation result of such a Model.

**Example**: this figure shows two simulations: lower graphs have been processed without integration (Segment number=1), upper graphs with an integration and segments number of 10. Thus upper graphs better reflect the reality.

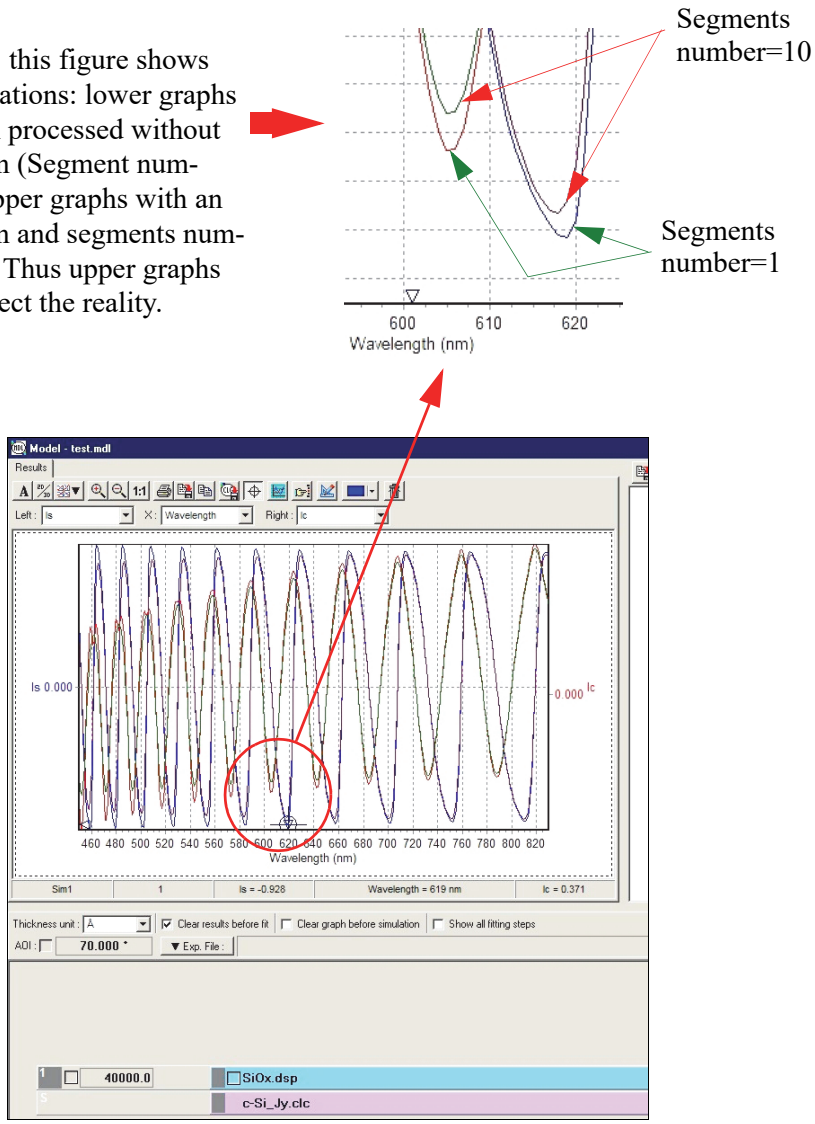



**Figure 13-12 Spectral Resolution effect**

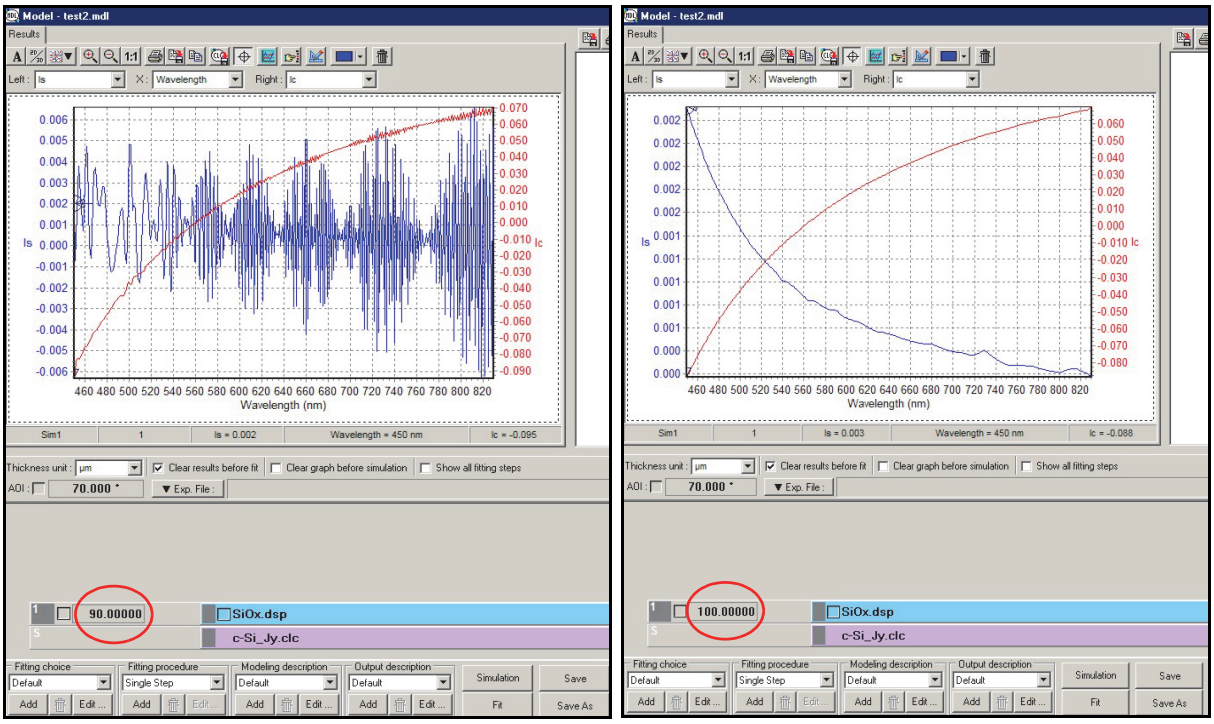
**Setting the *Shape*:** We have previously seen that each data point will be integrated on a $\pm\Delta\lambda/2$ on several segments. The integrated results can be then smoothed using specific shapes: ***Rectangle*** means that the smoothed action will not be effective between the integrated calculated points on $\pm\Delta\lambda/2$. If ***Gaussian*** shape is chosen, a Gaussian smoothed profile will be activated. The user have also the choice to use the displayed formula with the $\alpha$ variable. If $\alpha$ is equal to 2, the shape is Gaussian. By lowering the $\alpha$ value, the smoothing will be more effective.

- **Example 2: Effect of Coherency length**

In the case of thick layers, the interferences over the spectral range of the instrument are too numerous and then too close to be resolved by the monochromator limited resolution. Thus these layers must be treated differently in the calculations to be compatible with what is measured by the instrument. Usually these kind of layers above 100 µ (apart from polymer sheets) are substrates (silicon, glass...) and are not thin films anymore with thickness to be fitted. This is why coherency length value has been set to 99µm. If a layer is found to

be 80µm, a good idea will be to change the coherency value to 70µm and verify the obtained result.

In this example, the Spectral Resolution parameters will be set as shown on the Example 1 with a Segments number equal to 1.




**90µm $SiO_x$ layer** **100µm $SiO_x$ layer**

**Figure 13-13 Example2: Effect of coherency length**

- **Δ AOI Settings**

  As for the wavelengths, the integration can be performed on each V.A.S.E. data point on a Δ AOI using several ***Segments***.

## Step 3: *Fitting parameters* screen

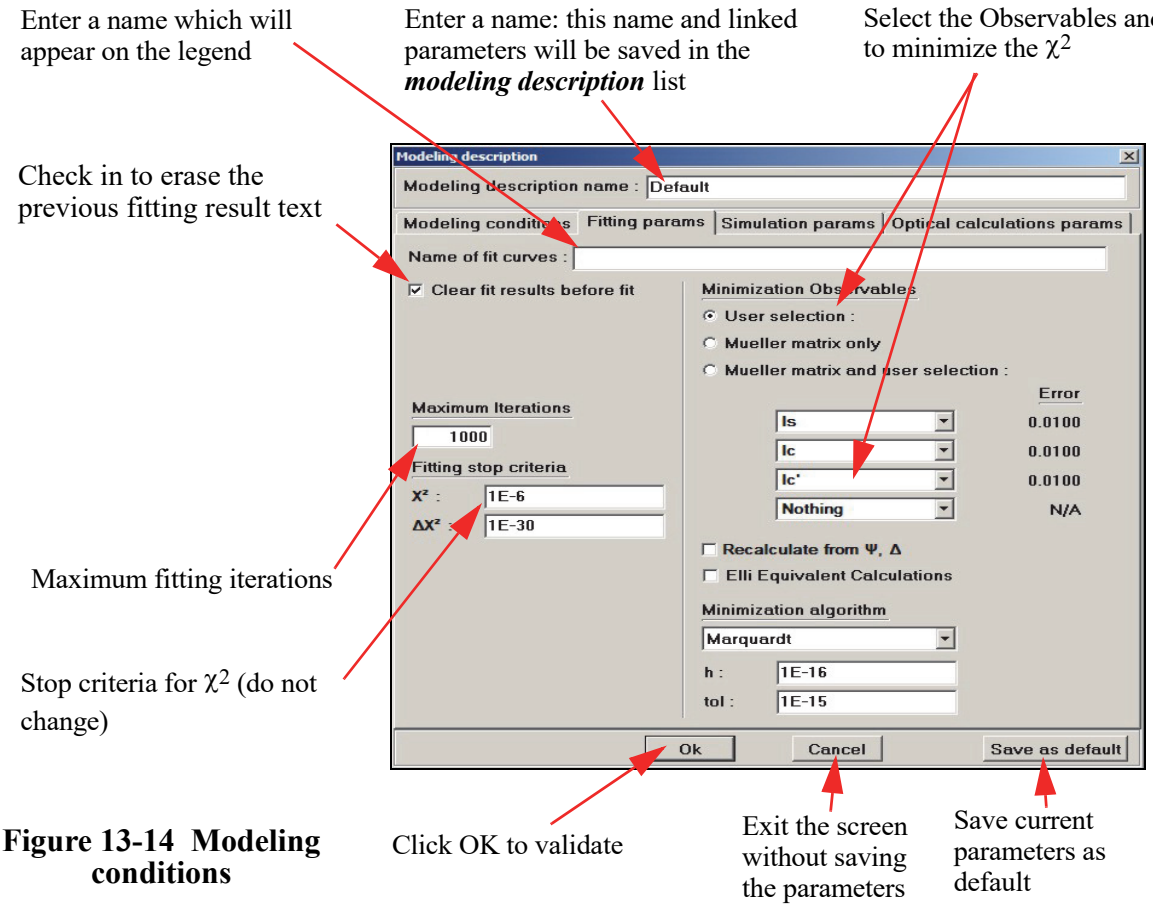



**Figure 13-14 Modeling conditions**

- **Minimization Observables**

  **User selection**: if checked, a 4-observable table is displayed. Select the observables which will be use for the fitting or simulation. Usually Is and Ic are selected for the UVISEL. If the model is standard without anisotropy Is, Ic, Ic' will be a good choice for the MM-16 and Auto SE.

  **Mueller matrix only**: the fitting and simulation processes will only use the Mueller matrix. In this case, the selection table is not displayed. This choice will be good for anisotropic samples. The fitting or simulation will be performed on all the Mueller matrix parameters.

  **Mueller matrix and user selection**: the Mueller matrix and user selected observables will be used for the fitting or simulation. This choice is more marginal and concerns only specific samples measured with the MM-16 or Auto SE Ellipsometer. Up to 4 additional observables can be added with the Mueller matrix.

  **Recalculate from Ψ and Δ**: Select this option

  **Elli Equivalent Calculations**: default is not checked. Only for oldest HORIBA Jobin Yvon Ellipsometers.

  Minimization algorithm: several algorithm can be selected. The most used are Marquart and Simplex. Marquart is the fastest with repetitive results. Simplex often needs more iterations as it generates by itself data points by successive approximations to reach the best $\chi^2$, thus the results are each time different.

**h and Tol**: do not change the default values.

## Step 4: Thickness Fitting Choice Properties

This screen can be accessed from a layer thickness (in this case ***Material*** tab is not displayed).

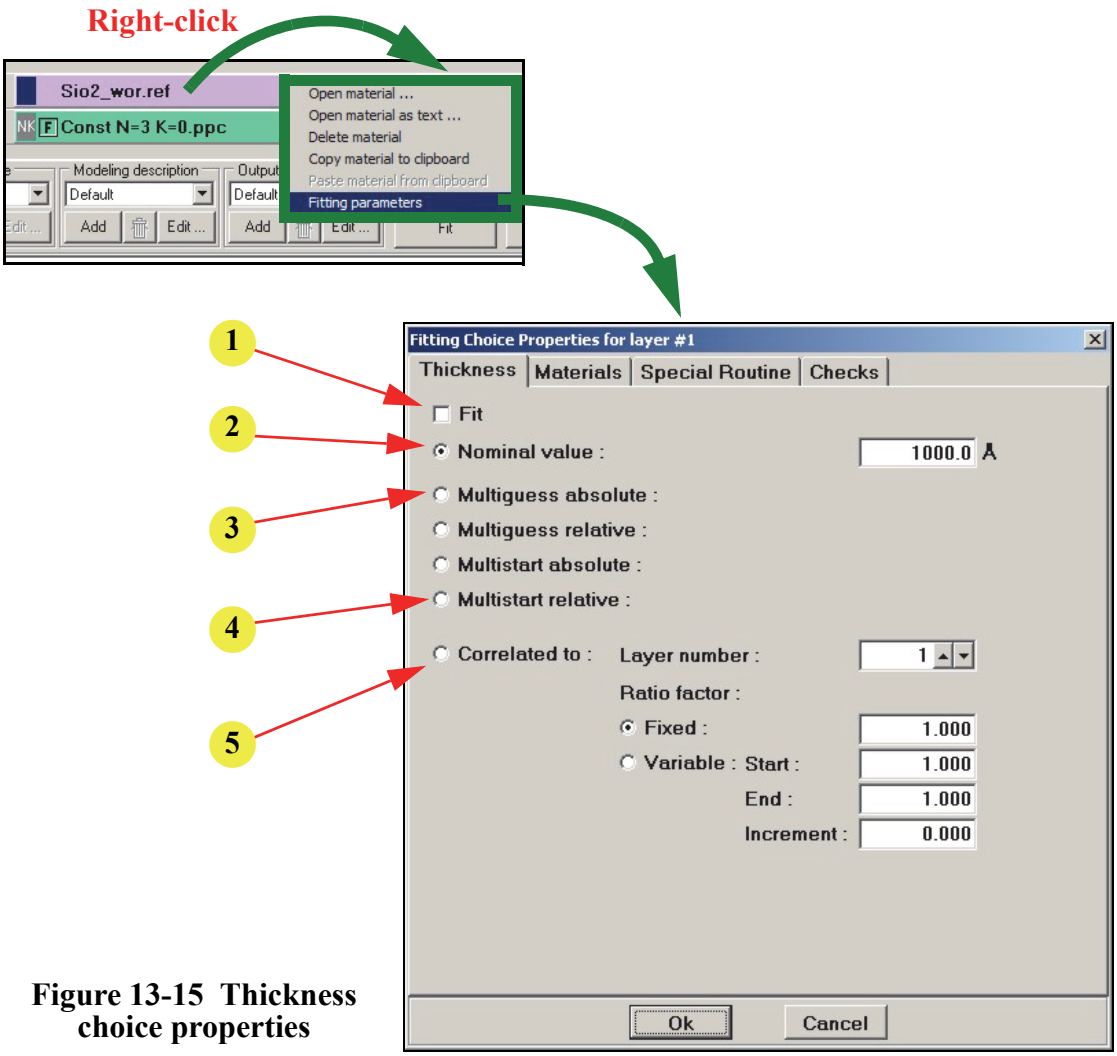



**Figure 13-15 Thickness choice properties**

***Notice***: The labels which define the fitting procedures are the same for several components. The same choices can be found, at different location, for a layer thickness, a material volume, the angle of incidence (AOI) and the dispersion parameters. They are described below:

| | |
|---|---|
| **Fit** 1 | Check the box to activate the fitting process on the current component: layer thickness. |

05/20/2016

| | |
|---|---|
| **Nominal Value**<br>**2** | Enter or select the nominal value of the current parameter (thickness, volume, angle of incidence or dispersion parameter). |
| **Multiguess**<br>**3** | The ***Multiguess*** choices allow the user to perform a fitting process with different initial/end values of thickness, volumes, AOI (angle of incidence) or dispersion parameter. These initial/end values (and related increment) will be used as a plain algorithmic values to find the right thickness, volume, AOI or dispersion parameter. The retained fitted parameter is directly related to the better $\chi^2$.<br>Several layers or slots can be setup for ***Multiguess***: in this case, the fit process will cover all the combinations and the better $\chi^2$ will be selected.<br>Choose the guess range with care as the Fitting process will scan all of the possibilities and it could drastically increase the process time. |
| **Multistart**<br>**4** | When a fit is launched with the ***Multistart*** option, the process will adjust all of the checked-for-fit parameters for every defined and fixed ***Multistart*** values. The retained value is directly related to the better $\chi^2$.<br>In other words, all of the checked-for-fit values will be fitted for a fixed value of ***Multistart*** (this value will not be fitted).<br>The retained thickness is directly related to the better $\chi^2$.<br>Several layers or slots can be setup for ***Multistart***: in this case, the fit process will cover all the combinations and the better $\chi^2$ will be selected. |
| **Correlated**<br>**5** | The ***Correlated*** option allows the current layer to be correlated to another selected layer defined by its number. A ratio can be apply through this correlation. This ratio can be constant or variable. When the fit is launched, the retained values of the checked-for-fit parameters are those which will reach the better $\chi^2$. |

## Step 5: Fitting Materials Choice Properties

Right-click on a layer material name to display the following Fitting Choice Properties:

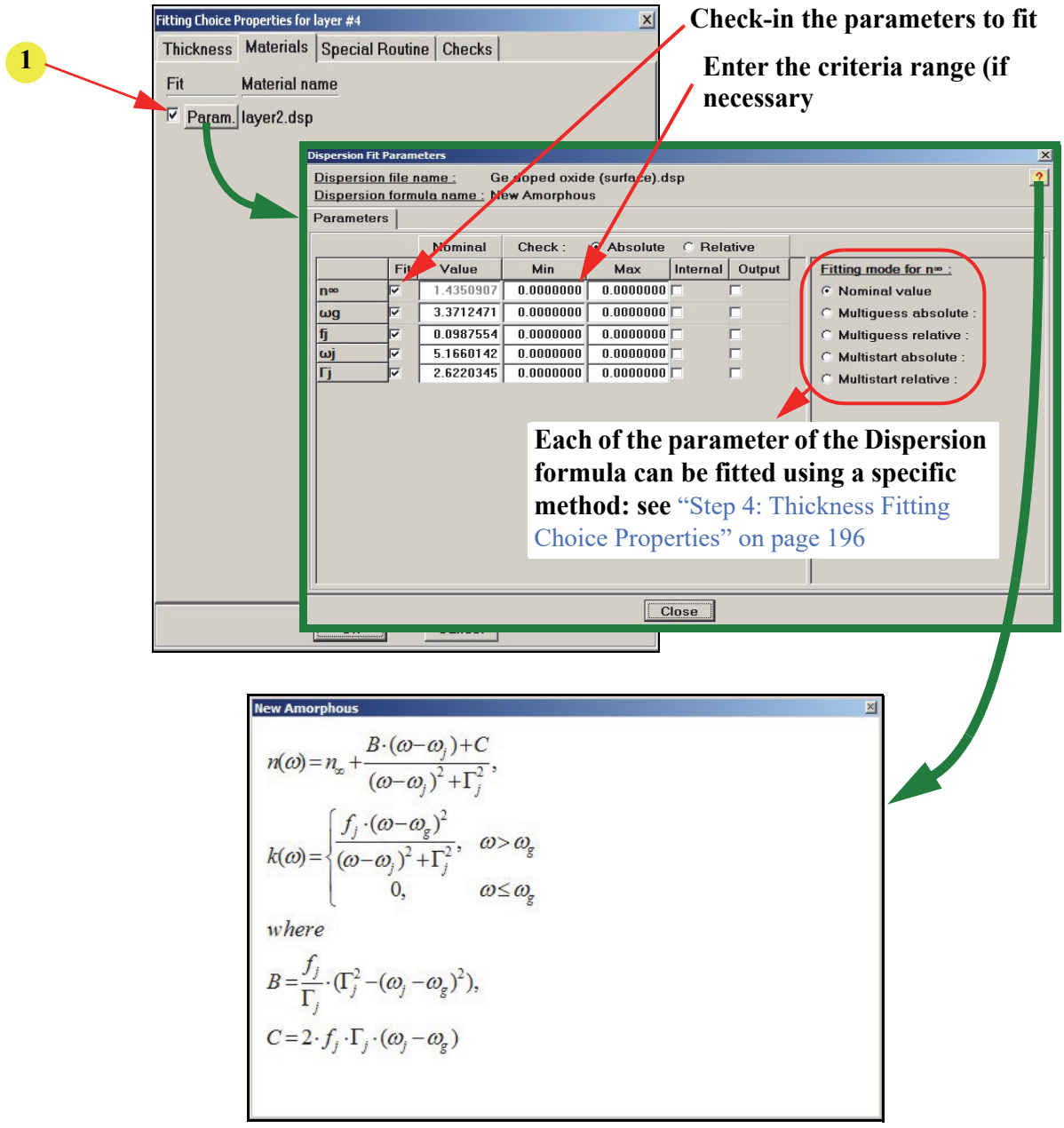



**Figure 13-16 Dispersion fit parameters**

05/20/2016

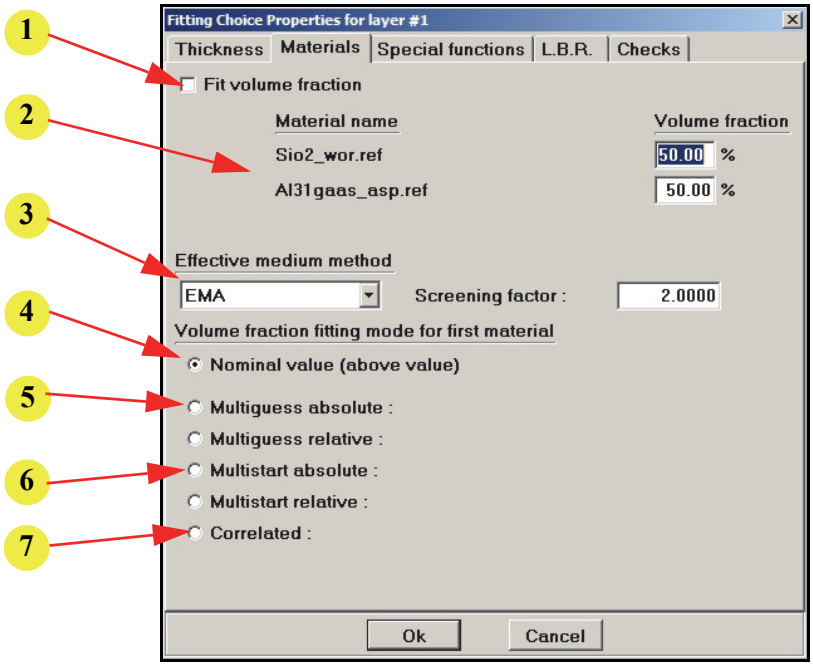



**Figure 13-17 Materials choice properties (several materials)**

| | |
|---|---|
| **Fit Volume fraction** 1 | Check the box to activate the fitting process on the Volume fraction of the current layer or slot. |
| **Fit Material name** 2 | The materials which are included in the layer are listed here. Check the material to fit and adjust the volume fraction if necessary; total must be 100%. |

05/20/2016

| | |
|---|---|
| **Effective medium method**<br>3 | **Select the appropriate medium method.**<br>This algorithm performs a specific materials distribution. It supposes that each of the layers, is composed of two or three materials. In such a case the effective dielectric function (EDF) of the substrate and the layers is calculated with one of the following effective medium models:<br>• **Maxwell-Garnett (MG) model**<br>The Maxwell-Garnett (MG) model, which describes a composite, consisting of inclusions completely surrounded by host material. In this model volume fractions $f_1$ and $f_2$ of inclusions 1 and 2 have to satisfy the following conditions: $f_1$ «1 and $f_2$ «1. The model includes a characteristic parameter-depolarization factor p, which depends on the constituent fraction shape. Then, p=1/3 for a spherical shape of inclusions.<br>• **Bruggeman effective medium approximation (EMA)**<br>The Bruggeman effective medium approximation (EMA) model describes a composite of aggregated phases or random-mixture microstructure. The model includes a characteristic parameter screen factor q, which depends on the accumulation of charge at the boundaries between separate aggregate phases (grain boundaries). Therefore, q=1 if the composite is microscopically isotropic in two dimensions, q=2 if the composite is isotropic in three dimensions, q=0 if an electric field of light wave is applied perpendicularly to the grain boundaries, q»1 if internal boundaries are parallel to the applied field.<br>• **SSC (Sen Scala Cohen) Model**<br>This effective medium theory model describes certain polycrystalline metals, whose boundaries optically isolate neighboring grains, metal and its oxide (or grain boundary component) in low fraction. |
| **Volume fraction fitting mode**<br>4 | **Nominal Value**<br>If checked, the values entered above will be used. |
| **Multiguess**<br>5 | The ***Multiguess*** choices allow the user to perform a fitting process with different initial/end values of ***Volume fraction***. The retained fitted ***volume fraction*** is directly related to the better $\chi^2$.<br>Choose the guess range with care as the Fitting process will scan all of the possibilities and it could drastically increase the process time. |

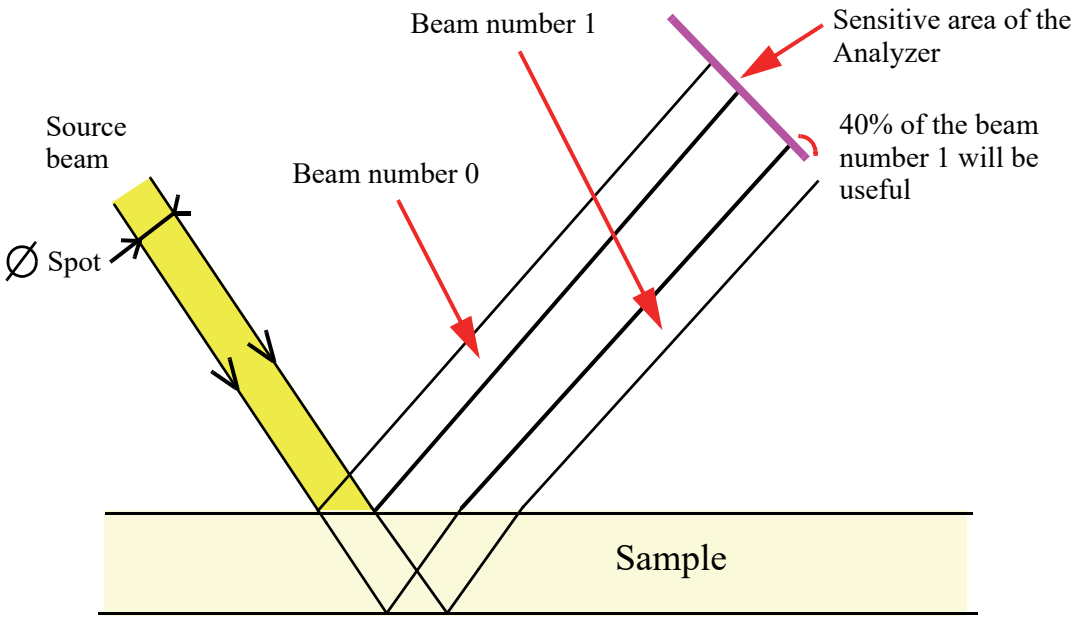
| | |
|---|---|
| **Multistart** 6 | The ***Multistart*** option will perform a fit on all of the fraction volume defined from the entered ***Variation*** and ***Increment*** values. These fraction volumes are not fitted as for the Multiguess option. The retained fraction volume is directly related to the better $\chi^2$. |
| **Correlated** 7 | Using this option, a layer thickness is proportionally progressed with another layer during the fitting. The ***«source»*** layer can be chosen from the Fitting parameters of the correlated layer. A fix or variable ratio factor can be applied |

**Special functions**

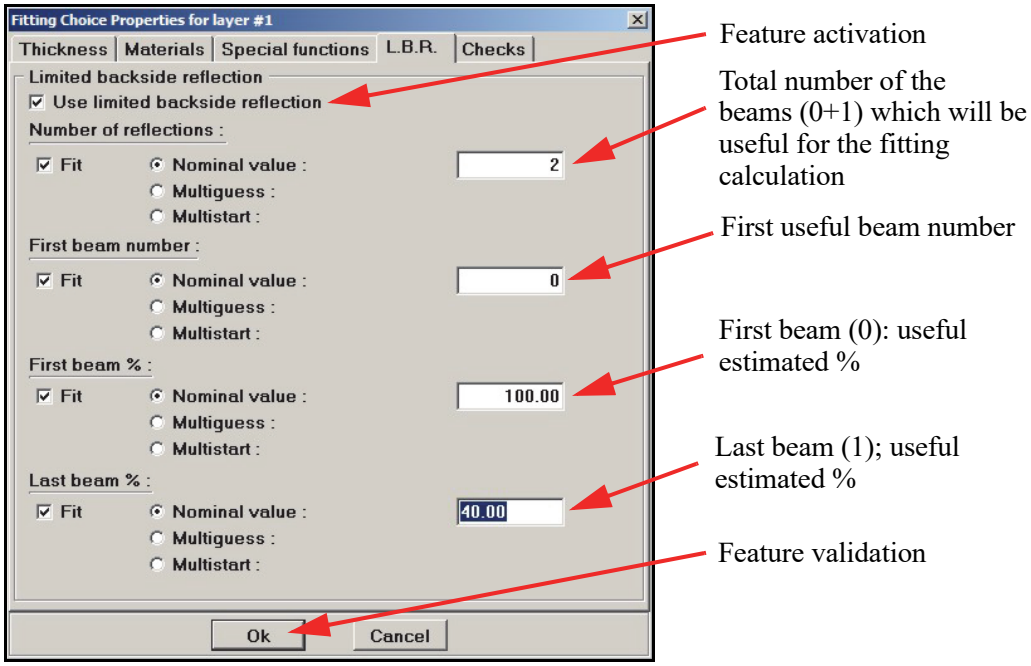
This feature is not yet implemented

**LBR (Limited Backside Reflection)**

The LBR feature is used to assign a specific percentage of the backside reflection for a modeling process. This could be useful when the detector aperture is able to capture the backside of multiple reflections. The diagram below shows an example of collecting less than two complete reflections:




According to the diagram above, the entered parameters for LBR fitting will be as shown below:




## Step 6: *Output check criteria* setup

The output check criteria, which limits values when a simulation or a fitting is running, is located on the level of the concerned parameters. For example, the figure below shows the check criteria for a layer.

**Access**: right-click on the layer and select ***fitting parameters*** choice, then the ***checks*** tab. By definition, substrate has not thickness, nor limits.

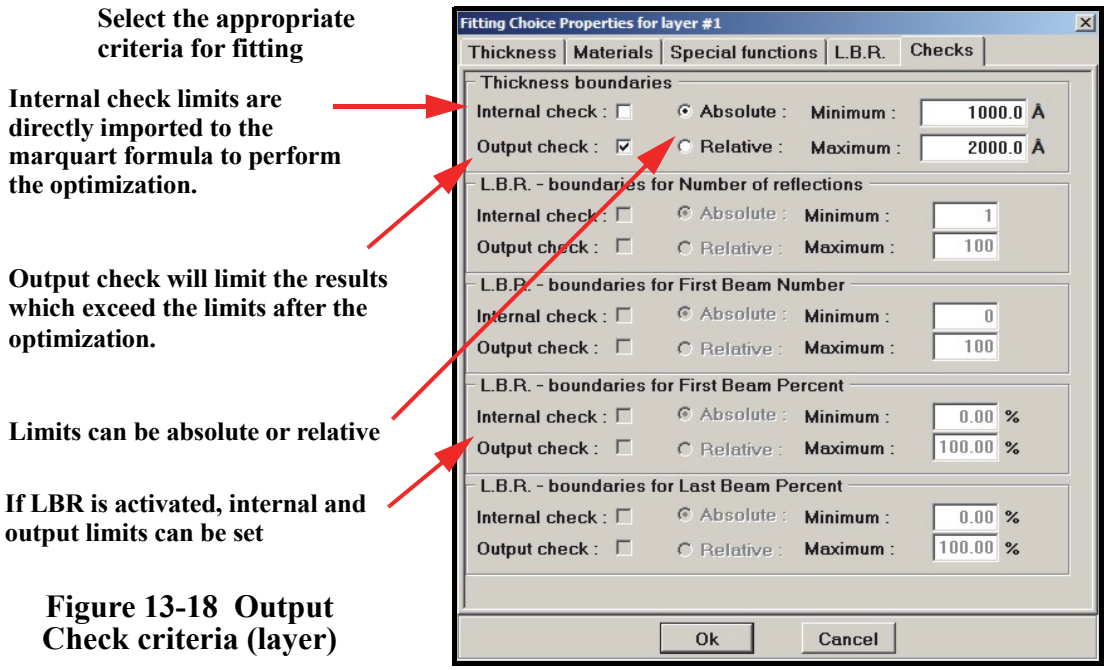



**Figure 13-18 Output Check criteria (layer)**

05/20/2016

The figure below shows the check criteria for the angle of incidence. The general operating diagram is the same as for the fitted layers.

**Internal check limits are directly imported to the marquart formula to perform the optimization.**

**Output check will limit the results which exceed the limits after the optimization.**

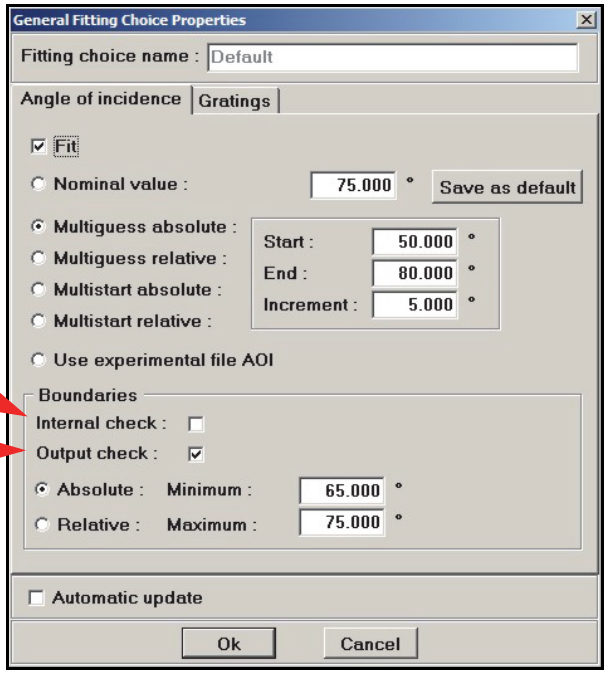
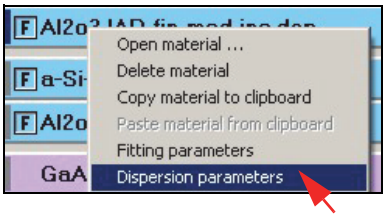



**Figure 13-19 Output Check criteria (Angle of incidence)**

### Fast Access to Dispersion parameters

- **Accessing**

  Right-click on the material name and select ***Dispersion Parameters*** choice.




## Step 7: Launching Fitting process

When the Model has been built, adjusted, and the elements configured to fit, check the following procedure:

- Click on ***Save*** or ***Save As*** button to save the built and configured model. It is recommended to save with the ***Both*** option (Sample and Model).

**Clear Results before fit**: checking this option, located in the main Model screen, will erase the previous fitting result text. If not checked, each fitting result text will be added.

- Click on the ***Fit*** button to launch the fitting process. During the process a new real time display screen will be opened. This screen shows, using graphic and text mode, the evolution of the fit. Here below is an example of this screen:

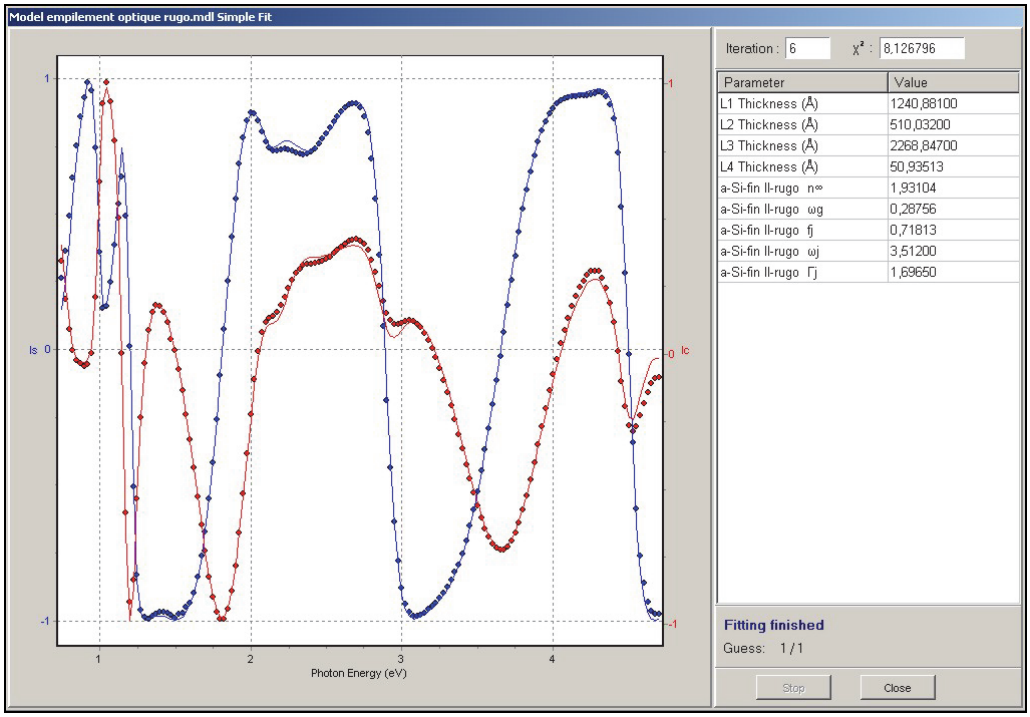



**Figure 13-20**

- This screen can be then close as all of the values are displayed on the main Model screen:

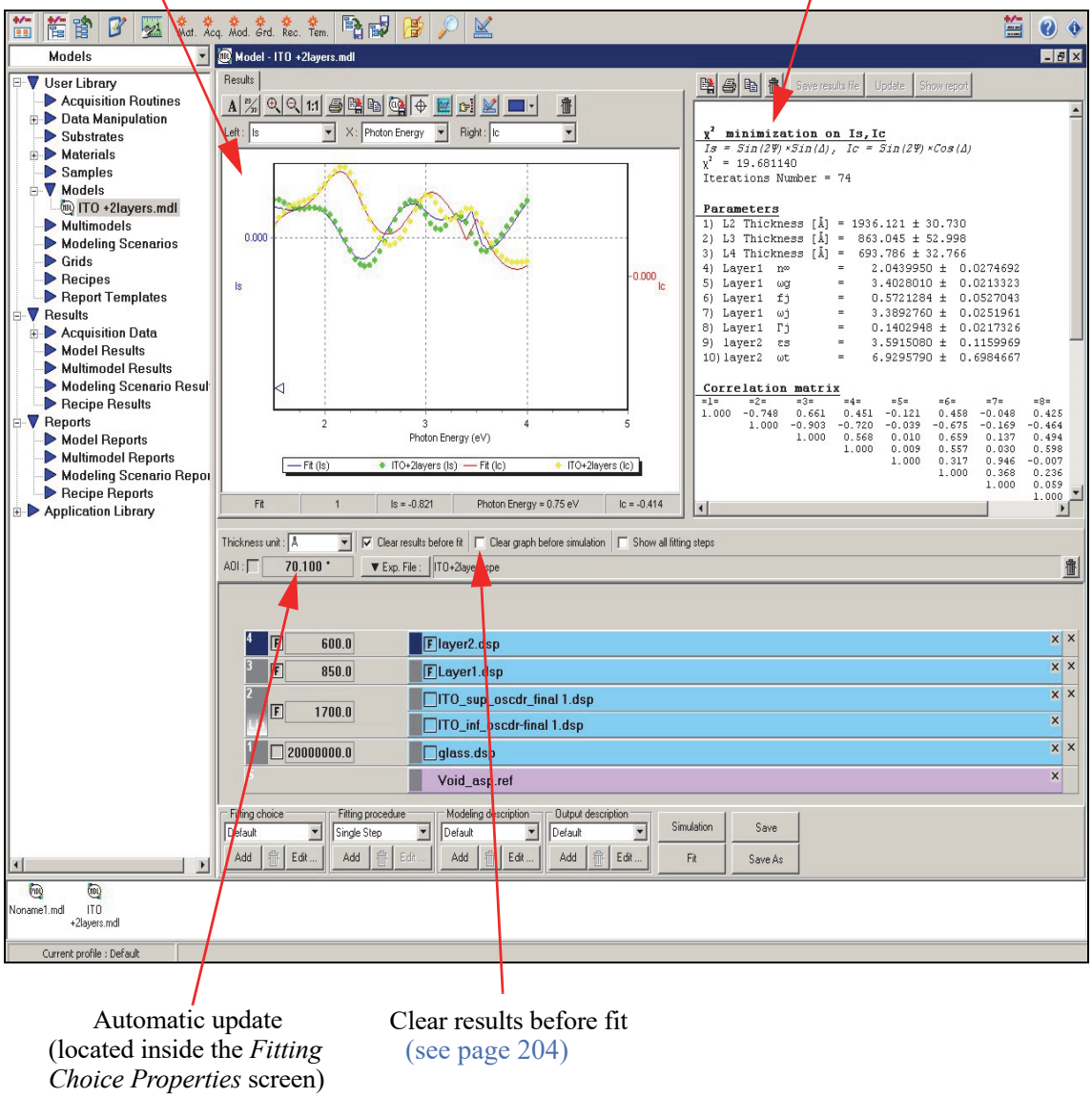



**Figure 13-21 Fitting results**

# 13.7 Advanced Model Post-calculation features

From the ***Output Description*** screen, two additional tabs include parameters which can be added to the «text mode» fitting results or to a report. The chapter will detail the ***Model Post-calculations*** features. Figure 13-22 shows the ***Model Post-calculations*** screen.

**NOTICE:** additional Observables will be added and calculated only if a fitting is launched.

## 13.7.1 Adding an additional Observable

An additional Observable **R, T,** α, 1/α can be added to the fitting results. The following procedure describes the steps:

- The user can add four additional ***Observables*** which will be calculated and add to the fitting results as a text on the upper right hand of the fitting screen or to an output reporting layout.
- Select an ***Observable***,
- Do not choose the range, this option is not yet implemented,
- Check in the ***Table*** parameter and select the unit,
- Click on the Add value to open a table. Enter the first wavelength value and check in to activate it for the calculations; boundaries values are optional,
- Any amount of values can be added in this way.
- If necessary, select another ***Observable*** and proceed in the same way.
- If you wish to add calculated thicknesses, follow the chapter "Thicknesses calculations" on page 207, if not click on the Save to file button or validate the screen by clicking on the ***OK*** button.

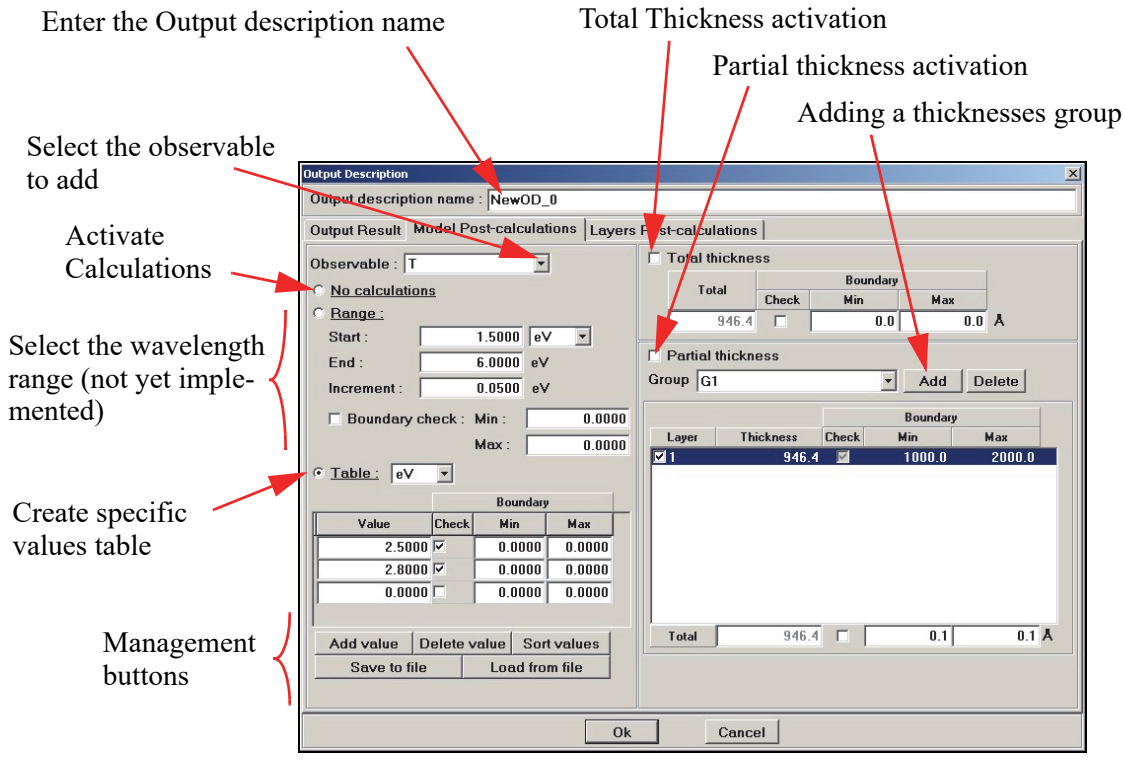



**Figure 13-22 Model Post-Calculations**

### 13.7.2 Thicknesses calculations

The right half part of the ***Model Post-calculations*** screen concerns the thicknesses calculations. Then, each layer or a grouped layers can be added to the «text mode» fitting results or/and to a report. Figure 13-22 shows the ***Model Postcalculations*** screen. The following procedure describes the steps:

**1** Select the ***Total thickness*** option to calculate and added the value to the fitting results or/and to the report,

**2** Select the ***Partial thickness*** to group layers and calculate the thickness. The interface is very flexible, as many groups of layers can be created.

- Click on the Add to open a new group of layers. A pop-up screen is then displayed and asks for a group name. Enter a name, then click OK to validate.
- The Group is created and a table lists all the Model layers.
- Check in the layers number (located on the table) on which the thickness calculation must be applied. Here below is an example:

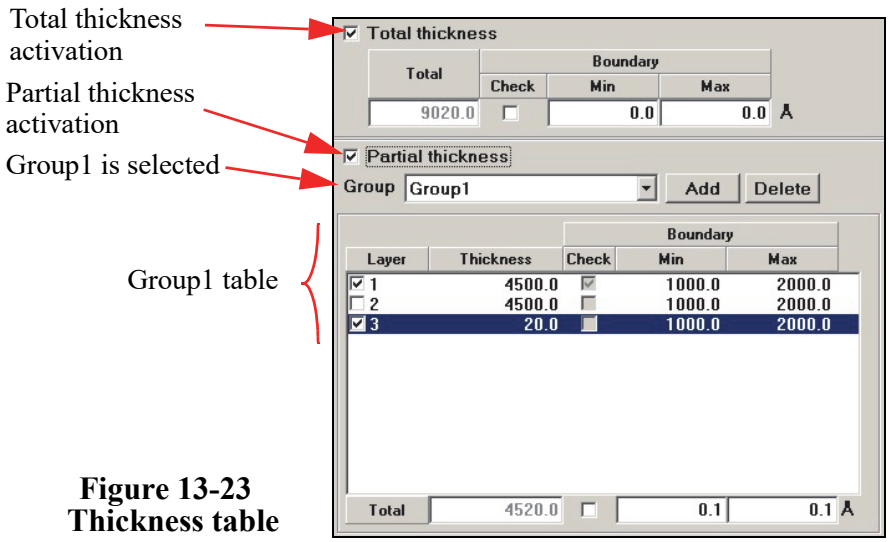



**Figure 13-23**
**Thickness table**

- Figure 13-23 shows the Total and Partial thicknesses activation. Group1 has been created and the table shows the selected first and third layer. The total selected thickness is displayed on the bottom and the total boundaries thickness can be specified and activated. The individual boundaries checking cannot be activated from this table; please refer to the Fitting parameters choices (see "Parameters Setting" on page 186).
- Once the groups have been setup, click on the Save to file button or validate the screen by clicking on the ***OK*** button.

# 13.8 Advanced Layers Post-calculations

From the ***Output Description*** screen, two additional tabs include parameters which can be added to the «text mode» fitting results or to a report. The chapter will detail the ***Layers Post-calculations*** features. Figure 13-24 shows the ***Layers Post-calculations*** screen.

**NOTICE:** additional Observables will be added and calculated only if a fitting is launched.

## 13.8.1 Adding an additional Observable

An additional Observable **n, k,** εr, εi can be added to the fitting results. The following procedure describes the steps:

- The user can add four additional ***Observables*** on each layer of the Model. These ***Observables*** will be calculated and added to the fitting results as a text on the upper right hand of the fitting screen or to an output reporting layout.
- Select an ***Observable***,
- Do not choose the range, this option is not yet implemented,
- Check in the ***Table*** parameter and select the unit,
- Click on the Add value to open a table. Enter the first wavelength value and check in to activate it for the calculations; boundaries values are optional,
- Any amount of values can be added in this way.
- If necessary, select another ***Observable*** and proceed in the same way.
- If you wish to add the ***Ge composition*** and/or the ***Eg*** parameters, follow the chapter "Adding Ge composition and Eg parameters" on page 209, if not click on the Save to file button or validate the screen by clicking on the ***OK*** button.

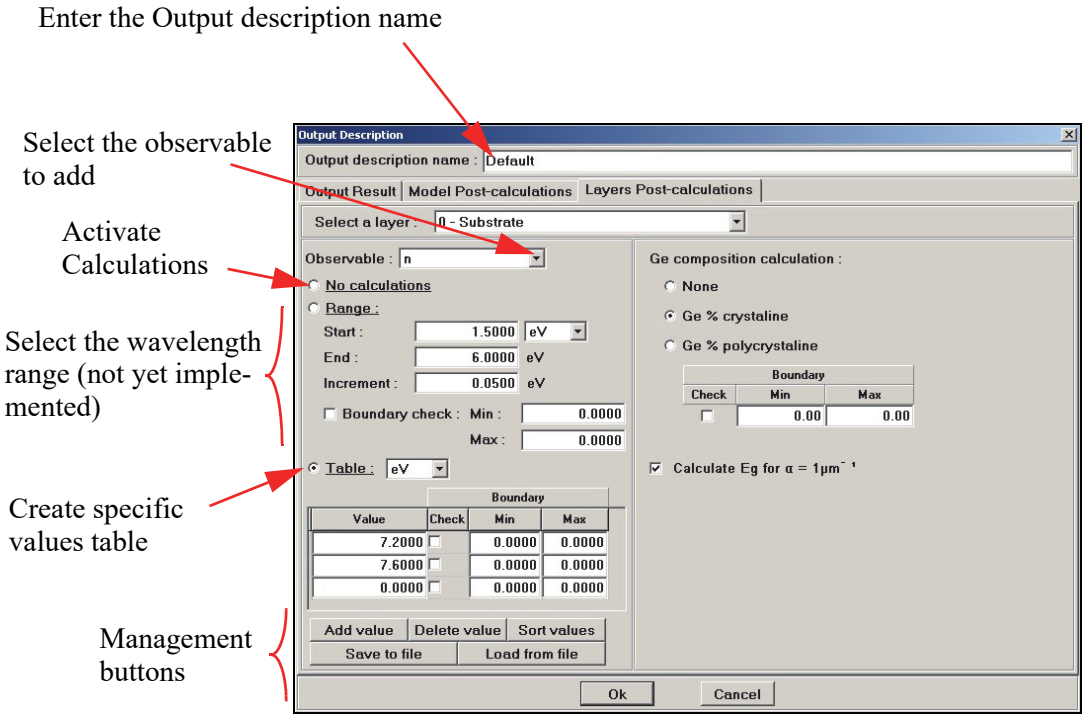



**Figure 13-24 Layers Post-Calculation**

### 13.8.2 Adding Ge composition and Eg parameters

The right half part of the ***Layers Post-calculations*** screen concerns the **Ge** composition (%) and **Eg** (for $\alpha$=1µm$^{-1}$) calculations.

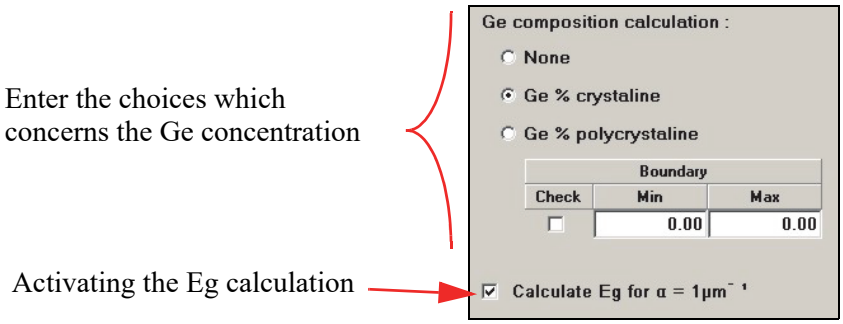



**Figure 13-25 Ge and Eg Post-calculations**

# 14 Serial Multimodel

## 14.1 Definition

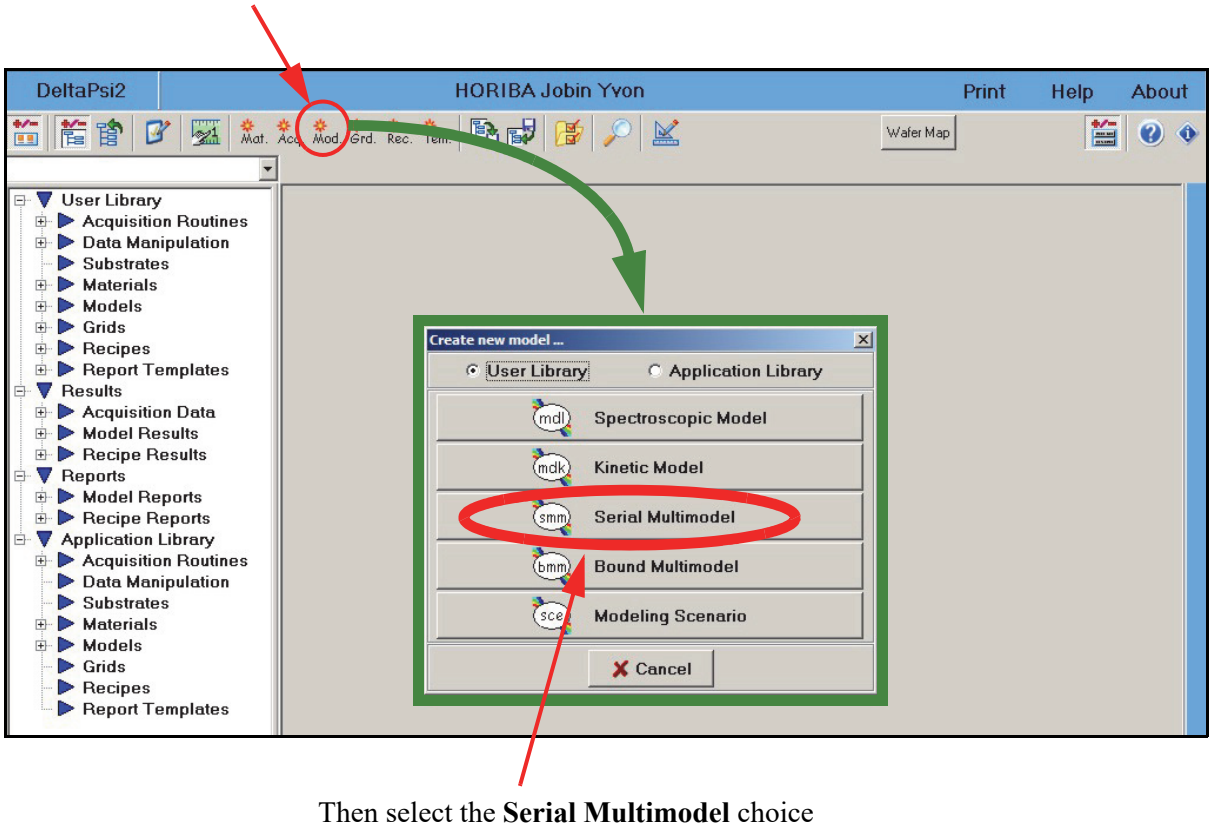
The Serial Multimodel is a useful feature which performs a fit on many models using one common Acquisition Data. However, a multispectra Acquisition Data can be used; in this case the index of the multispectra file must be specified for each Model.

The results are separate for each Model.

## 14.2 Accessing

The figure below shows how to access to the ***Serial Multimodel*** window:

Click on the ***Mod*** icon located on the Main icons bar




Then select the **Serial Multimodel** choice

## 14.3 Description

The figure below shows the main Serial Multimodel screen.

05/20/2016

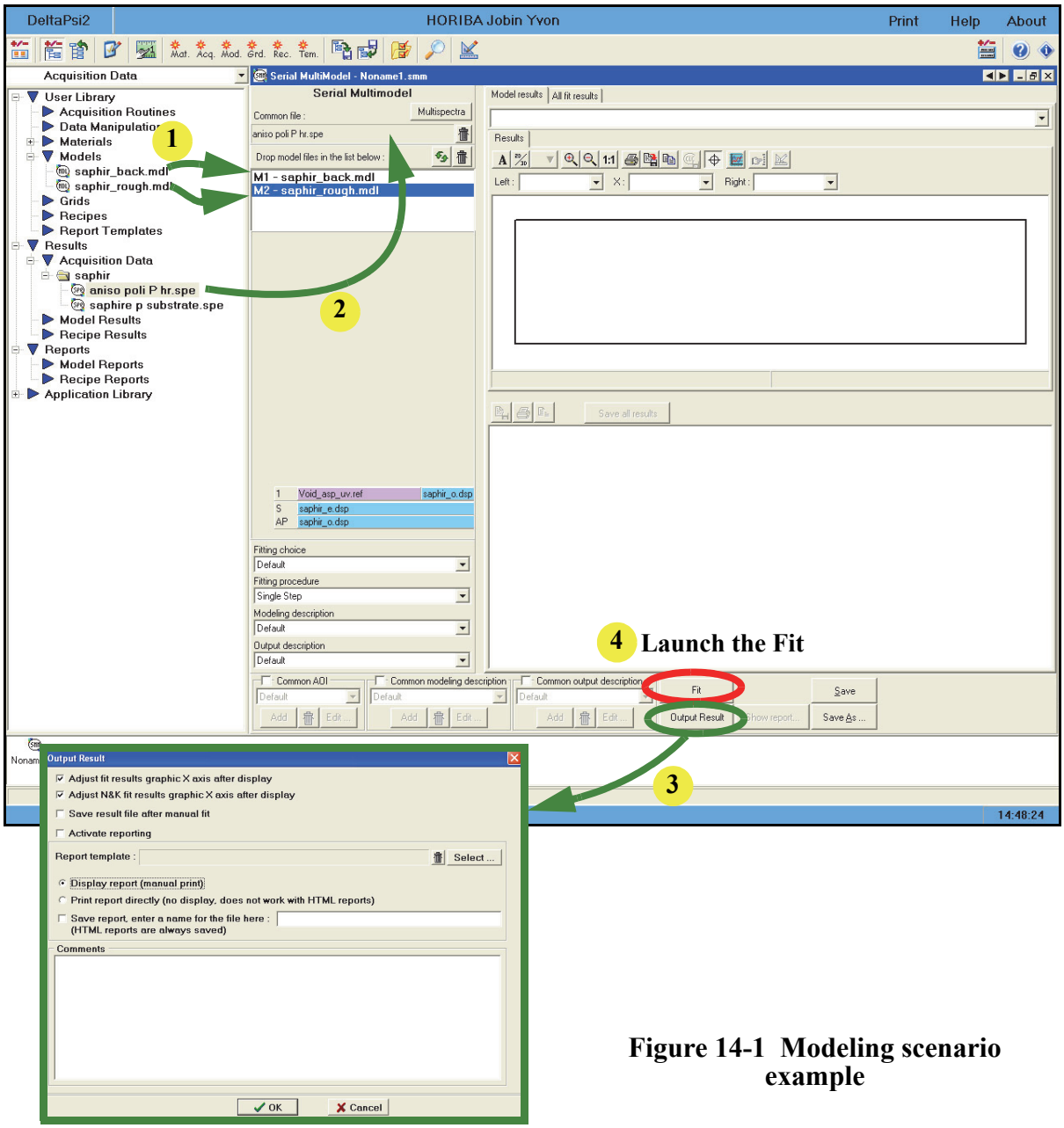



**Figure 14-1 Modeling scenario example**

| | |
|---|---|
| 1 | Drag and drop all the models you want to apply to a specific Acquisition Data |
| 2 | Drag and drop the Acquisition Data file you want to apply to the Models.<br>This unique Acquisition Data file can be mono-spectrum data (.spe, .ref, .clc, .spb) or multispectra data (.acr). In case of multispectra data, you must specify the index number of spectrum to use for each Model (See detailed explanation "How to assign an index number" on page 212). |

05/20/2016

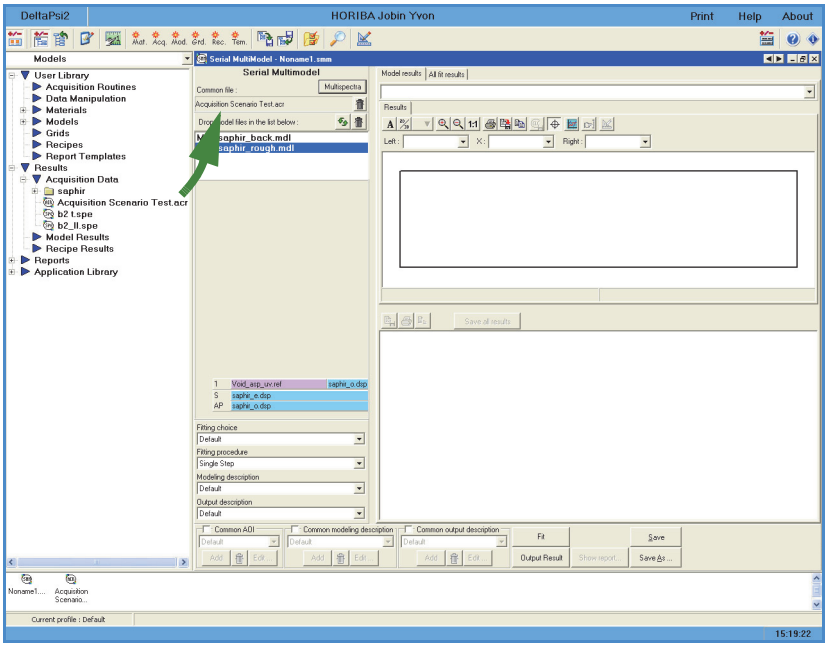
| | |
|---|---|
| 3 | Fill the Output Result screen to adapt the output format and presentation. |
| 4 | Launch the Fit process to all the listed Models with the selected Acquisition Data file. |

## 14.4 How to assign an index number

Multispectra Files have a specific data format. The included spectra are grouped as results of Acquisition or Data Manipulation.

When using a Multispectra file in the Serial Multimodel window, an index number is required. The figure below shows how to assign an index to a specific spectrum. Follow the procedure shown below:

**1** Drag and drop an **.acr** file into the ***Common file*** field



**2** Double-click on the .acr file. This action will open and display the file parameters. It becomes possible to know what kind of spectra are included inside.

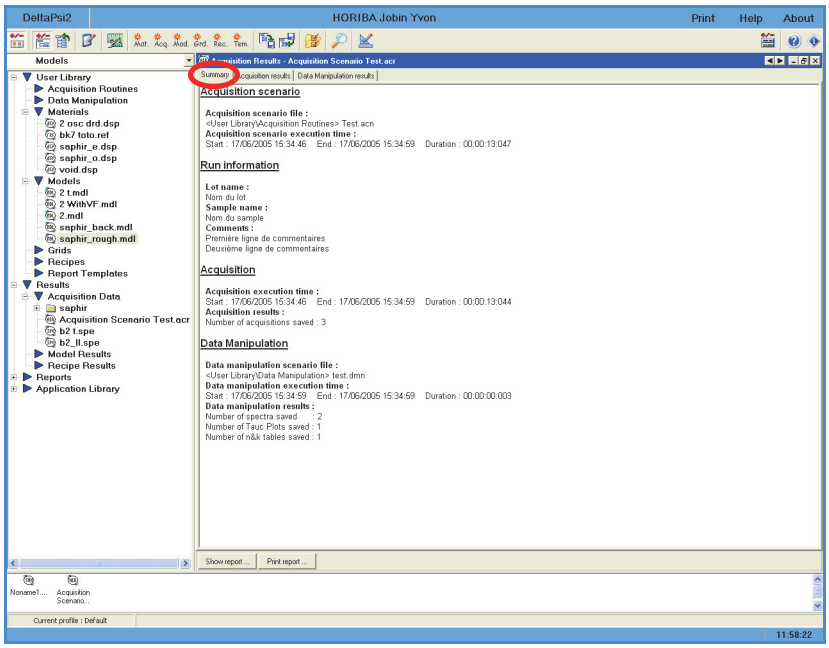
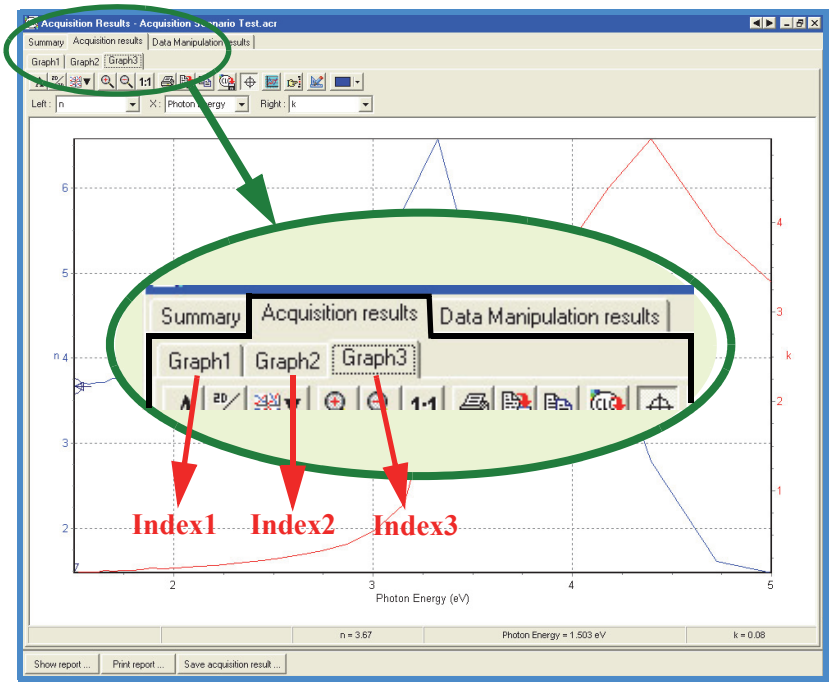
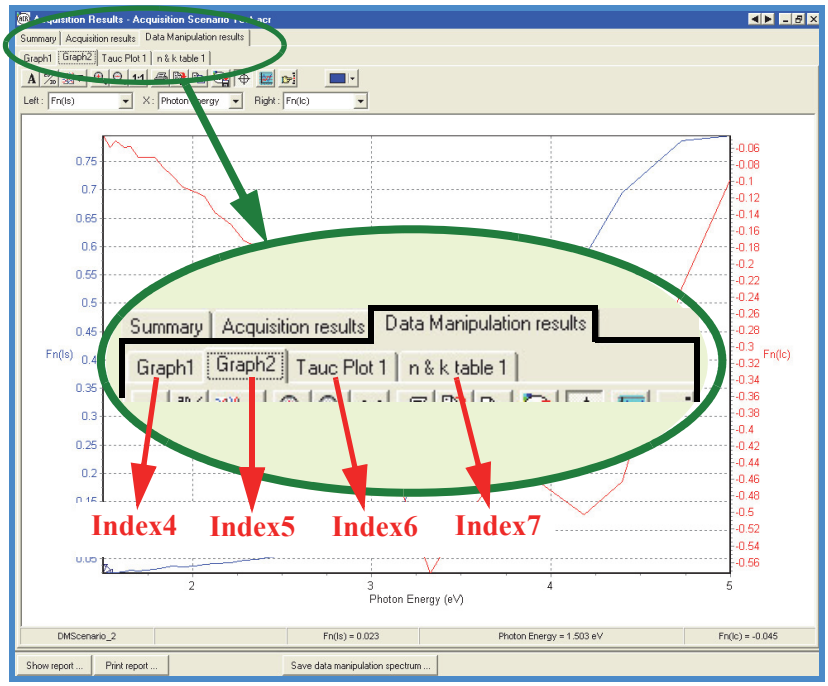



**Figure 14-2 File Summary**

Figure 14-2 shows the ***Summary*** of the file. A text details the characteristics of the **acquisition** and the **data manipulation** operations used in the file. In the ***Acquisition*** paragraph, we can see that 3 acquisitions have been saved and in the ***Data Manipulation*** paragraph, two files have been saved and one Tauc Plots manipulation and one n&k tables.

On the upper part of the screen, two additional tabs labelled ***Acquisition Results*** and ***Data Manipulation*** results graphically displays the results of each included file.






On the figures above, the spectra are renammed in indexes required by the **Serial Multimodel** feature. The first index is the first spectrum of the ***Acquisition Results*** tab. The numbering continues on the next tab ***Data Manipulation Results***.

# 15 Bound Multimodel

## 15.1 Definition

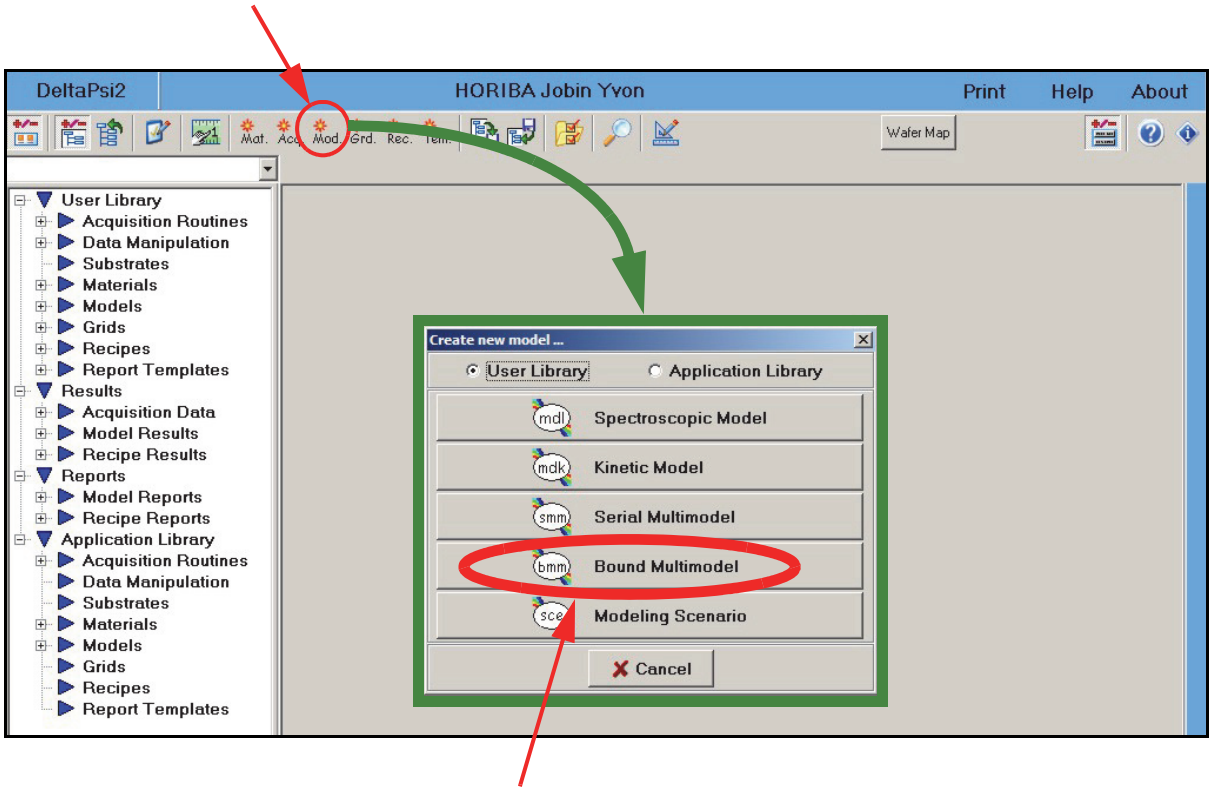
The Bound Multimodel is a powerful feature which performs fits on many different models. The most interesting is the possibility to bind parameters between the models. The resulted fit shows the fitted parameters, bound and not-bound.

Each model can keep its own Acquisition Data or can be shared a common Acquisition Data. This common Acquisition Data could be a multispectra. In this case a multispectra index must be assign to each model.

## 15.2 Accessing

The figure below shows how to access to the ***Bound Multimodel*** window:

Click on the ***Mod*** icon located on the Main icons bar

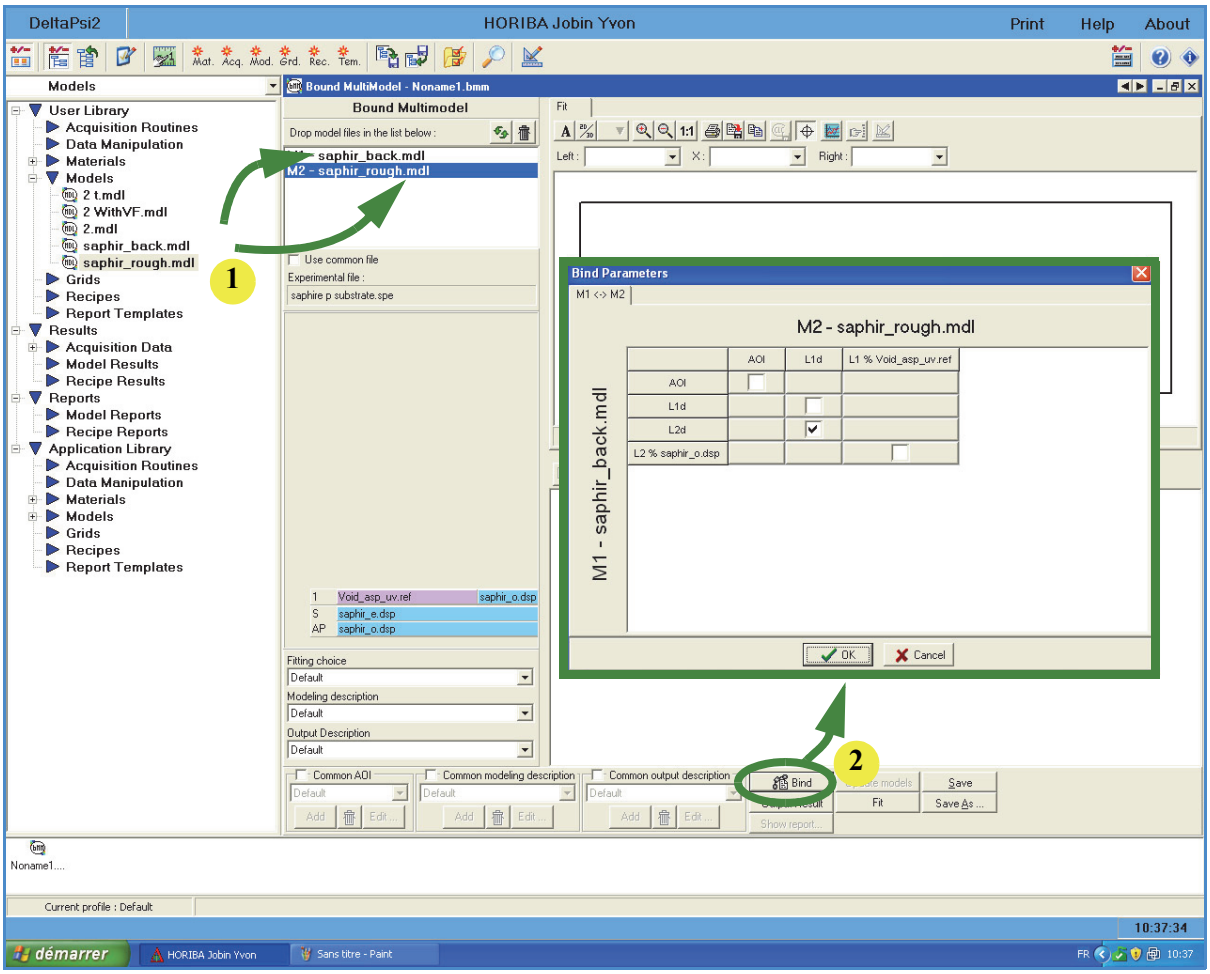



Then select the **Bound Multimodel** choice

## 15.3 Description

The figure below shows the main Bound Multimodel screen.

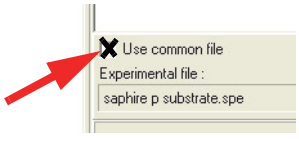
05/20/2016



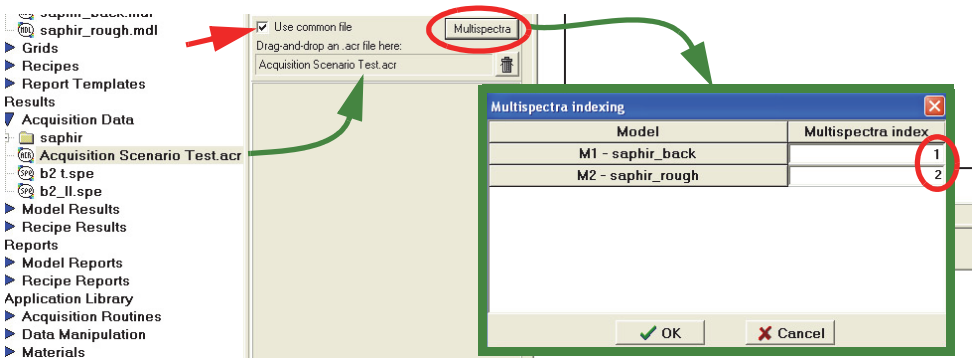

## 1st Case: Models use their own Acquisition Data file

| | |
|---|---|
| 1 | Drag and drop all the models you want to process. By default, all the models keep its own ***Acquisition Data***. |
| 2 | Click on the ***Bind*** button to open the ***Bind Parameters*** window. Check in the parameters to bind. It means that each bound parameter will be fitted to identical value. |

## 2nd Case: Models use a common Acquisition Data file

| | |
|---|---|
| 1 | Drag and drop all the models you want to process. By default, all the models keep its own ***Acquisition Data***. |
| | Check the ***Use Common file*** box, then drag and drop the Acquisition Data file you want to apply to all the models.  |
| 2 | Click on the ***Bind*** button to open the ***Bind Parameters*** window. Check in the parameters to bind. It means that each bound parameter will be fitted to identical value. |

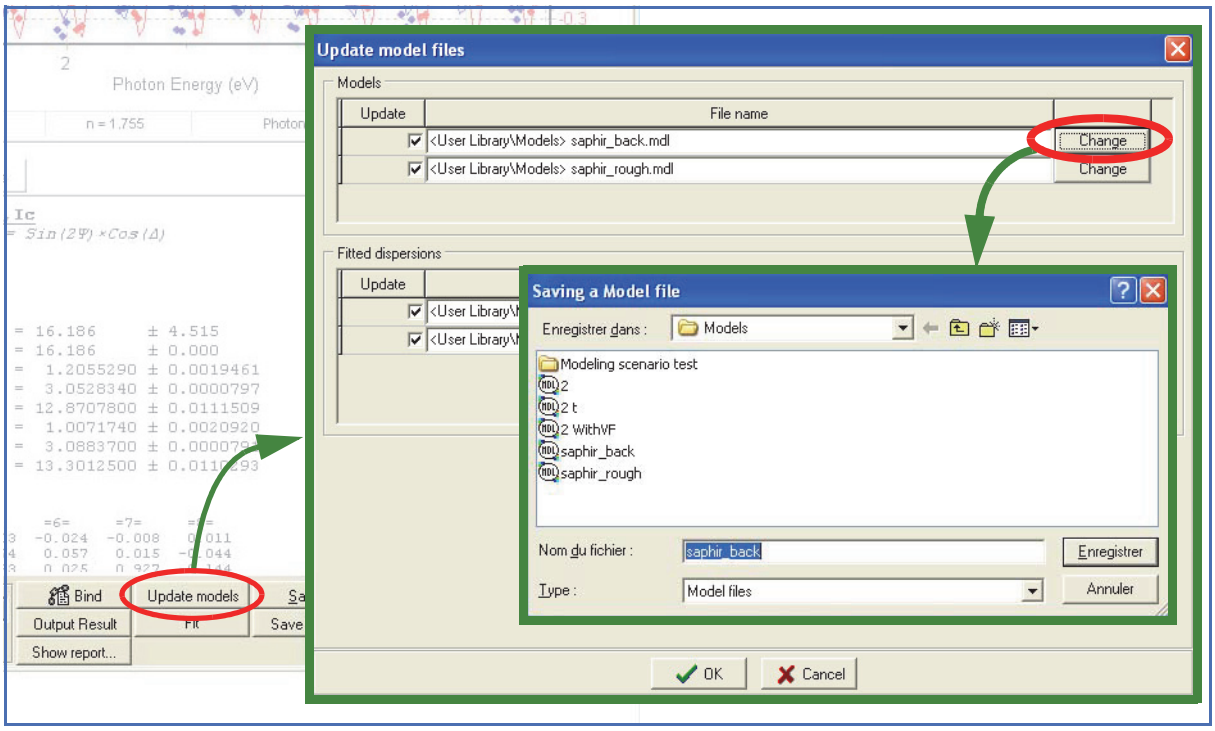
### 3rd Case: Models use a common Multispectra Acquisition Data file

| | |
|---|---|
| 1 | Drag and drop all the models you want to process. By default, all the models keep its own ***Acquisition Data***. |
| | Check the ***Use Common file*** box, then drag and drop the Multispectra Acquisition Data file you want to apply to all the models.<br>Click on the ***Multispectra*** button to display the Multispectra indexing window: enter an index for each model. See the chapter "How to assign an index number" on page 212.<br>  |
| 2 | Click on the ***Bind*** button to open the ***Bind Parameters*** window. Check in the parameters to bind. It means that each bound parameter will be fitted to identical value. |

### Update Models

The ***Update Models*** button is located on the right hand side of the ***bind*** button. This feature allows the user to update models files.

Once clicking on the ***Update Models*** button the following screen will be displayed:




By default, all the modified parameters by the fit are checked and if you click on the ***OK*** button, these parameters will immediately overwrite the initial ones.

If you want to keep the initial parameters files, click on the ***Change*** button to rename the modified parameters to a new filename.

# 16 Modeling Scenario

## 16.1 Definition

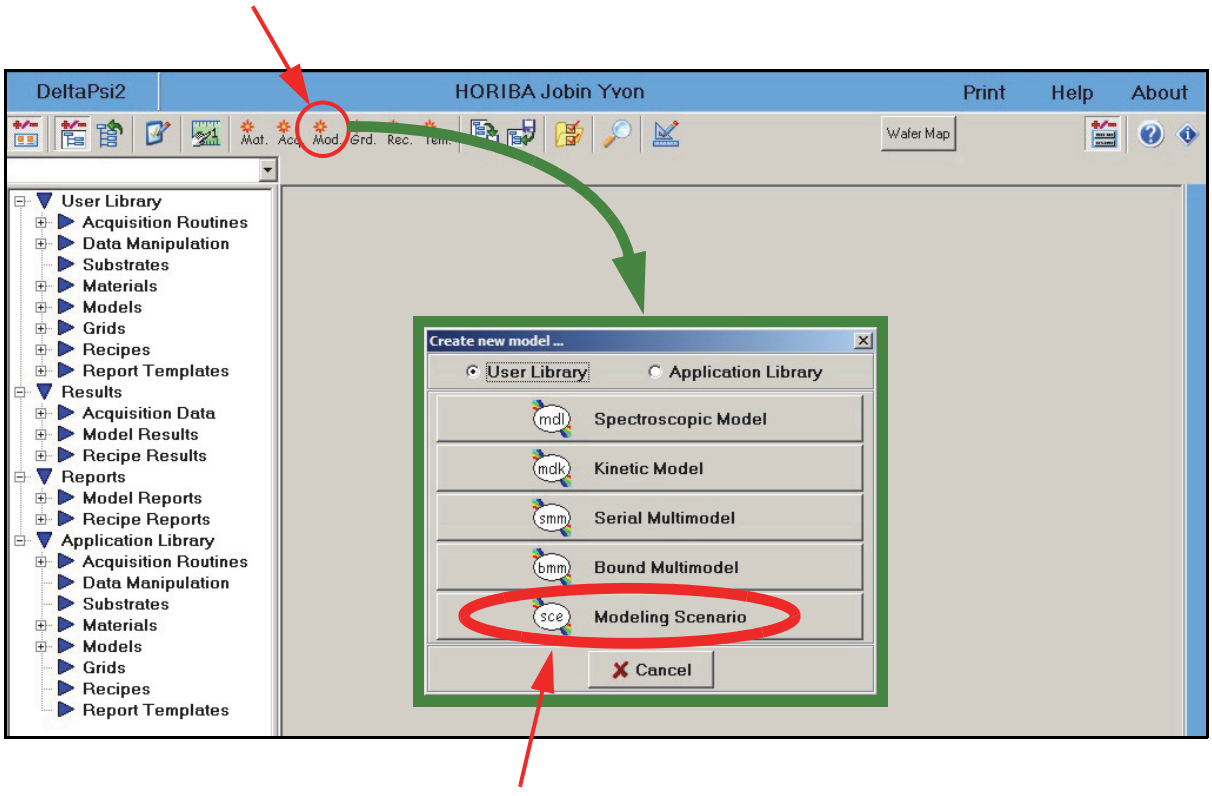
The Modeling Scenario has been created to automate the Modeling around many different Models using one Acquisition Experimental file. Moreover to this automation, an extraordinary scripting tool has been added: Each modeling step can be joined to a dedicated script which can extract results, performs additional calculations, compares results from different steps and, on the fly, selects and performs the best modeling on an entire mapping.

The detailed use of the script is explained in the "DeltaPsi2 Scripting Reference", page 287.

## 16.2 Accessing

The figure below shows how to access to the ***Modeling Scenario*** window:

Click on the ***Mod*** icon located on the Main icons bar




Then select the **Modeling Scenario** choice

## 16.3 Description

The figure below shows the main Modeling scenario screen.

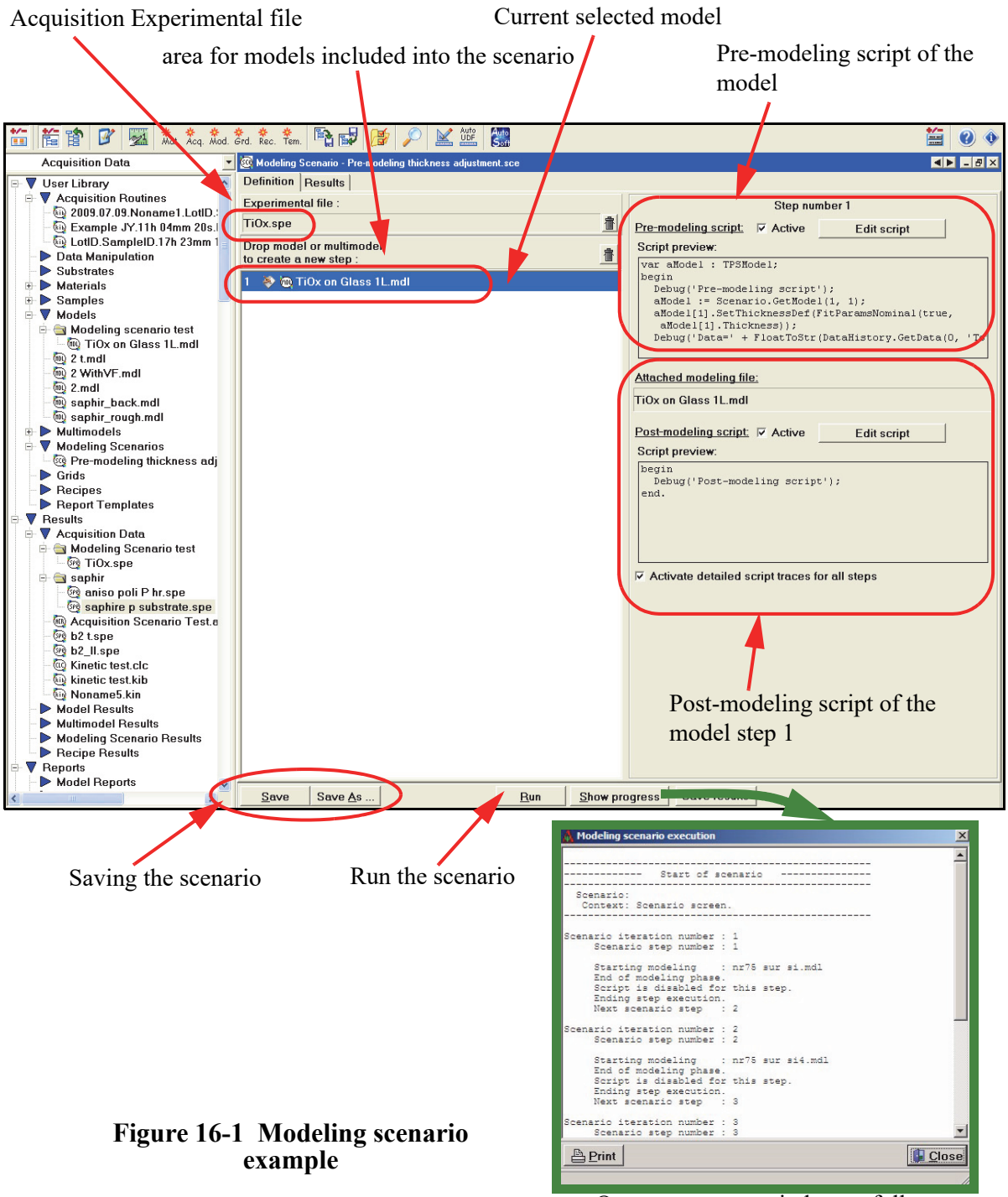



**Figure 16-1 Modeling scenario example**

### 16.3.1 Procedure

The scenario is based on steps linked if necessary to scripts. When a scenario is launched, it follows the steps incrementally (if the script does not include an instruction to change the step execution order).

In fact, without script, the Modeling Scenario will generate a modeling result for each step and select the last one as the best result. The most powerful part of the

05/20/2016

Modeling Scenario concerns the ability to program an "intelligent" judgement and a discriminated answer to the results of each step. The above scenario shows an example of such application.

Scenario example details:

- Modeling of TiOx material on c-Si substrate.
- The first step will fit the thickness from 500Å to 5000Å with an increment of 500Å.
- The second step uses a model with an additional roughness layer of 20Å located above the main TiOx layer.
- The third step uses another model with an additional interface layer of 20Å below the main TiOx layer.
- The goal is to determine the best model and the dispersion fitted values will be transferred as a new starting guess values after each execution of the scenario.
- The complete 3 scripts are reported below and each instruction is described.

- This scenario does not have a pre-modeling script (do not activate it)

## 1st Script

```
/
// Roughness-Interface selection demo optimized
// for quick recipe execution
//
// This scenario shows how to make a simple modeling,
// transfer dispersion parameters to more advanced models,
// run roughness and interface models, then select the
// best result from the three models.
//
// The selection is performed only once in a recipe execution.
// The best model is then used alone for the following points
// of the recipe.
// The dispersion fitted values are also transferred as new
// starting guess values after each execution of the scenario.
//
// These variables will be used to access the different models
var aModel1 : TPSModel;
    aModel2 : TPSModel;
    aModel3 : TPSModel;
    // This variable is used to know if we are on the first
    // step of a recipe execution
    aFirst  : boolean;
begin
  // Point on model of step 1 (single layer, the current one)
  aModel1 := Scenario.GetModel(1, 1);
  //
  // Test if we are on the first execution of the scenario or not
  //
  aFirst := not(GetGlobalVar('FirstStepDone'));
  if aFirst
  // We are on the first execution of the scenario
  // Setting the global variable will be done in the last step
  // of the scenario, but we have to update the next 2 models,
  // continue the execution and make the final choice
  then
    begin
      // Point on model of step 2 (roughness model)
      aModel2 := Scenario.GetModel(2, 1);
      // Point on model of step 3 (interface model)
      aModel3 := Scenario.GetModel(3, 1);
      //
      // Save Khi2 of step 1 (current) for final comparison
      //
      SetGlobalVar('Model1Khi2', aModel1.Khi2);
      //
      // Make sure we are on the correct fitting choice in the next models
      // This is optional and needed only if you have several fitting choices
      // in the attached models and want to make sure of the one receiving
      // transferred parameters.
      //
      aModel2.SetFittingChoice('Default');
      aModel3.SetFittingChoice('Default');
      //
```

```
      // Transfer TiOx.dsp dispersion parameters values to the all the models
      // as fitted nominal values
      //
      // Note : working with the file names, you do not need to perform this
      //        copy for every occurrence of the dispersion within the model.
      //        The parameters will be automatically set in all the occurrences
      //        of the named dispersion present in the destination model.
      //
      // Current model : update the dispersion
      aModel1.CopyFormulaParameters(aModel1,
        aModel1[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        aModel1[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
      // Other models : set optimized starting values
      aModel1.CopyFormulaParameters(aModel2,
        aModel1[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        aModel2[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
      aModel1.CopyFormulaParameters(aModel3,
        aModel1[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        aModel3[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);

    end
  // We are not on the first execution of the scenario, so we have
  // to update the dispersion parameters of the current model and
  // exit the scenario with the final judgement
  else
    begin
      // Current model : update the dispersion
      aModel1.CopyFormulaParameters(aModel1,
        aModel1[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        aModel1[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
      // Exit from the scenario with the correct judgement
      if aModel1.Judgement
      then Scenario.StopScenario(sesExitGood, 'The modeling is good.')
      else Scenario.StopScenario(sesExitBad, 'The modeling is bad.');
    end;
  //
  // This step of the scenario does not have any more actions.
  //
end.
```

---

## 2nd Script

---

```
//
// Roughness-Interface selection demo
//
// This step continues with the roughness model execution.
//
// This variable will be used to access the model
var aModel : TPSModel;
    // This variable is used to know if we are on the first
    // step of a recipe execution
    aFirst : boolean;
begin
  // Point to the roughness model (step 2 is current)
  aModel := Scenario.GetModel(2, 1);
  //
  // Test if we are on the first execution of the scenario or not
  //
  aFirst := not(GetGlobalVar('FirstStepDone'));
  if aFirst
  // We are on the first execution of the scenario
  // Setting the global variable will be done in the last step
  // of the scenario, but we have to memorize the Khi2 value,
  // then continue the execution and make the final choice
  then
    begin
      //
      // Save the Khi2 value of the roughness model for final comparison
      //
      SetGlobalVar('Model2Khi2', aModel.Khi2);
      // Current model : update the dispersion
      aModel.CopyFormulaParameters(aModel,
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
```

```
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
    end
  // We are not on the first execution of the scenario, so we have
  // to update the dispersion parameters of the current model and
  // exit the scenario with the final judgement
  else
    begin
      // Current model : update the dispersion
      aModel.CopyFormulaParameters(aModel,
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
      // Exit from the scenario with the correct judgement
      if aModel.Judgement
      then Scenario.StopScenario(sesExitGood, 'The modeling is good.')
      else Scenario.StopScenario(sesExitBad, 'The modeling is bad.');
    end;
  //
  // This step of the scenario does not have any more actions.
  //
end.
```

---

## 3rd Script

```
//
// Roughness-Interface selection demo
//
// This step continues with he roughness model execution.
// Then, it performs the choice of the best result compared to
// the previous models.
//
// This variable will be used to access the interface model
var aModel      : TPSModel;
    // These variables are used to read the different Khi2 values
    aModel1Khi2 : single;
    aModel2Khi2 : single;
    aModel3Khi2 : single;
    aBestKhi2   : single;
    // This variable is used to know if we are on the first
    // step of a recipe execution
    aFirst      : boolean;
begin
  // Point to the interface model (current step is number 3)
  aModel := Scenario.GetModel(3, 1);
  //
  // Test if we are on the first execution of the scenario or not
  //
  aFirst := not(GetGlobalVar('FirstStepDone'));
  if aFirst
  // We are on the first execution of the scenario
  // The variable will now be set for next steps of the recipe execution,
  // then make the final choice of best result and best model for next
  // steps of the recipe
  then
    begin
      //
      // First, indicate that the first step is done
      //
      SetGlobalVar('FirstStepDone', true);
      //
      // Get the Khi2 value of the interface model
      //
      aModel3Khi2 := aModel.Khi2;
      //
      // Current model : update the dispersion
      //
      aModel.CopyFormulaParameters(aModel,
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
      //
      // Retrieve the Khi2 values of previous models
      // Step 1 : simple layer model
      aModel1Khi2 := GetGlobalVar('Model1Khi2');
      // Step 2 : roughness model
```

```
      aModel2Khi2 := GetGlobalVar('Model2Khi2');
      //
      // Optional debug traces to see the calculated Khi2 values
      // inside the scenario step summary
      //
      Debug('Khi2 - Single layer = ' + FloatToStr(aModel1Khi2));
      Debug('Khi2 - Roughness = ' + FloatToStr(aModel2Khi2));
      Debug('Khi2 - Interface = ' + FloatToStr(aModel3Khi2));
      //
      // This is the judgement block where we choose
      // the final result by the best Khi2 value and
      // set the starting step of the scenario according to the best
      // model thus found.
      //
      // By default, consider the current model as the best one
      aBestKhi2 := aModel3Khi2;
      Scenario.StartingStep := 3;
      // Test if the roughness model gives better results
      if (aModel2Khi2 < aBestKhi2) then
      begin
        // Roughness model gives now the best Khi2
        aBestKhi2 := aModel2Khi2;
        // We select its result as final result
        Scenario.BestResult := 2;
        // And select the second model as starting step
        // for next recipe steps
        Scenario.StartingStep := 2;
      end;
      // Then, we test the single layer model
      if (aModel1Khi2 < aBestKhi2) then
      begin
        // We select its result as final result if the Khi2 value is better
        Scenario.BestResult := 1;
        // And select the second model as starting step
        // for next recipe steps
        Scenario.StartingStep := 1;
      end;
    end
  else
    begin
      // Current model : update the dispersion
      aModel.CopyFormulaParameters(aModel,
        aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
         aModel[LI_LAYER1][SI_NONE][MI_MATERIAL1].FullName,
        true);
      // Exit from the scenario with the correct judgement
      if aModel.Judgement
      then Scenario.StopScenario(sesExitGood, 'The modeling is good.')
      else Scenario.StopScenario(sesExitBad, 'The modeling is bad.');
    end;
  //
  // The scenario ends here
  //
end.
```

---

**Example of pre-modeling script:**

If a Model includes a multiguess layer (see “Manipulation on a layer” on page 170), it could be useful to fix this layer thickness before a post-modeling. The pre-modeling script can fix a thickness value for this layer.

```
 var aModel : TPSModel;
begin
  Debug('Pre-modeling script');
  aModel := Scenario.GetModel(1, 1);
  aModel[1].SetThicknessDef(FitParamsNominal(true,
   aModel[1].Thickness));
  Debug('Data=' + FloatToStr(DataHistory.GetData(0, 'Toto')));
end.
```

# 17 GridEditor

## 17.1 What is the GridEditor?

GridEditor is a specific feature of the DeltaPsi2 Software. It allows to create and design an analysis path in a wide variety of substrate formats.

GridEditor is much more than just an editor and designer: it is a complete high precision mapper for analysis and control process with a high level repeatability control when it is used with the HORIBA Scientific Systems.

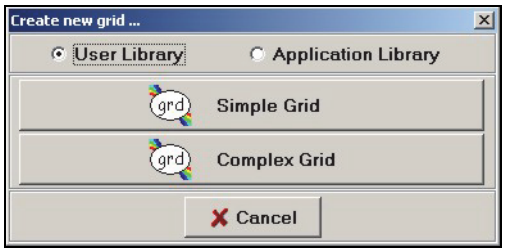
## 17.2 Starting GridEditor

From the TreeView menu, right-click on the ***Grids*** node and choose the ***simple grid*** or ***complex grid*** option.

Double-click on the «Grd.» icon from Button Bar

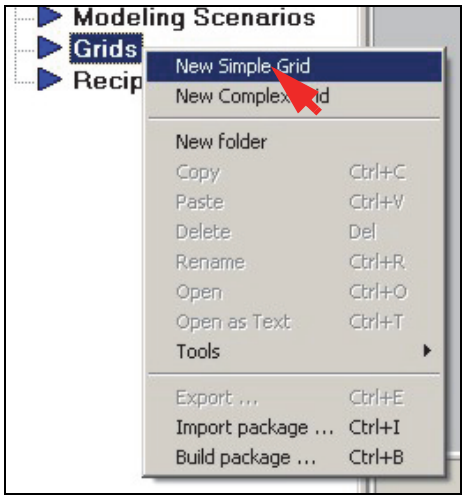

Then select the «Grid» type from the Selection screen:

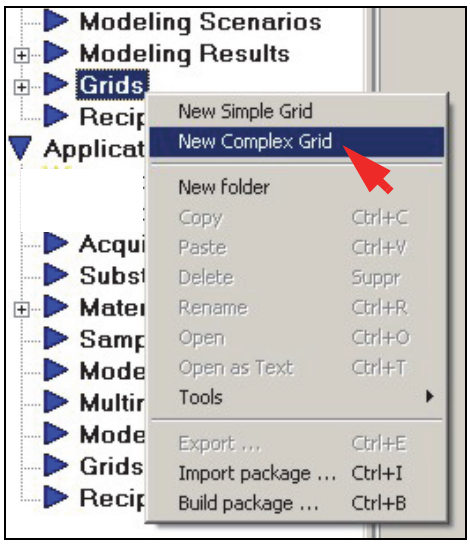
See Using Simple Grid chapter page 226




See Using Complex Grid chapter page 244

OR




05/20/2016

## 17.3 Using Simple Grid

Once the ***Simple Grid*** choice is selected, the following screen will be displayed:

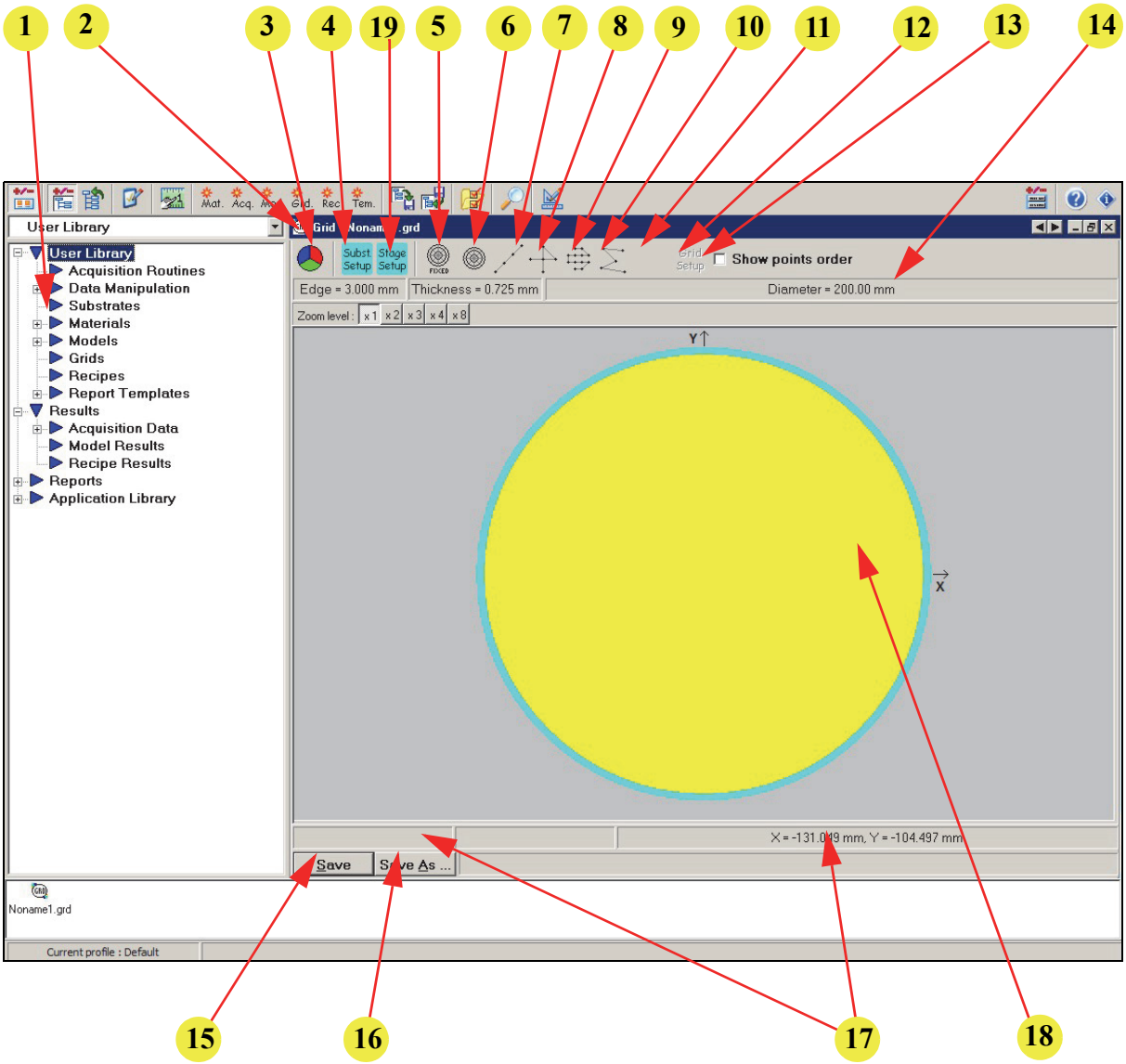



**Figure 17-1 Simple Grid start-up screen**

**1** **TreeView area**

**2** **Current grid name**

**10** **Manual grid**

**11** **Defined grid using Pattern Recognition**

**- This feature is not yet implemented -**

*05/20/2016*

3 **Color grid setup**
4 **Substrate setup**
5 **Fixed Circular grid**
6 **Customized Circular grid**
7 **linear grid**
8 **Cross grid**
9 **Cartesian array grid**
12 **Grid setup**
13 **Points numbering activation**
14 **Substrate information bar**
15 **Save current grid**
16 **Save current grid using a new name**
17 **measurement points information bar**
18 **Defined grid area**
19 **Grid Setup**

### Color Grid Setup

Click on the  button to display the following color grid setup:

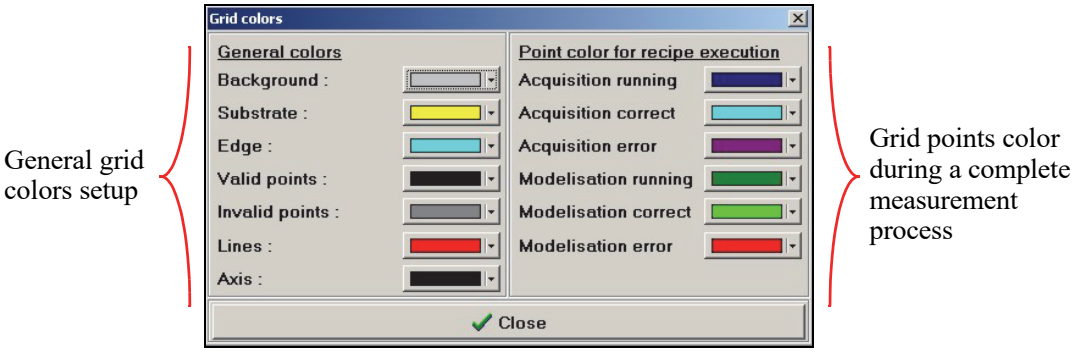



**Figure 17-2 Color Grid Setup screen**

For each of the listed parameters, a specific color can be chosen. Click on the arrow button to select a color from a large range of colors. The limit depends on your graphics card.

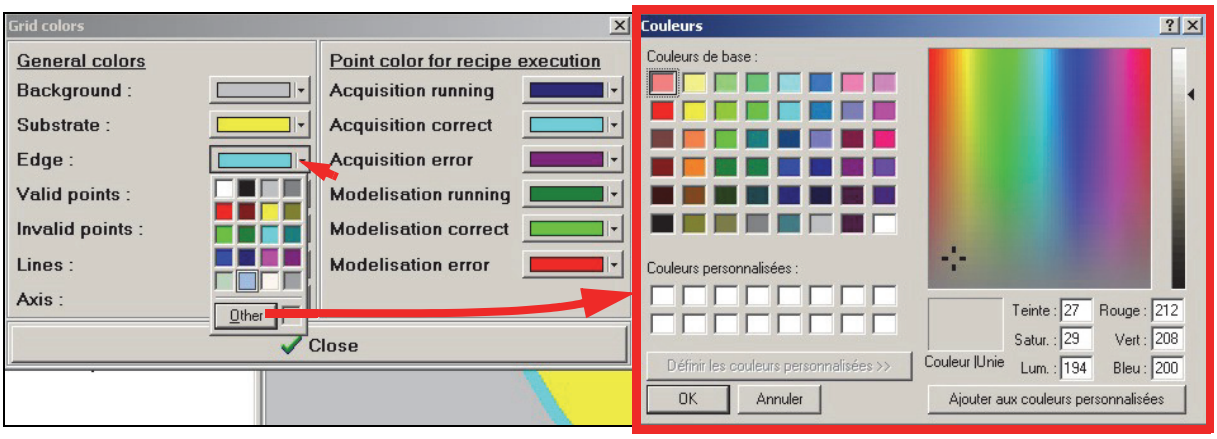
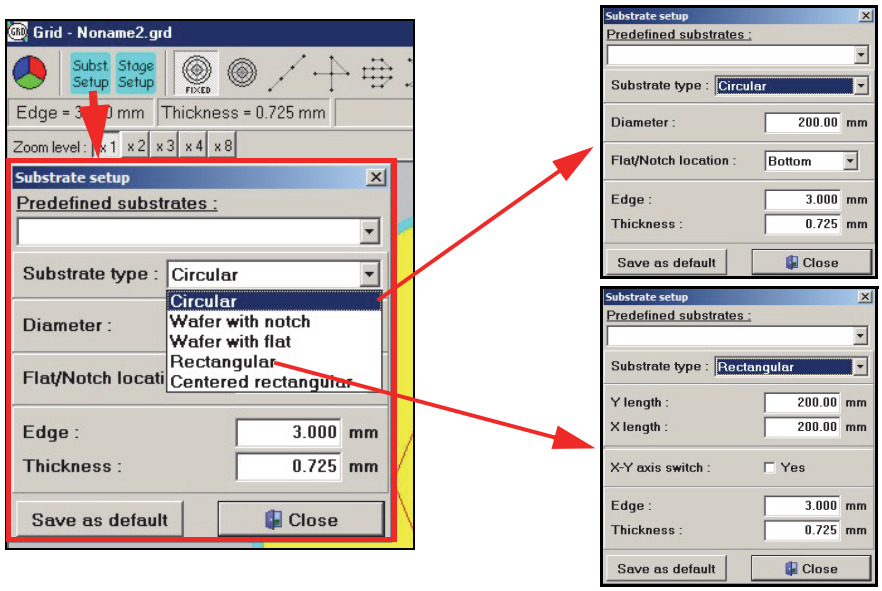



**Figure 17-3 Colors range**

### Substrate Setup

This is, in fact, the first step which must be performed. Clicking on the Subst Setup icon will display the following screen:




- **Predefined Substrates**

  Clicking on the arrow button will activate a menu including predefined substrates shapes. If your substrate shape is not listed, fill the parameters fields described below. If you will work with the same types of substrate, read the paragraph "How to build a pre-defined substrate", page 242 to create your own predefined substrates formats.

NOTICE: If ***Rectangular*** substrate is chosen, X and Y axis can be switch. This could be useful for some specific analysis (for example LCD screens)

05/20/2016

- **Diameter or Width/Height**

  Enter the diameter if a circular substrate has been chosen or enter the width and height size for a rectangular substrate.

- **Flat/Notch location** (if applicable)

  The flat or the notch location determines the substrate orientation in DeltaPsi2 according to the substrate on the XY Stage.

- **Edge**

  The Edge is the distance between the physical and analysis external limits of the substrate.

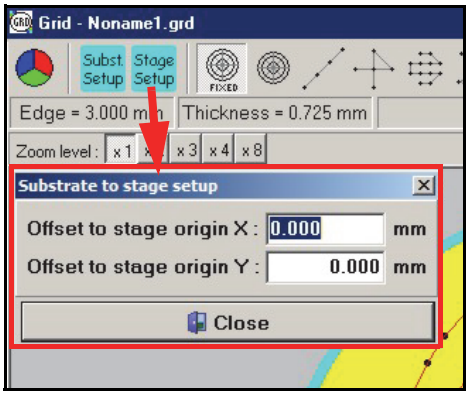
- **Thickness**

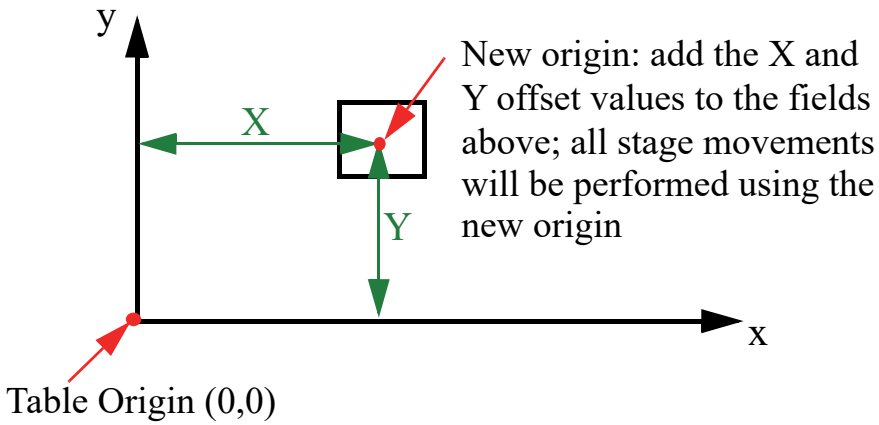
  This is only an information parameter. Enter the substrate thickness. This value will be recorded with the saved grid.

---

**Stage Setup**

The Stage Setup settings allows the user to define a new origin for a motorized Sample Stage. This is very useful when a non standard sample is used.



### Fixed Circular Grid

The fixed circular grid generates circular concentric measurement points. The user can determine all of the parameters excepting the distance between the concentric circles which is constant and calculated from the entered parameters.

Another specification for this grid concern the points distribution: all of the points are located on radii.

Click on the  icon to display the setup screen:



**Figure 17-4 Fixed Circular Grid**

#### 1 Grid X offset - Grid Y offset

As shown above, the complete grid can be shifted by a constant ordinate (x, y) value.

Enter these values in the appropriated fields.

#### 2 Points number

Choose the number of points to distribute on the grid. Please note that the choice is limited because the points must be located on radii.

### 3 Diameter

Enter the total diameter of the grid.

### 4 Angle

The ***Angle*** parameter activates a rotation around the center of the grid. Enter the rotation value in degree. The + and - are accepted.

### 5 Close

Validate the entered parameters.

### 6 Information tab

The information tab displays the information relating to the grid:

- number of validated measurement points
- number of out-grid points,
- real time mouse ordinates.

Once the grid has been setup, click on ***Save As*** (or ***Save*** if the grid has been already named) to store the grid.

NOTICE: To modify the grid, click on the 

icon to display the parameters screen.

### Customized Circular Grid

The Customized Circular Grid allows the user to determine completely a circular distribution of the measurement points. This option is more flexible than the Fixed Circular Grid seen above. Click on the  icon to display the setup screen.

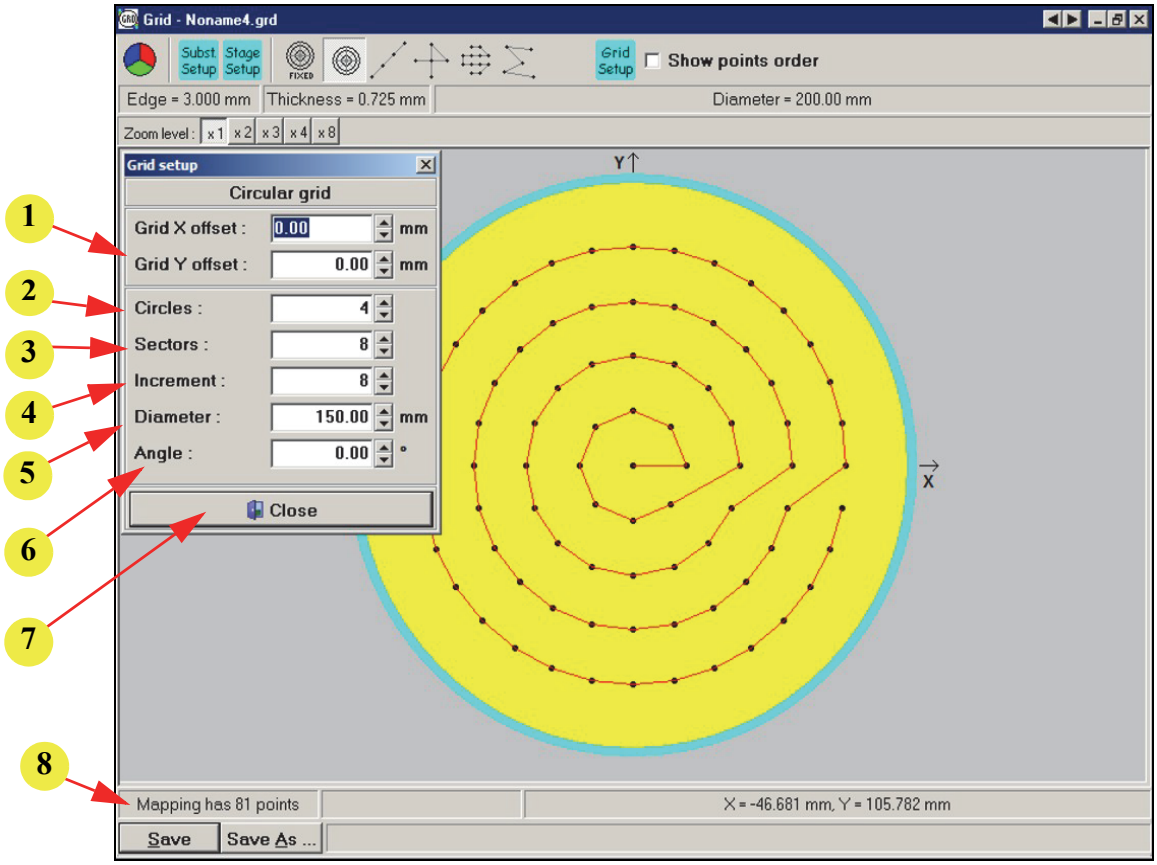



**Figure 17-5 Customized Circular Grid**

#### 1 Grid X offset - Grid Y offset

As shown above, the complete grid can be shifted by a constant ordinate (x, y) value.

Enter these values in the appropriated fields.

#### 2 Circles

Enter the number of circles to set. The first measurement point is the center of the grid.
NB: This point is of course not considered as the first circle.

#### 3 Sectors

The number of sectors determined here is the number of sectors of the **first circle**.

05/20/2016

**4 Increment**

The Increment parameter set the added measurement points for each additional circle. For example, on the figure above, the first circle has 8 sectors (8 measurement points), the second circle has 8 (sectors) + 8 (Increment) = 16 sectors (16 measurement points). The third circle has 16 (sectors of previous circle) + 8 (Increment) = 24 sectors. The same calculation is applied for the other circles.

**5 Diameter**

Enter the total diameter of the grid

**6 Angle**

The ***Angle*** parameter activates a rotation around the center of the grid. Enter the rotation value in degree. The + and - are accepted. (see the figure 17-4)

**7 Close**

Validate the entered parameters.

**8 Information tab**

The information tab displays the information relating to the grid:

- number of validated measurement points
- number of out-grid points,
- real time mouse ordinates.

Once the grid has been setup, click on ***Save As*** (or ***Save*** if the grid has been already named) to store the grid.

NOTICE: To modify the grid, click on the 

icon to display the parameters screen.

---

**Line Grid**

The ***Line Grid*** allows the user to build a grid on which all of the measurement points are located on the same segment of line.

Click on the  icon to display the following setup screen:

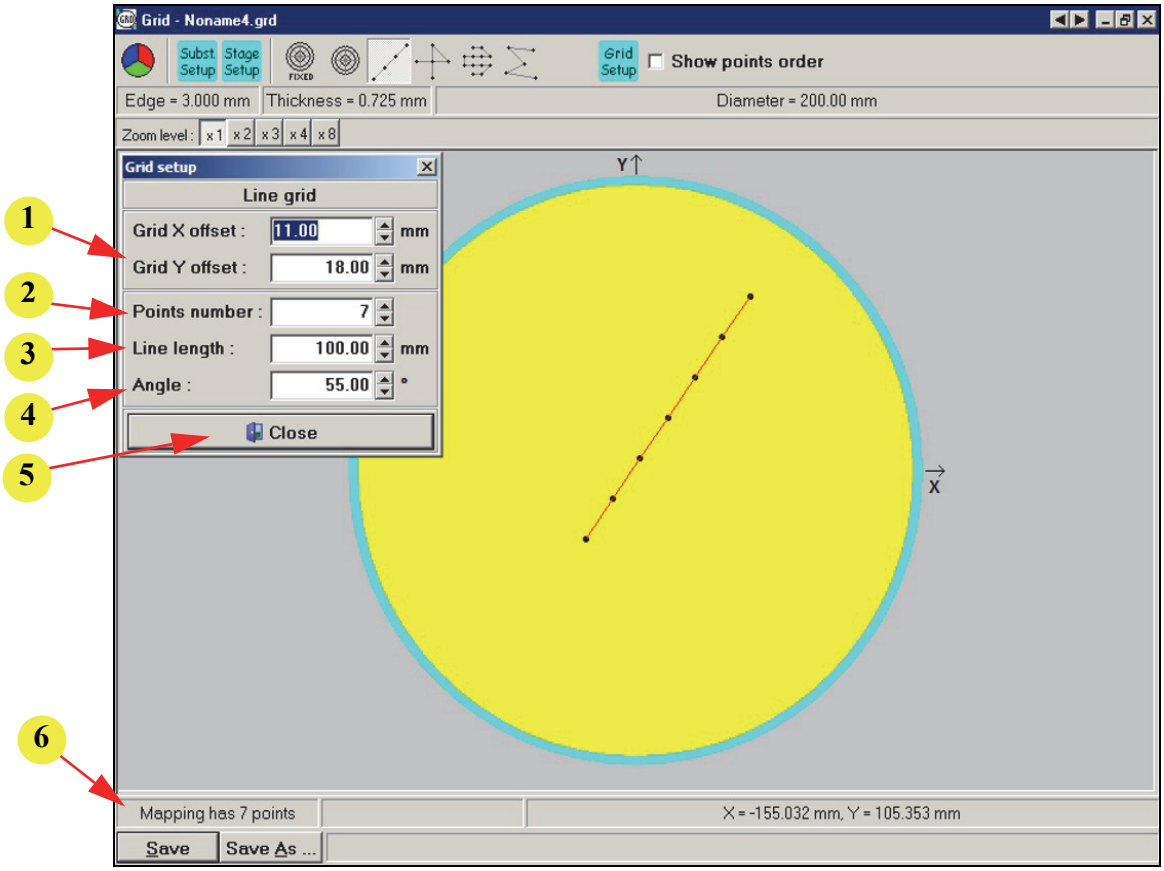



**Figure 17-6 Line Grid**

### 1 Grid X offset - Grid Y offset

As shown above, the complete grid can be shifted by a constant ordinate (x, y) value.

Enter these values in the appropriated fields.

### 2 Points number

Choose the number of points to distribute on the line grid.

### 3 Line Length

Enter the total length of the line grid.

### 4 Angle

The ***Angle*** parameter activates a rotation around the center of the line grid. Enter the rotation value in degree. The + and - is accepted.

### 5 Close

Validate the entered parameters.

### 6 Information tab

The information tab displays the information relating to the grid:

- number of validated measurement points
- number of out-grid points,
- real time mouse ordinates.

Once the grid has been setup, click on ***Save As*** (or ***Save*** if the grid has been already named) to store the grid.

NOTICE: To modify the grid, click on the Grid Setup icon to display the parameters screen.

---

## Cross Grid

The Cross Grid allows the user to build another specific grid pattern. This is an extension of the line grid in the sense that two perpendicular segments of lines are created and each of the segment can be set separately.

Click on the icon to display the following setup screen:

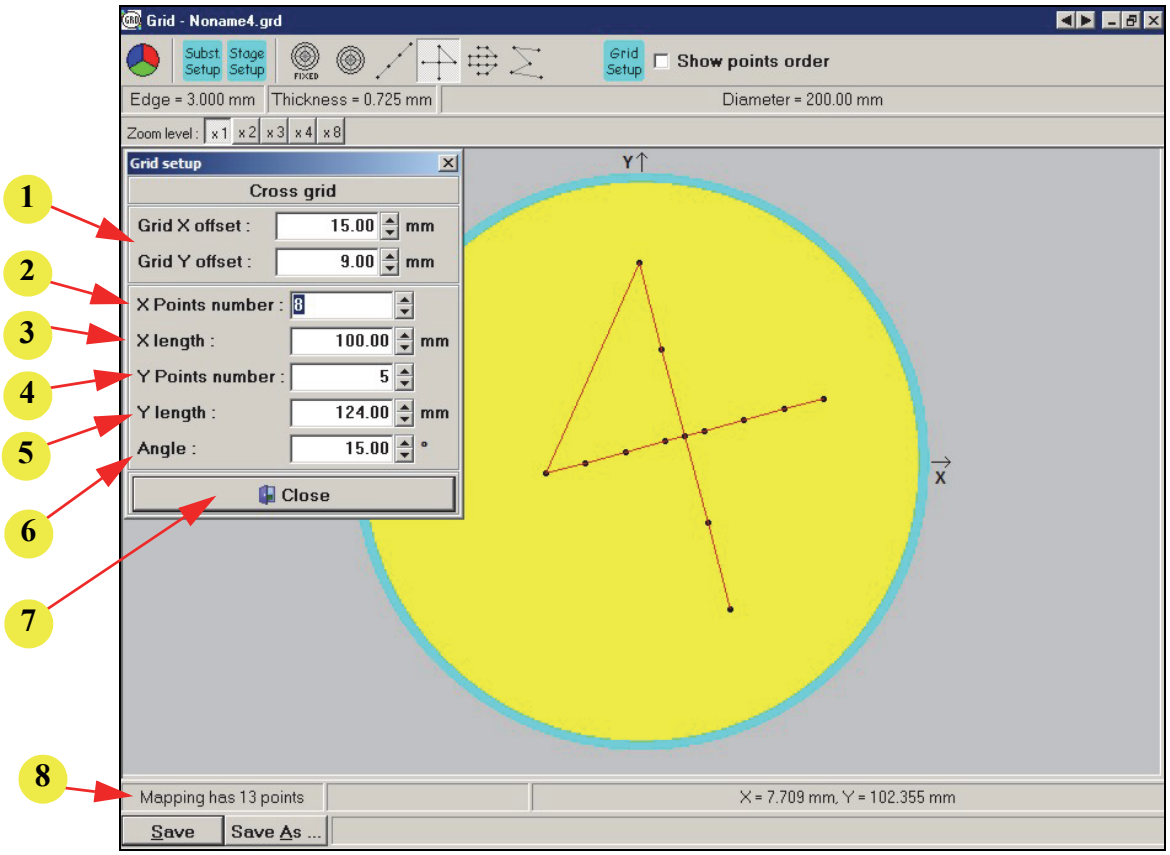



**Figure 17-7 Cross Grid**

### 1 Grid X offset - Grid Y offset

As shown above, the complete grid can be shifted by a constant ordinate (x, y) value.

Enter these values in the appropriated fields.

**2 X Points number**

Enter the number of measurement points to set on the X axis of the *Cross Grid*.

**3 X Length**

Enter the total length of the X axis segment of line.

**4 Y Points number**

Enter the number of measurement points to set on the Y axis of the *Cross Grid*.

**5 Y Length**

Enter the total length of the Y axis segment of line.

**6 Angle**

The ***Angle*** parameter activates a rotation around the center of the Cross Grid. Enter the rotation value in degree. The + and - are accepted. (see the figure 17-4)

**7 Close**

Validate the entered parameters.

**8 Information tab**

The information tab displays the information relating to the grid:

- number of validated measurement points
- number of out-grid points,
- real time mouse ordinates.

Once the grid has been setup, click on ***Save As*** (or ***Save*** if the grid has been already named) to store the grid.

NOTICE: To modify the grid, click on the Grid Setup icon to display the parameters screen.

---

**Cartesian Array Grid**

The Cartesian Array Grid has been developed to regularly distribute the measurement points on a substrate or a part of substrate. The grid can exceed the substrate without any problem: the software automatically adapts the grid to the substrate limits. The information bar explains the number valid measurement points and those which are outside the substrate.

Click on the icon to display the following setup screen:

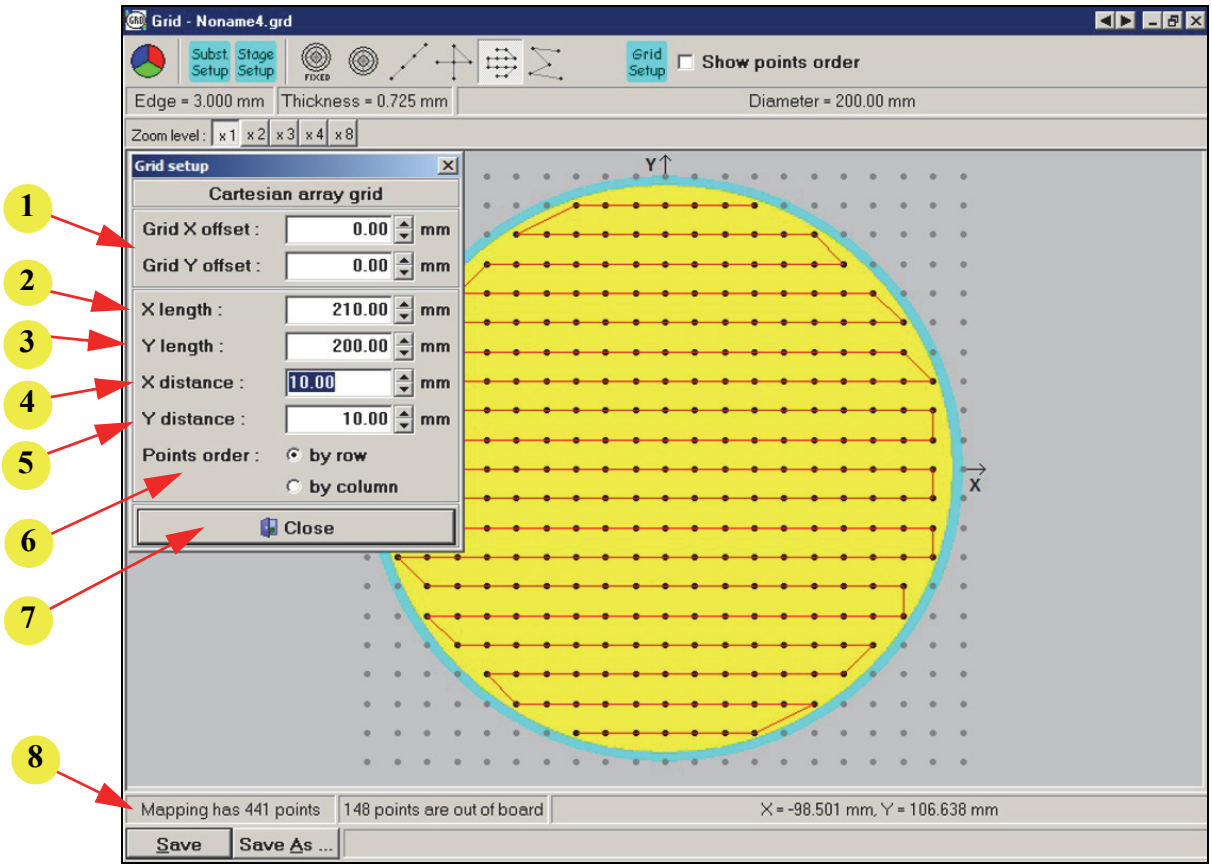



**Figure 17-8 Cartesian Array Grid**

### 1 Grid X offset - Grid Y offset

The complete grid can be shifted by a constant ordinate (x, y) value.

Enter these values in the appropriated fields.

### 2 X length

As the points distribution is rectangular, enter the total ***X length*** value of the grid.

### 3 Y length

Enter the total ***Y length*** value of the grid

### 4 X distance

***X distance*** is the distance between each measurement on the X axis.

### 5 Y distance

***Y distance*** is the distance between each measurement on the Y axis.

### 6 Points order

The ***points order*** parameter activates a row or a column points distribution.

### 7 Close

Validate the entered parameters.

### 8 Information tab

The information tab displays the information relating to the grid:

- number of validated measurement points
- number of out-grid points,
- real time mouse ordinates.

Once the grid has been setup, click on ***Save As*** (or ***Save*** if the grid has been already named) to store the grid.

NOTICE: To modify the grid, click on the Grid Setup icon to display the parameters screen.

---

**Manual Grid**

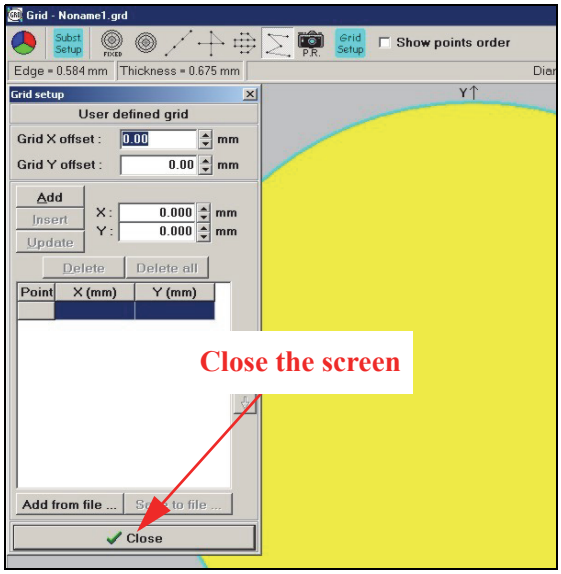
The Manual Grid is the most versatile grid building procedure. Each measurement point can be determined manually.

Two ways are offered to build the grid; the first method positions the points using the mouse, the second method uses a table where each point is set manually.

Click on the icon to display the setup screen.

- **Building Grid using the mouse**

  First, close the parameters screen displayed over the main screen:




A grid can be then built manually using the mouse. The figure 17-9 shows how to manipulate the mouse to create a grid.

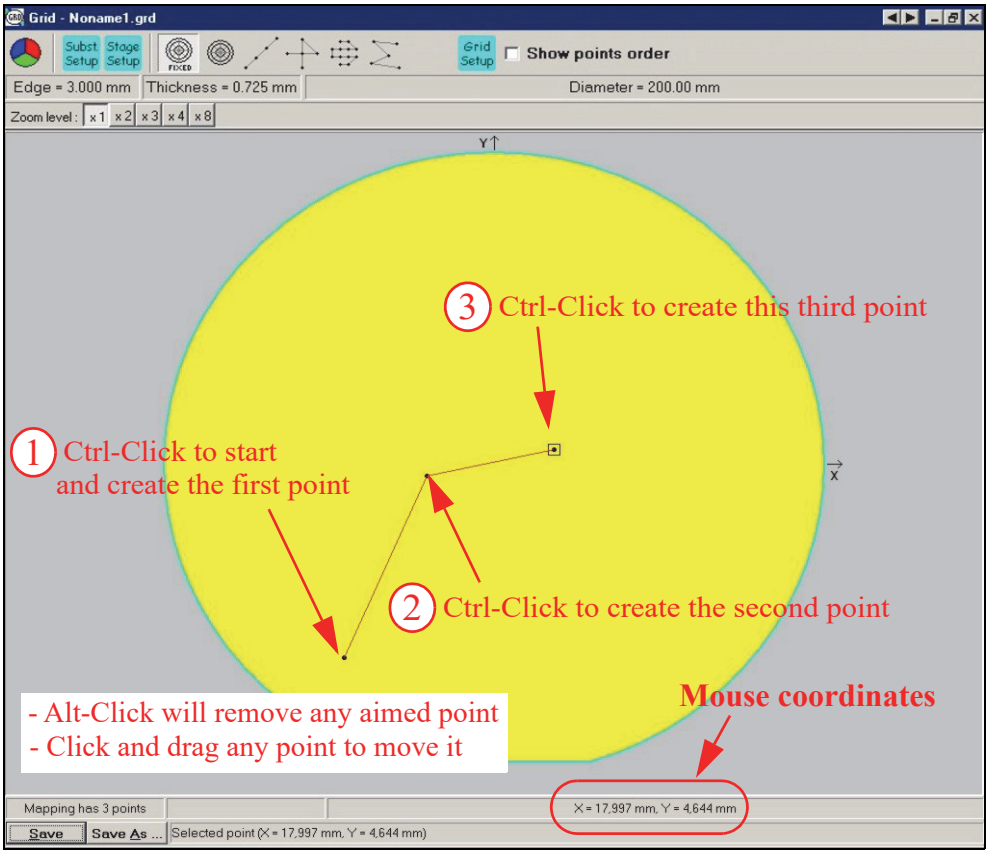



**Figure 17-9 Build Grid using a mouse**

NOTICE: The grid can also be modified using coordinates listed in a table. To open this feature, click on the Grid Setup icon to display the parameters screen. Next paragraph explains how to use the table.

- **Building Grid using the table**

  This procedure uses the table displayed once the ***Manual Grid*** is launched. This table lists the points coordinates and the grid offset. A measurement point can be added, deleted, moved from this table. The procedure below describes how to create a grid.

**1** Grid X offset - Grid Y offset

The complete grid can be shifted by a constant ordinate (x, y) value.

Enter the offset values in the appropriated fields ( **1** ).

If the substrate is blank, the ***X*** and ***Y*** fields ( **5** ) display the (0.000, 0.000) coordinates which correspond to the center of the grid which is the point of reference. To enter the first measurement point, enter the coordinate in the ***X*** and ***Y*** fields then press on Add **2** button to validate. The same procedure can be followed to enter any points as necessary for the analysis.

If a point needs to be modified, just highlighted **6** the coordinate with the mouse, then enter the new values in the *X* and *Y* fields. Click on *Update* button **4** to validate the changes.

To insert a measurement point, highlight the underneath point then enter the new coordinate values then click on the ***Insert*** button.

The figure below shows an example of Manual Grid editing layout.

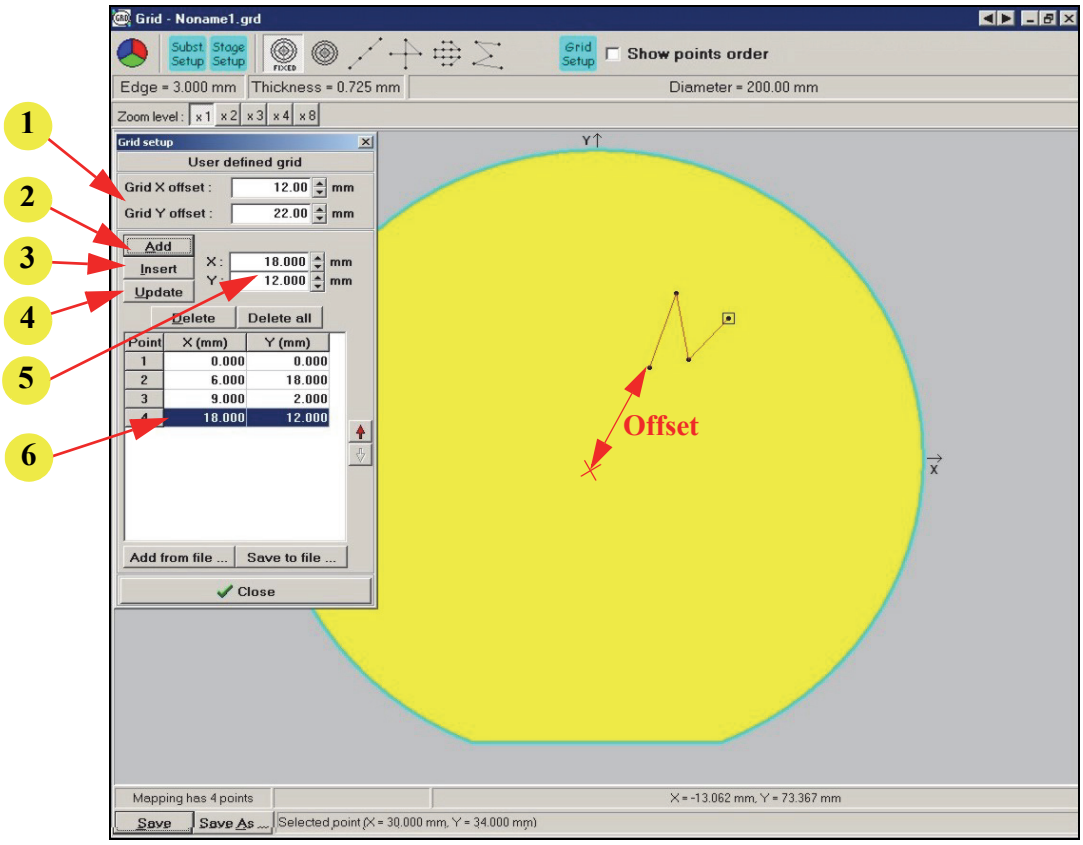


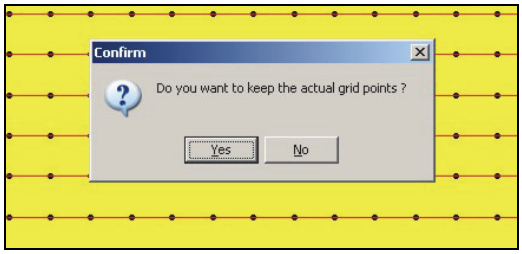

**Figure 17-10 Manual Grid using table**

### Modifying an existing grid

Any grid which has been created can be modified using the following procedure:

- From the ***Grids*** node located in the ***TreeView*** (***User Library***), double-click on the grid file which must be modified. The grid is automatically displayed.

- Click on the  Manual Grid icon. A pop-up message is then displayed: click on YES button to keep the grid (see the screen below).



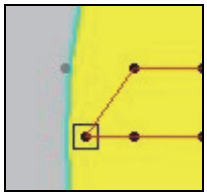
- **Method using the mouse**

  Close first the parameters screen.

  Using the mouse, it becomes easy to modify any points of the grid: just click on the point and drag it on the area. The cursor coordinates are displayed on the information bar.

  This method is so flexible that a point which is outside the substrate can be moved inside. This is useful to validate a point located very close to substrate.

  The following figure shows an example:



- **Method using the parameters screen**

  Here below is an example of a previously created grid. As shown below, the parameters screen includes a table which lists the coordinates of all valid points of the substrate.

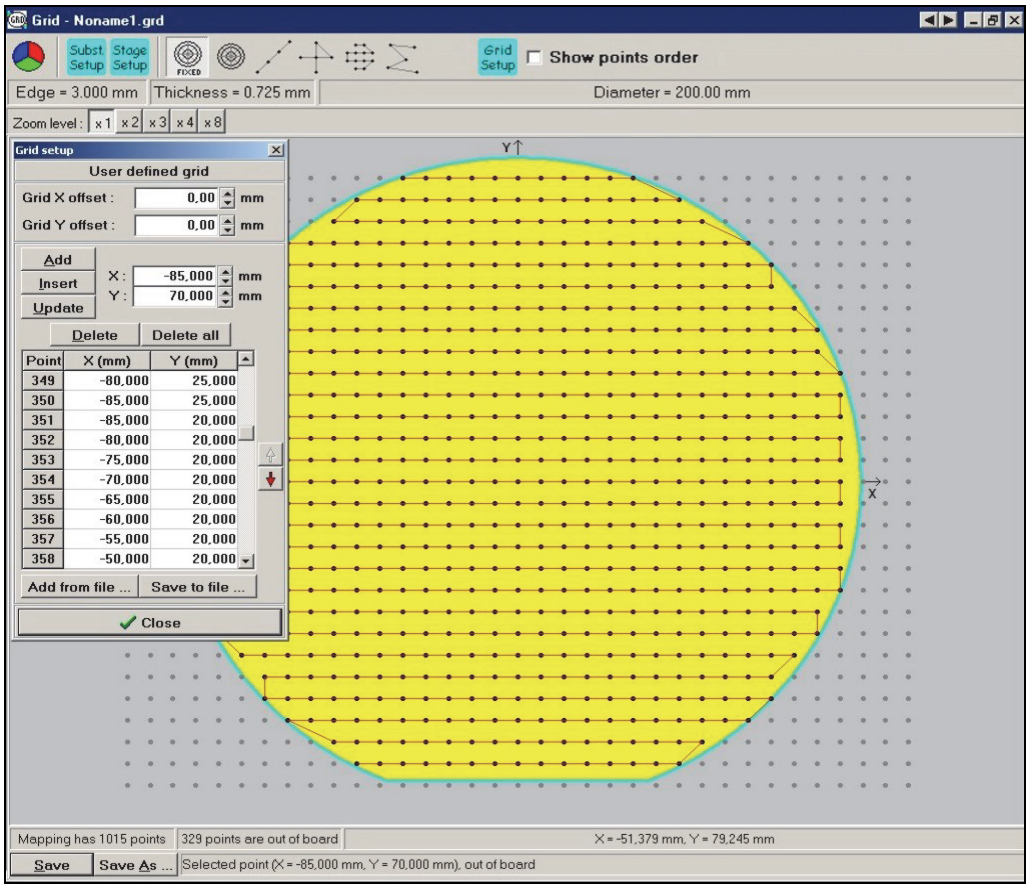
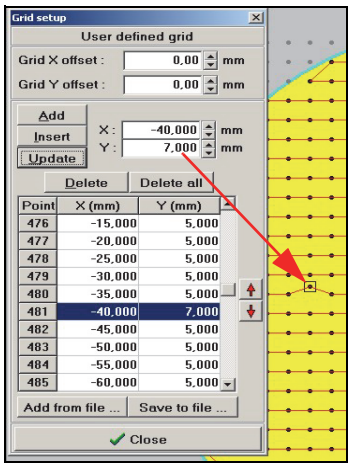



**Figure 17-11**

Using the mouse, if a coordinate is selected on the table (parameters screen), the related point on the substrate will be immediately highlighted.

Once the point to modify is found, enter the new coordinate in the fields labeled ***X*** and ***Y***, then press the ***Update*** button. The new coordinates of the point are now modified and listed in the table. The same procedure can be then performed for any other point.




As shown on the figure nearby, the same procedure can be used to insert a new point.

The ***Delete*** and ***Delete All*** button is easy to use to delete a point or delete all the grid.

The ***Save to file...*** feature allow the user to save the coordinate points to **.SCV** format (Microsoft Excel).

The red Up/Down Arrow located near to the table are used to move a point inside the table. It is then possible to modify the analysis priority order.

The ***Add from files...*** feature is useful to load a grid which has been previously prepared from Microsoft Excel or any simple text editor like notepad. Using a text editor, it is mandatory to keep the syntax and modify the extension to **.SCV**.

Here below is an example of the syntax to use:

-50.000,50.000
-40.000,50.000
-30.000,50.000
-20.000,50.000
-10.000,50.000
etc....

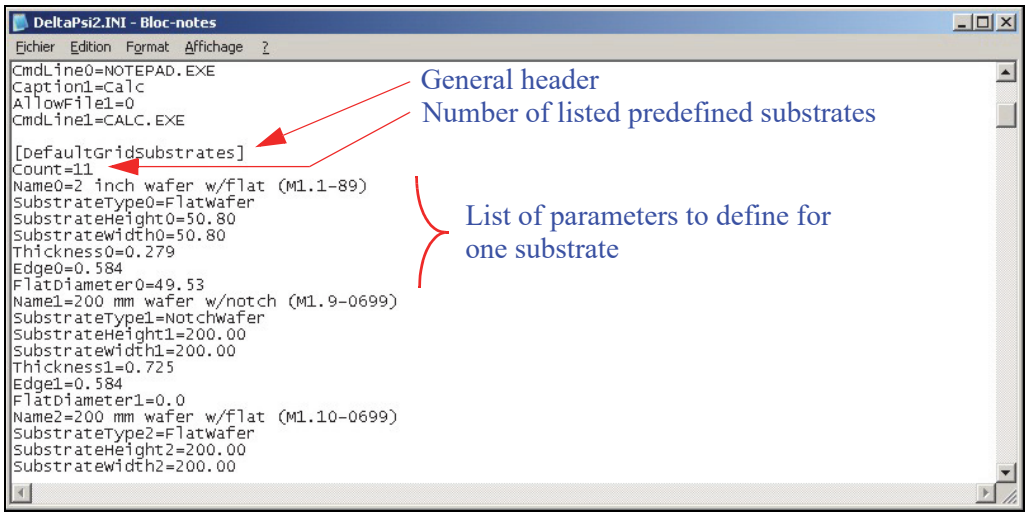
---

**How to build a predefined substrate**

From the Substrate Setup screen, we have seen that a predefined substrate can be selected. In this chapter we will learn how to create your own predefined substrate.

**Caution: do not change any parameters excepting those defined in the procedure below**.

- Exit the DeltaPsi2 software (if still running).
- Run the Windows® Explorer program (delivered with the Windows® operating system) and double-click on the DeltaPsi2.ini file located in the DeltaPsi2 folder. The DeltaPsi2 configuration file is displayed in edit mode.
- In the configuration list, find the header `[DefaultGridSubstrates],`
- Next line the parameter `Count=11` is displayed. The value (here 11) is the total number of predefined substrates which will be listed. Once the appropriate value has been entered, all of the predefined substrates will be edited.




- On the screen displayed above, it is easy to determine the syntax used to define the substrate parameters.

  The required syntaxes are listed below:

  Count=[number of listed predefined substrate]

  **NameN=A [unit] [user general information]**

  (N is an incremented value from 0 to Count-1)

  (A is the size of the substrate)

  **SubstrateTypeN=FlatWafer** or **NotchWafer** or **Cicular** or **Rectangular**

  (N must be the same as for the Name parameter)

  (Notice: W must be in capital letter)

  **SubstrateHeightN=**[value]

  (N must be the same as for the Name parameter)

  (Value must be in millimeters)

  **SubstrateWidthN=**[value]

  (N must be the same as for the Name parameter)

  (Value must be in millimeters)

  **ThicknessN=**[value]

  (N must be the same as for the Name parameter)

  (Value must be in millimeters)

  **EdgeN=** [value]

  (N must be the same as for the Name parameter)

  (Value must be in millimeters)

  **FlatDiameterN=** [value]

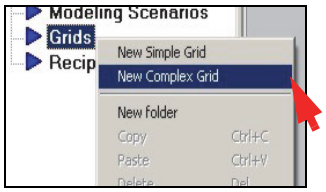
  (N must be the same as for the Name parameter and the value must be set to 0.0 for any substrateType other than FlatWafer)

Once performed, select the ***Save*** choice from the ***File Main Menu***.

*05/20/2016*

## 17.4 Using Complex Grid

From the TreeView menu, right-click on the ***Grids*** node and choose the ***complex grid*** option.




The following screen will be then displayed:

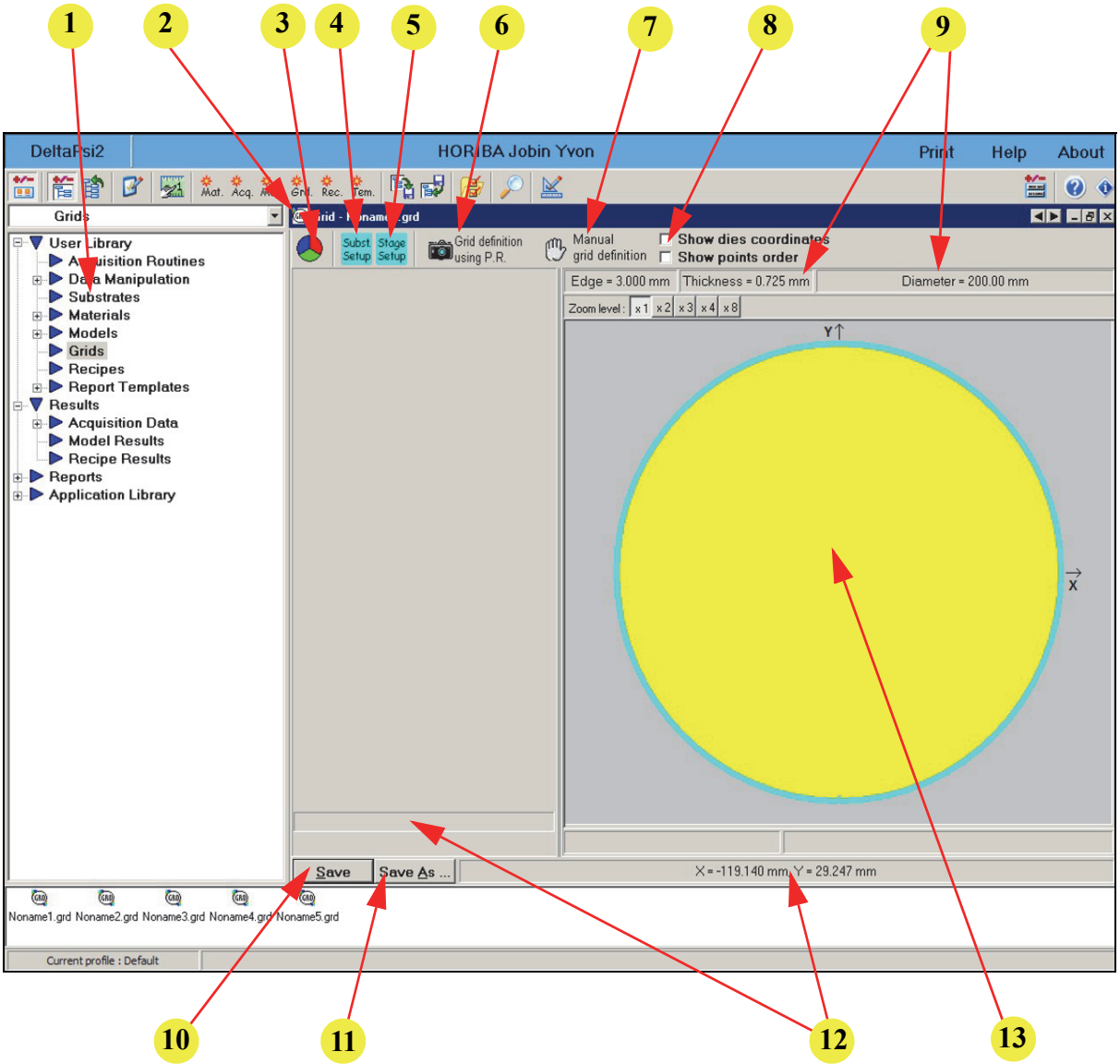



**Figure 17-12 Complex Grid start-up screen**

1. **TreeView area**
2. **Current grid name**
3. **Color grid setup**
4. **Substrate setup**
5. **Stage setup**
6. **Defined grid using Pattern Recognition**

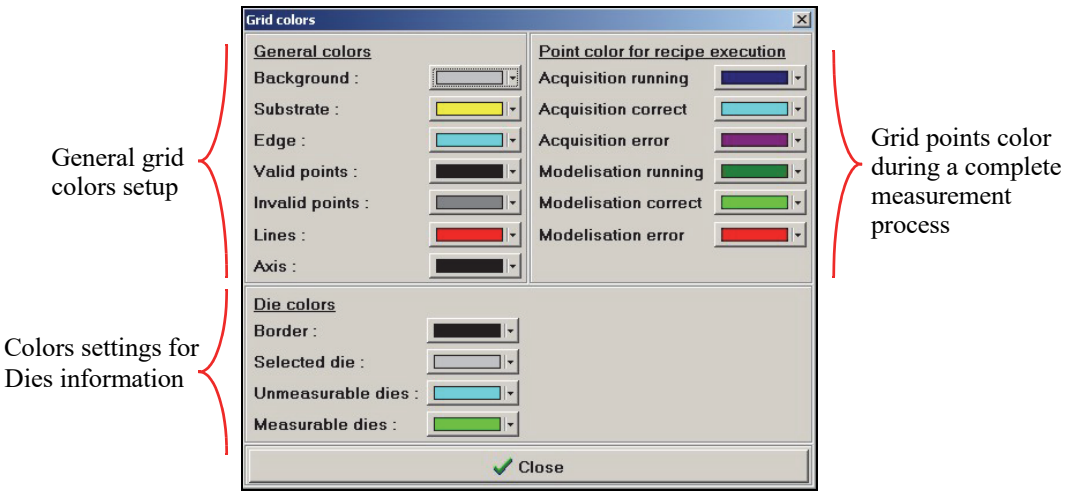
**- This feature is not yet implemented -**

7. **Manual grid definition**
8. **Points numbering and dies coordinates activation**
9. **Substrate information bar**
10. **Save current grid**
11. **Save current grid using a new name**
12. **measurement points information bar**
13. **Defined grid area**

---

### Color Grid Setup

Click on the  button to display the following color grid setup:




For each of the listed parameters, a specific color can be chosen. Click on the arrow button to select a color from a large range of colors. The limit depends on your graphics card.

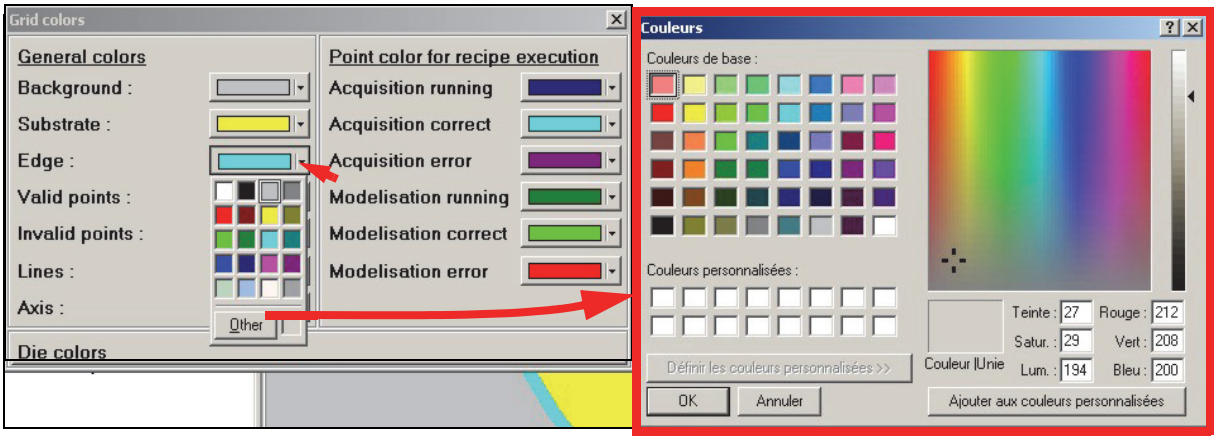
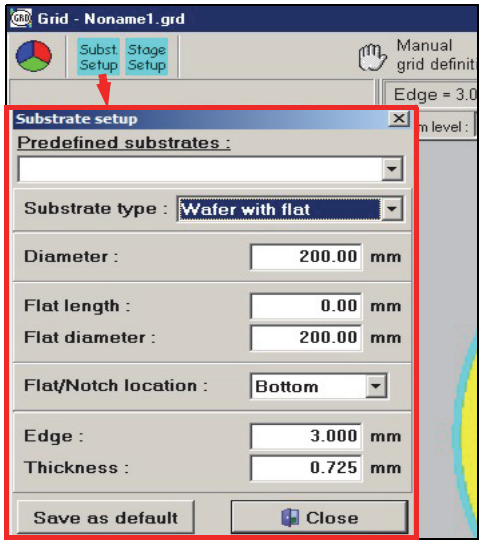



**Figure 17-14 Colors range**

### Substrate Setup

This is, in fact, the first step which must be performed. Clicking on the Subst Setup icon will display the following screen:




- **Predefined Substrates**

  Clicking on the arrow button will activate a menu including predefined substrates shapes. If your substrate shape is not listed, fill the parameters fields described below. If you will work with the same types of substrate, read the paragraph "How to build a pre-defined substrate", page 242 to create your own predefined substrates formats.

- **Diameter or Width/Height**

  Enter the diameter if a circular substrate has been chosen or enter the width and height size for a rectangular substrate.

- **Flat/Notch location** (if applicable)

  The flat or the notch location determines the substrate orientation in DeltaPsi2 according to the substrate on the XY Stage.

- **Edge**

  The Edge is the distance between the physical and analysis external limits of the substrate.

- **Thickness**

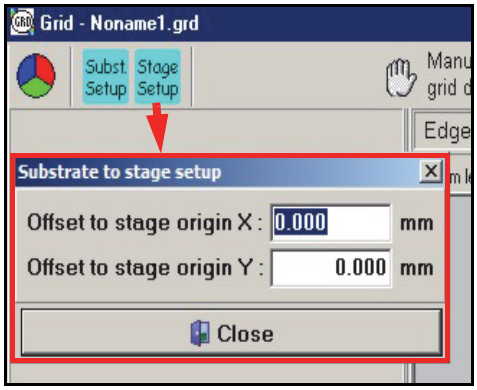
  This is only an information parameter. Enter the substrate thickness. This value will be recorded with the saved grid.

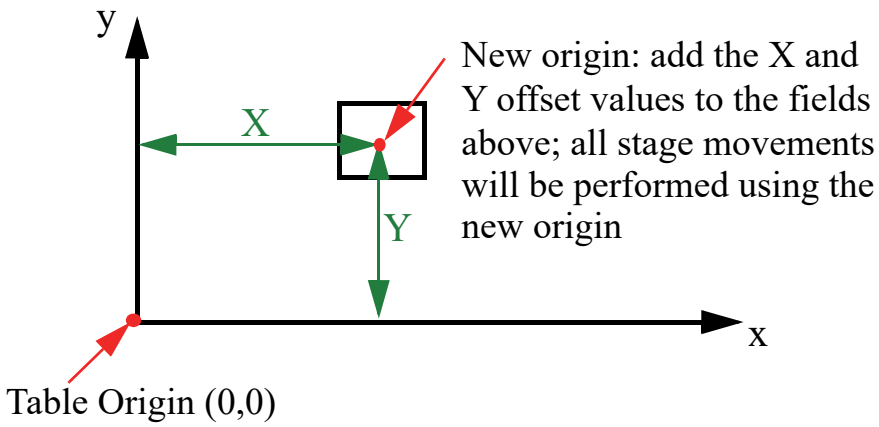
  recorded with the saved grid.

---

**Stage Setup**

The Stage Setup settings allows the user to define a new origin for a motorized Sample Stage. This is very useful when a non standard sample is used.



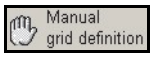




---

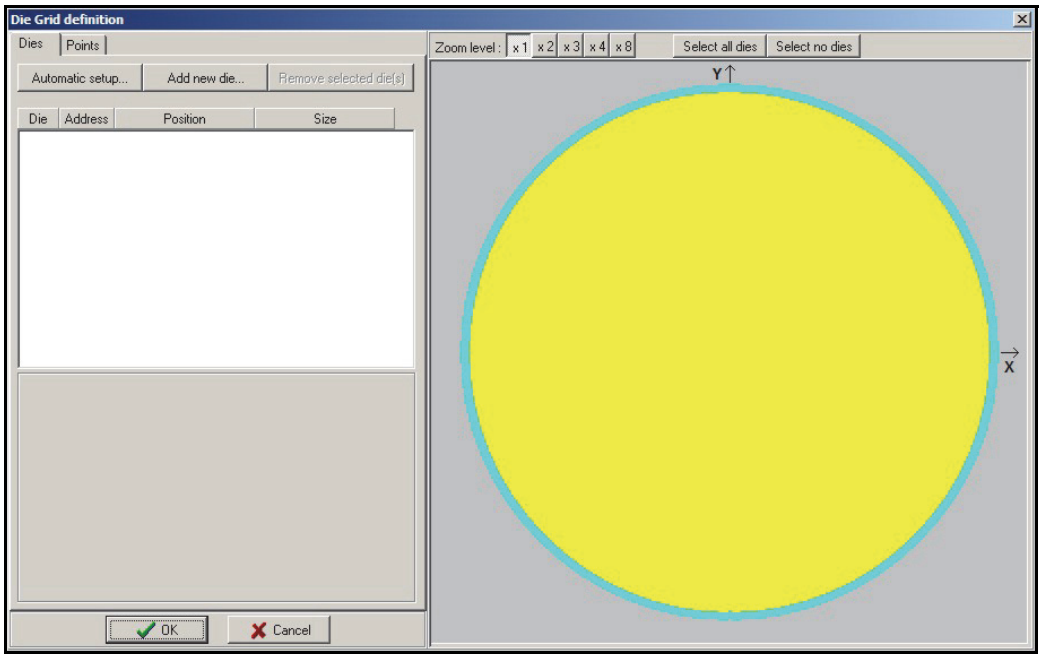
**Grid Definition using Pattern Recognition Feature**

This feature is currently implemented in the Software

### Manual Grid Definition

To set manually the grid on a substrate which contains dies, click on the 

button. The screen shown below is then displayed.

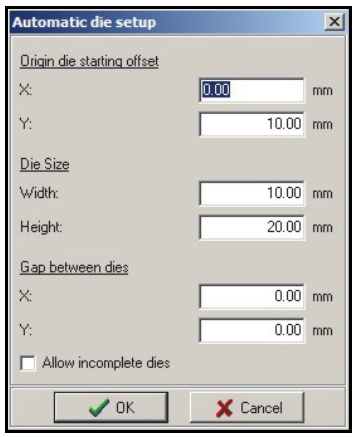



There is 2 choices to create dies on the substrate:

- **Automatic Setup:** The wizard will create automatically the dies according to the sizes entered by the user,
- **Add new die:** the user can add a die everywhere on the blank substrate or on an already dies defined substrate. In this case, the software verify that this new die will not overlap another already on the substrate.

#### ● Automatic Setup

**1** Click on the ***Automatic Setup*** button. The following pop-up screen will be displayed:




**Origin Die starting offset**: the substrate is considered as the reference scale on which dies can be positioned. The Offset coordinate (x, y) fine tunes the dies on the substrate. The defined offset will be always the left bottom corner of the origin die. To define the origin, enter the coordinate according to the xy axis.

**Die Size** (Width and Height): The Size of elementary die must be entered in the Height and Width fields.

**Gap between dies**: enter the X and Y gap between the dies.

**Allow incomplete dies**: on the external edge of the substrate, some dies are incomplete. Check on to see incomplete dies.

Click ***OK*** to validate: the values will be saved for a further

**Notice**: any changes are immediately effective on the graphic representation.

Select the dies on which measurement points will be applied.

Figure 17-15 shows and explain how to select the dies on the substrate.

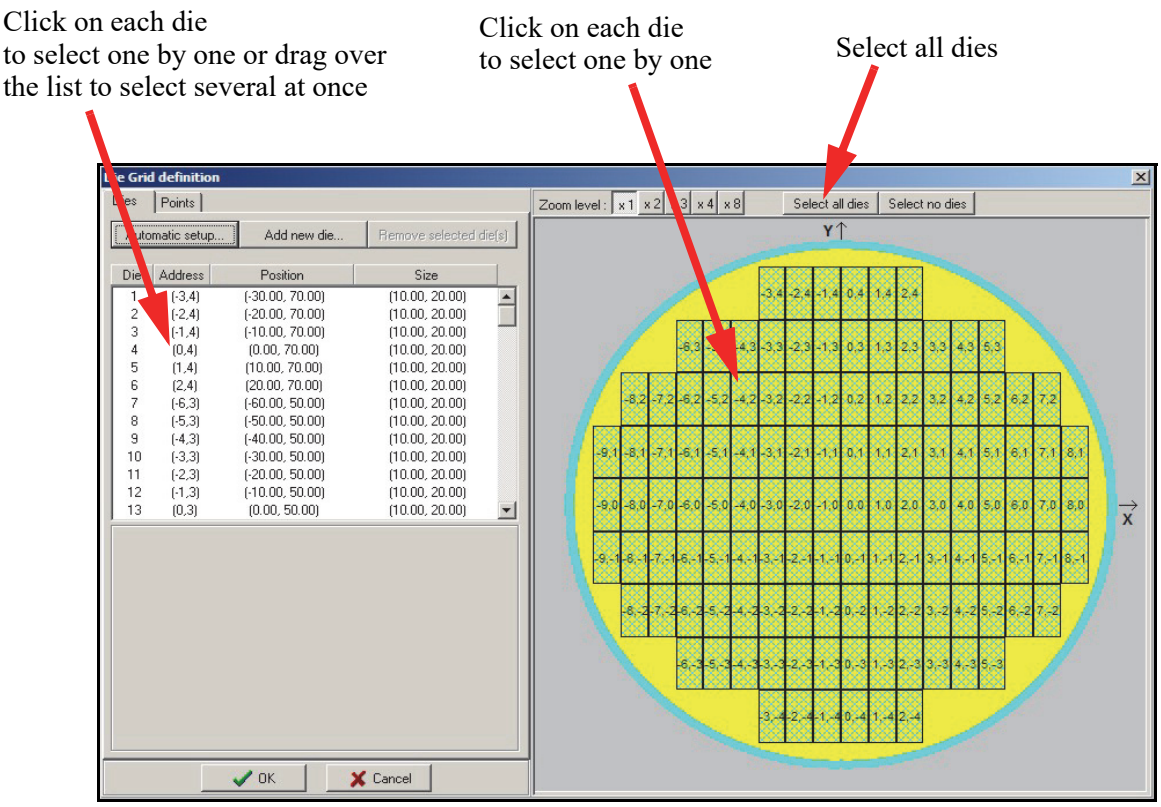



**Figure 17-15 Dies distribution and selection**

Once the dies has been selected, they become active to receive the measurement points.

Click on the ***Points*** tab to display the measurement grid editor on each die.

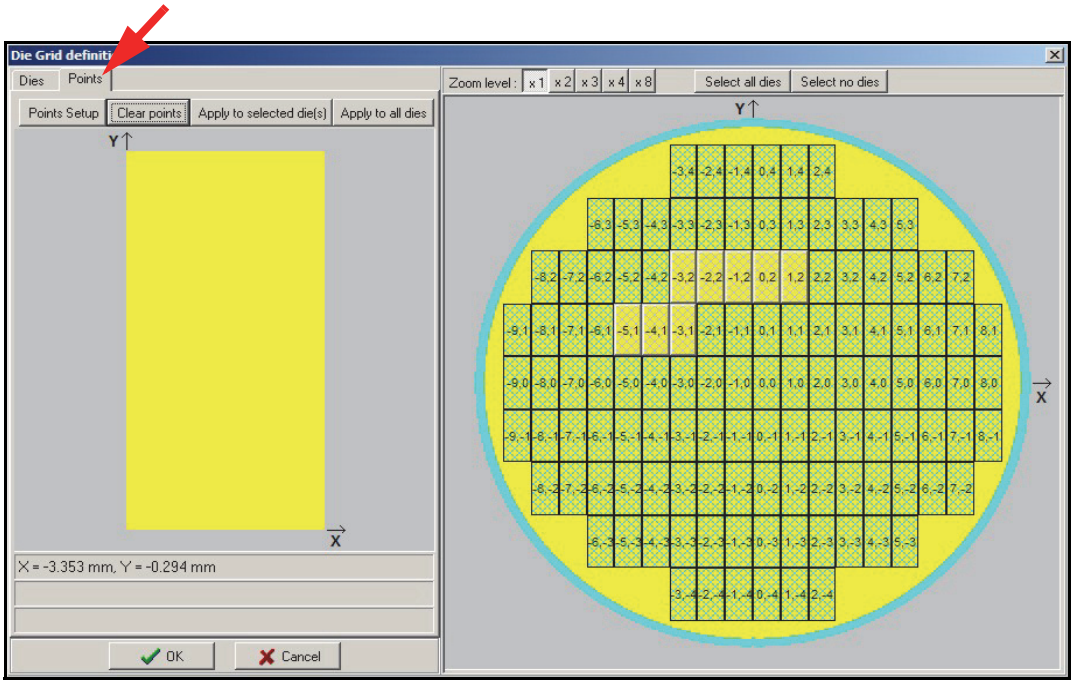
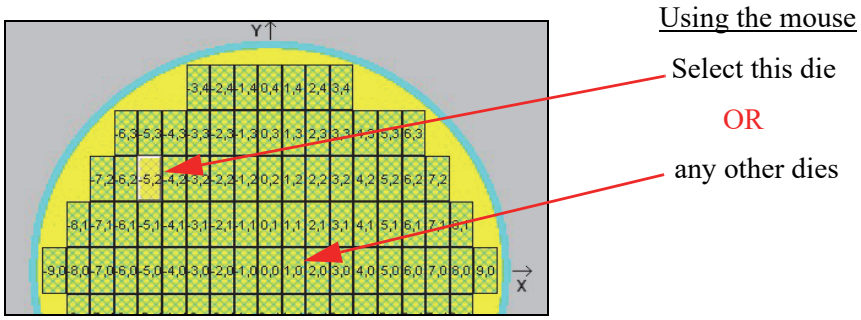



**Figure 17-16 measurement points selection**

05/20/2016

Select or correct the selected active dies. The measurement grid can be now created.




Each die can be setup with a different measurement grid: this action must be done in several step. See

Two ways are offered to build the grid; the first method positions the points using the mouse, the second method uses a table where each point is set manually.

- **Building Grid using the mouse**

  A grid can be built manually using the mouse. The figure 17-17 shows how to manipulate the mouse to create a grid.

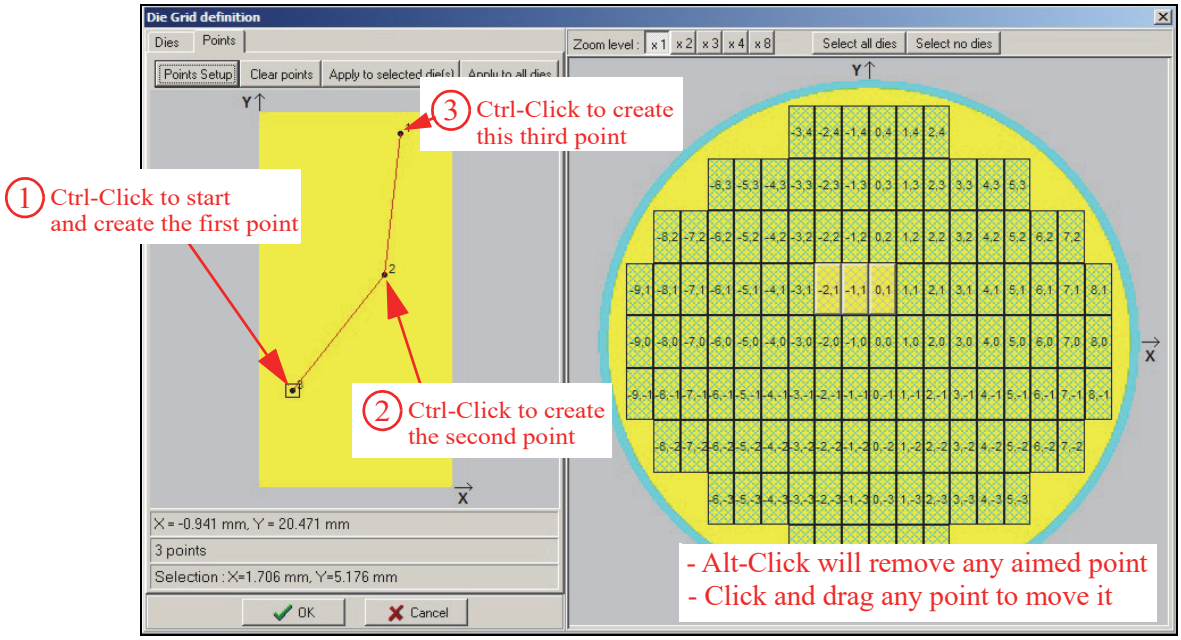



**Figure 17-17 Building a grid**

- **Building Grid using the table**

  Click on the ***Points Setup*** button to display the ***Grid Setup*** table. This table lists the points coordinates. A measurement point can be added, deleted, moved from this table. The procedure below describes how to create a grid.

1 **If the substrate is blank, the X and Y fields display the (0.000, 0.000) coordinates which correspond to the bottom-left corner of the grid which is the point of reference. To enter the first measurement point, enter the coordinate in the X and Y fields then press on the Add button to validate. The same procedure can be followed to enter any points as necessary for the analysis.**

2 **If a point needs to be modified, just highlight the coordinate with the mouse, then enter the new values in the X and Y fields. Click on the Update button to validate the changes.**

3 **To insert a measurement point, highlight the underneath point then enter the new coordinate values then click on the Insert button.**

The figure below shows an example of Manual Grid editing layout.

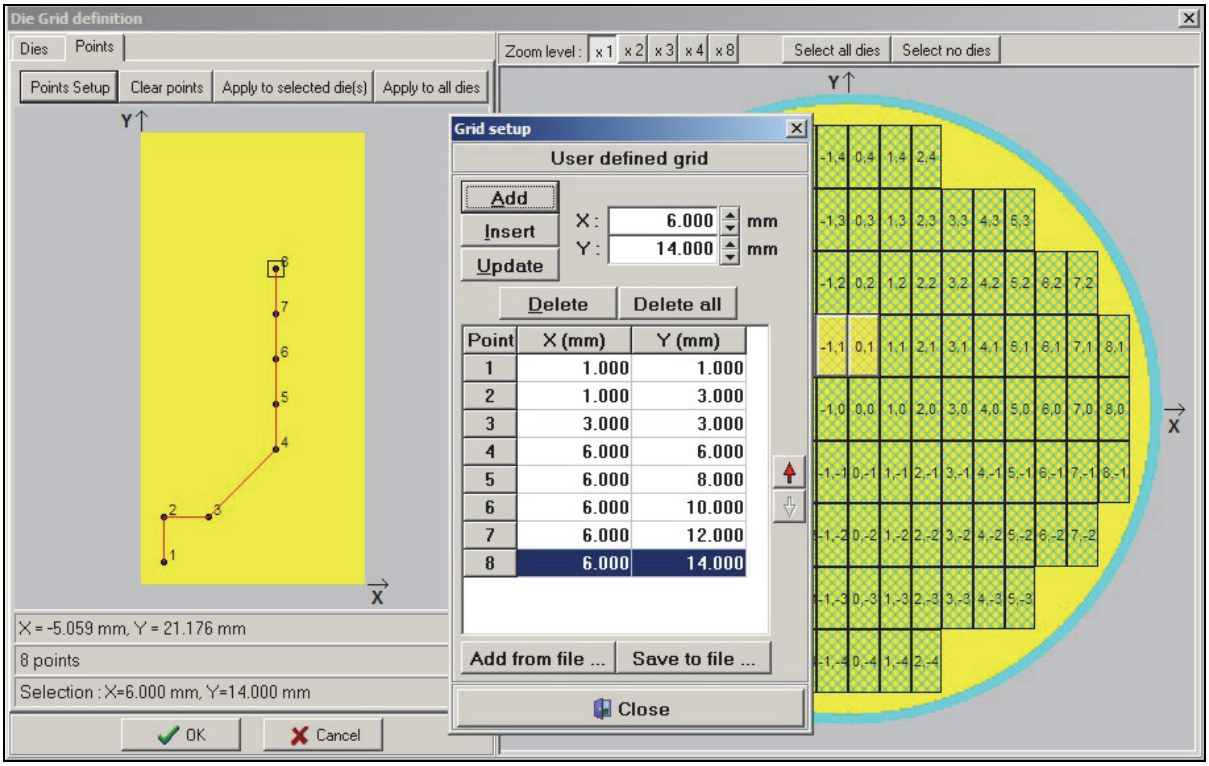



**Figure 17-18 Die grid manual editing**

- **Results preview**

  Click on the ***Apply to die*** or ***Apply to All*** button to visualize the results on the substrate layout (see figure 17-19).

1 Once the measurement points and the dies grid have been correctly set, press the ***OK*** button to validate and exit the setting procedure. The screen displays the created grid.

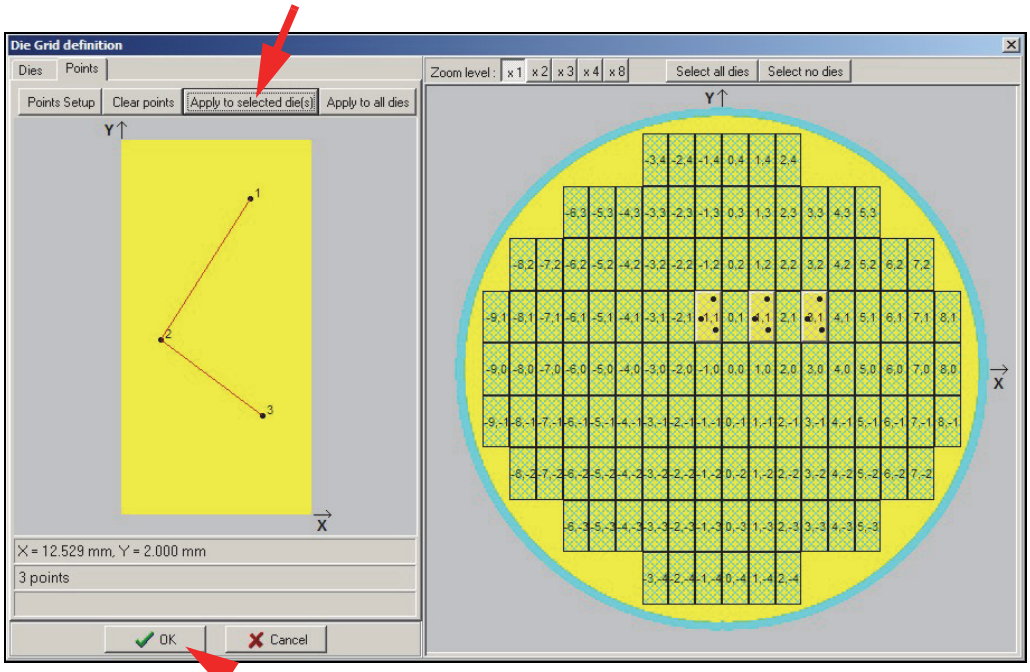
Once the grid has been setup, click on ***Save As*** (or ***Save*** if the grid has been already named) to store the grid.

NOTICE 1: The grid defined on each die can be different from another.

NOTICE 2: a saved grid can be edited later, for example to change dies points.

05/20/2016

**Do not forget to apply the grid to the selected dies**




**Validate the measurement grid on the selected dies**

**Figure 17-19 Dies grid results - example 1**

On the same substrate, each die can be link to a new measurement grid. Just repeat the procedure from step 2: "Select the dies on which measurement points will be applied." on page 249. On the example below different measurement grids have been assigned to dies.

**On this example, some dies receive different measurement grids**

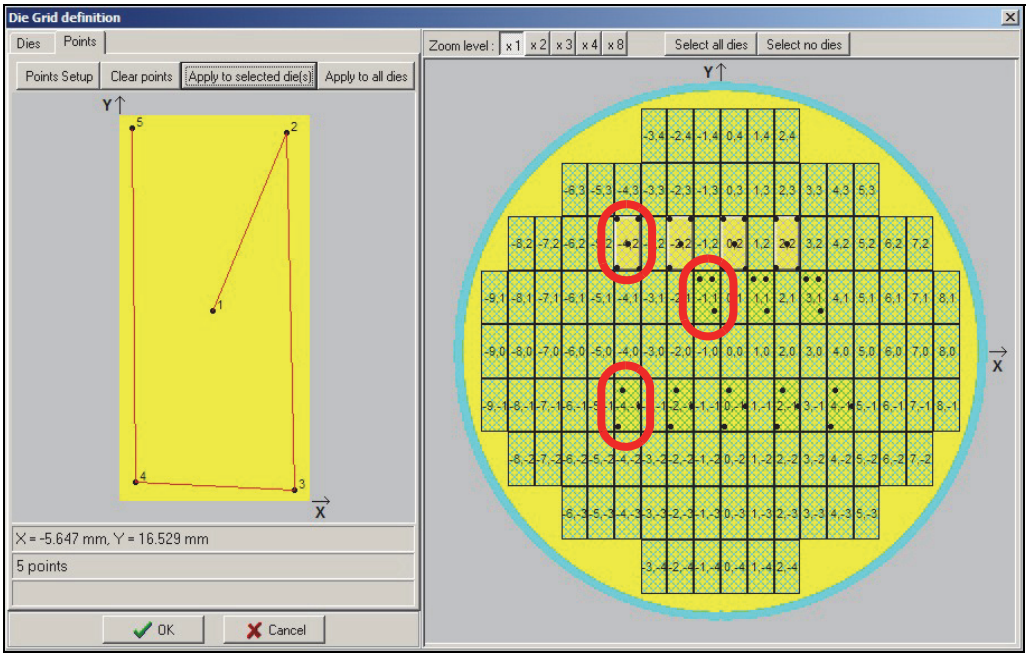


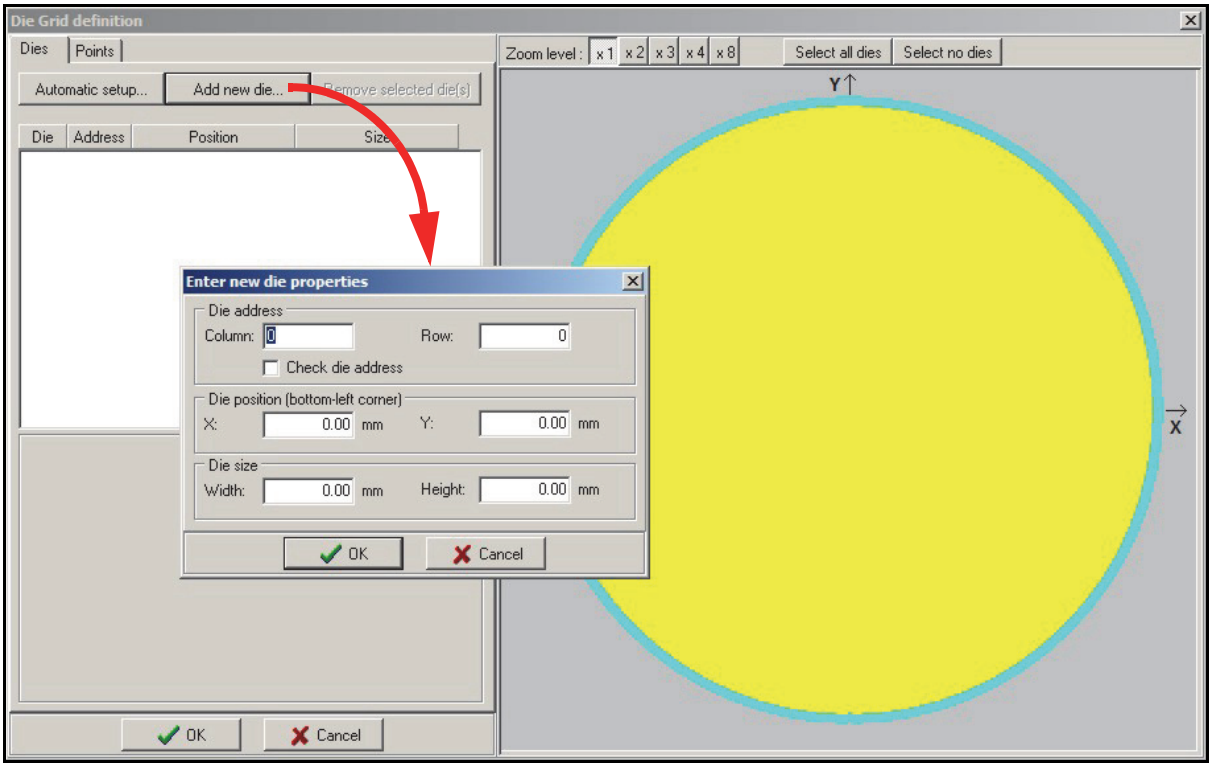

**Figure 17-20 Dies grid results - example 2**

### ● Add new die ... feature

On a new or an existing substrate, a die can be added everywhere on the substrate area with the exception of an already positioned die.

- **Procedure:**

**1** Click on the ***Add new die..*** button, a die properties pop-up screen will be displayed.




**2** Enter the values of the displayed parameters:

**Die address**: it concerns the SEMI regulation. Ignore it if you do not know what it means. The checkbox «check die address» verify if the entered address is unique,

**Die position (bottom-left corner)**: enter the coordinate (X,Y) of the bottom-left corner of the die,

**Die size**: enter the size of the die

Click ***OK*** to validate; the new die will be automatically positioned on the substrate.

# 18 Recipe

The Recipe feature is the most powerful feature of the DeltaPsi2 Software. Using this feature, you will be able to perform the following measurements:

- **Immediate Analysis Recipe**

The fitting process is performed on the current point using a specific acquisition routine with a specific model. This automated process can be recalled and the results saved into a file.

- **Mapping Recipe using a defined simple grid**

In this case, the fitting process is performed on a grid. On each processed point of the grid, it becomes possible to apply a specific acquisition routine with a specific model. The fitting process can be automatically performed point by point on a user predefined order: by incremental points coordinates or by incremental points groups.

- **Mapping Recipe using a defined complex grid**

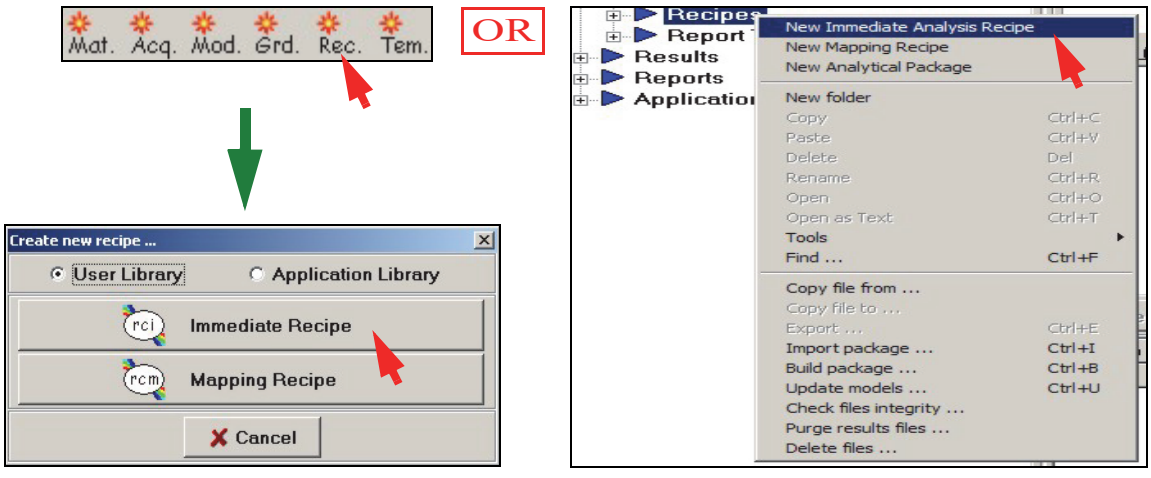
The fitting process is performed on points located on dies which constitute the Complex grid. As the Complex grid can be defined with various measurement points on the dies, the Mapping Recipe can be performed, by groups, on these measurement points. The chapter will describe the various analysis combinations.

## 18.1 Immediate Analysis Recipe

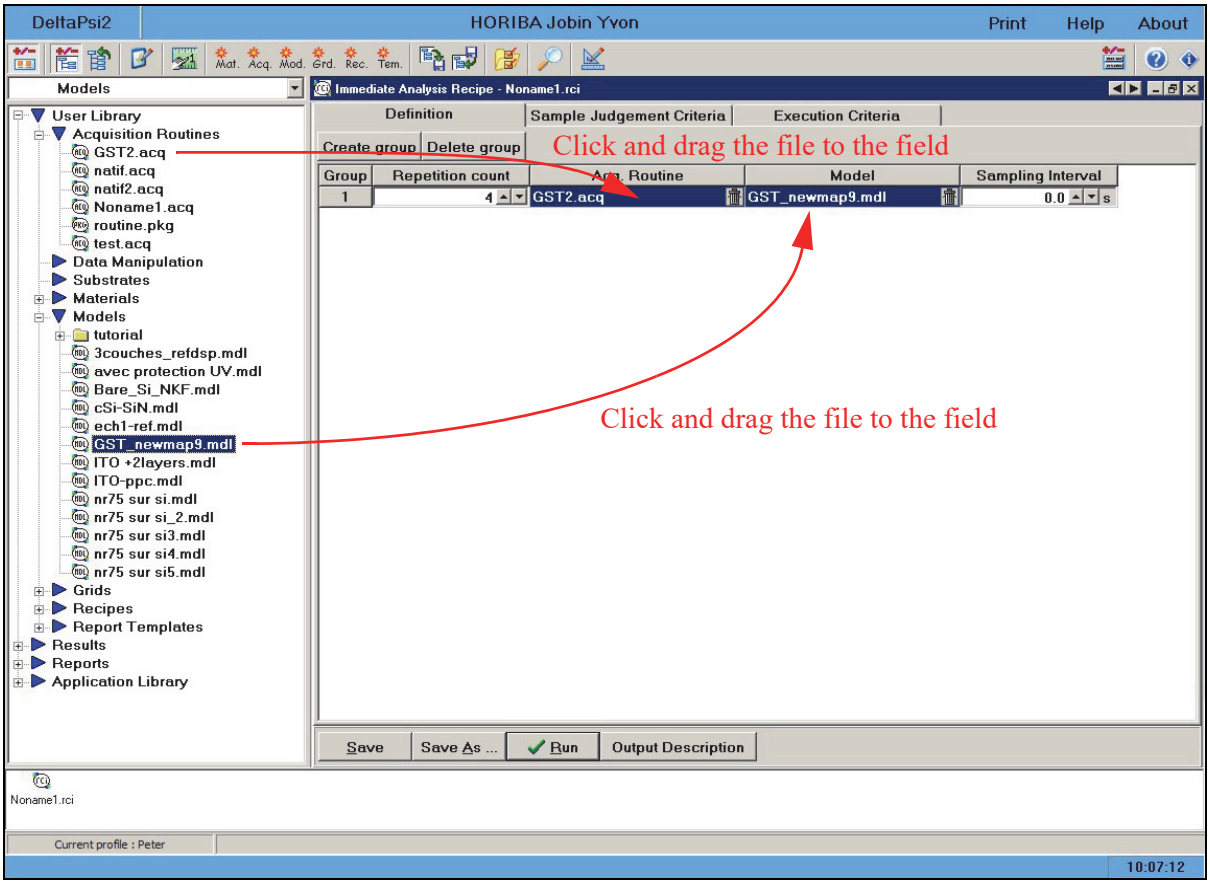
The Immediate Analysis Recipe is useful if daily repetitive fitting processes must be performed on samples. Thus, by opening one recipe file, a fitting can be launched with a specific acquisition routine and model. All the hardware adjustments and process steps are performed automatically. At the end, the results are displayed and saved into a file.

### 18.1.1 Accessing

From the ***Button Bar*** menu, click on the ***Rec.*** icon, then select the Immediate Recipe or from the TreeView menu, right-click on the ***Recipes*** node and choose the ***New Immediate Analysis Recipe*** choice.



The following screen will be then displayed:




The ***Repetition number*** N value will perform N complete cycles. The results are compiled in a single file.

Once the screen is fill out, save the current recipe (***Save*** or ***Save As*** button), then press the ***Run Recipe*** button to perform the measurement and fitting cycles.

The result will be saved in a file located in the ***Recipes*** node of the TreeView menu.

To display the results, double-click on the created file located in the ***Recipes*** node.

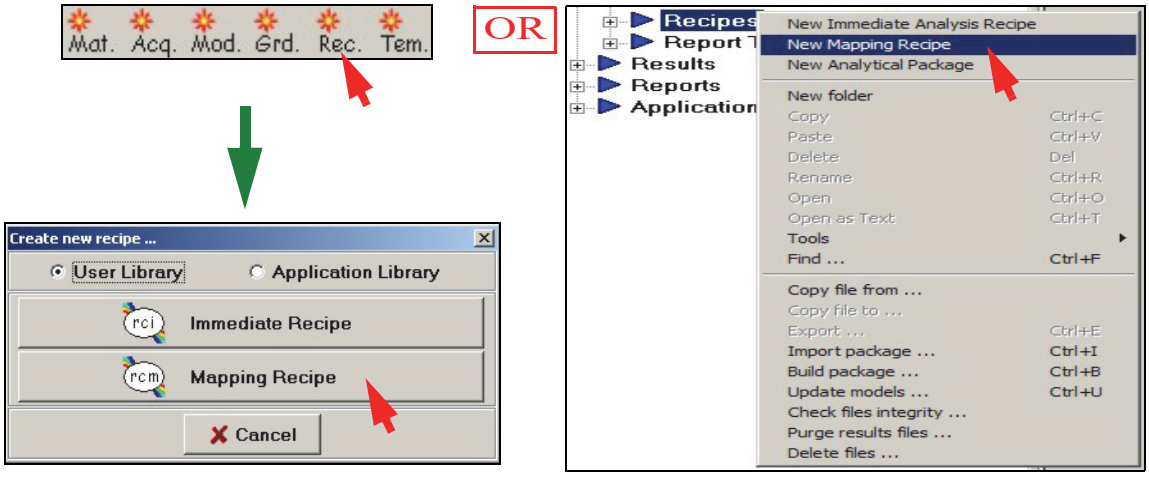
The ***Output Description*** button allows the user to set the parameters and graphics presentation once the complete measurement and fit have been performed. This Output Description can be of course edited after the process. The ***Output Description*** feature is detailed in the paragraph “Output Description feature” on page 264.

## 18.2 Mapping Recipe using a defined simple grid

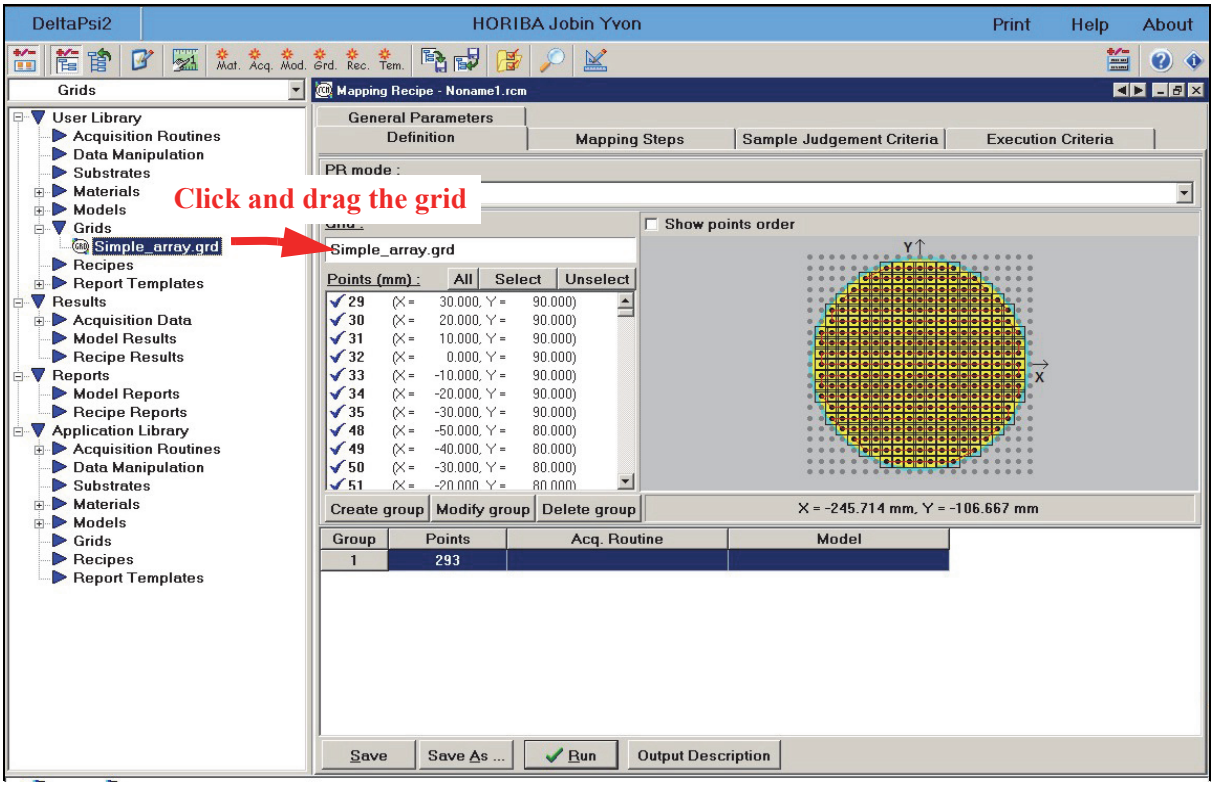
When a simple grid has been previously defined (see “Using Simple Grid” on page 226), the next step is to perform acquisitions and fittings on this grid. This chapter explains how to create a recipe file, which will use one or several acquisition routines and models. Thus, by opening the created recipe file, a fitting can be launched with a specific acquisition routine and model. All the hardware adjustments and process steps are performed automatically. At the end, the results are saved into a file.

### 18.2.1 Accessing

From the ***Button Bar*** menu, click on the ***Rec.*** icon, then select the ***Mapping Recipe*** or from the TreeView menu, right-click on the ***Recipes*** node and choose the ***New Mapping Recipe*** choice.




The following screen will be then displayed:

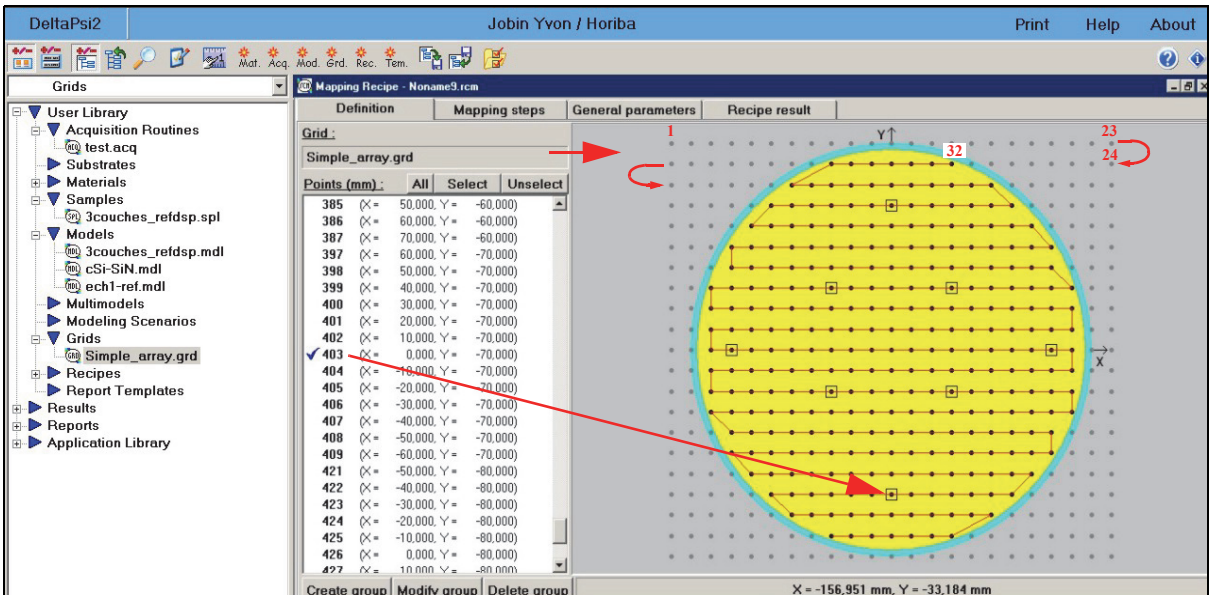



Open the ***Grids*** node, click and drag the grid from which the mapping steps will be created.

### 18.2.2 Mapping Steps creation

The procedure described below will show how to create and launch an automated process over many points located on a grid.

As soon as a grid is loaded, all of the points are listed and a graphical representation of the grid is displayed. Your choice is to select the measurement points, individually or grouped, and assign to each of them an acquisition routine with the adapted built model. The following screen shows an example of the upper part of the mapping recipe.




TreeView

List of points which are located on the substrate. The checked points are selected for measurement process.

This area displays graphically the built substrate. Each time a point is selected for the measurement process, a small square is displayed around this point. The figure shows also how the points are numbered. On the example above, the first valid point (located on the substrate) carries the number 32; it means also that the list will begin with the number 32.

**Figure 18-1 Selecting the points for measurement process**

The figure 18-1 shows how to choose and validate the points which will be included in a group.

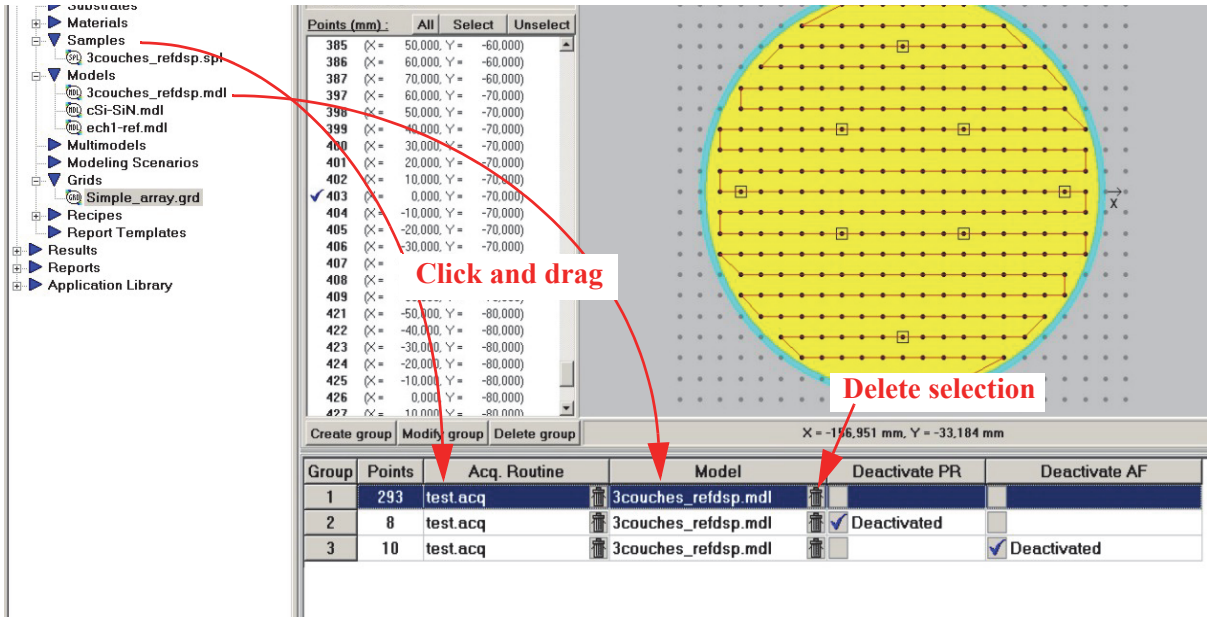
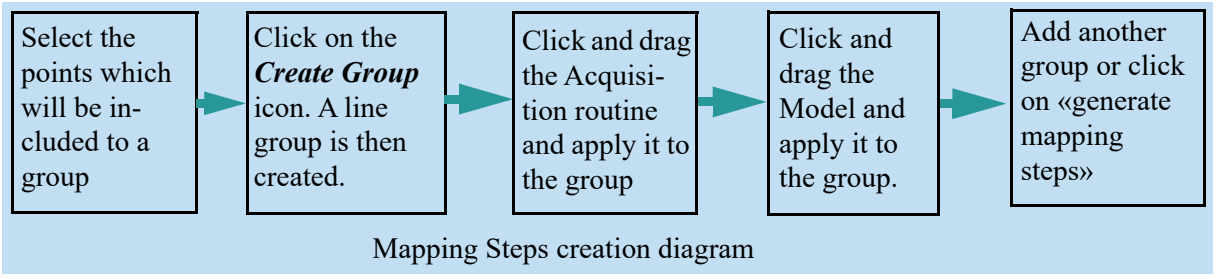



**Figure 18-2 Creating groups**



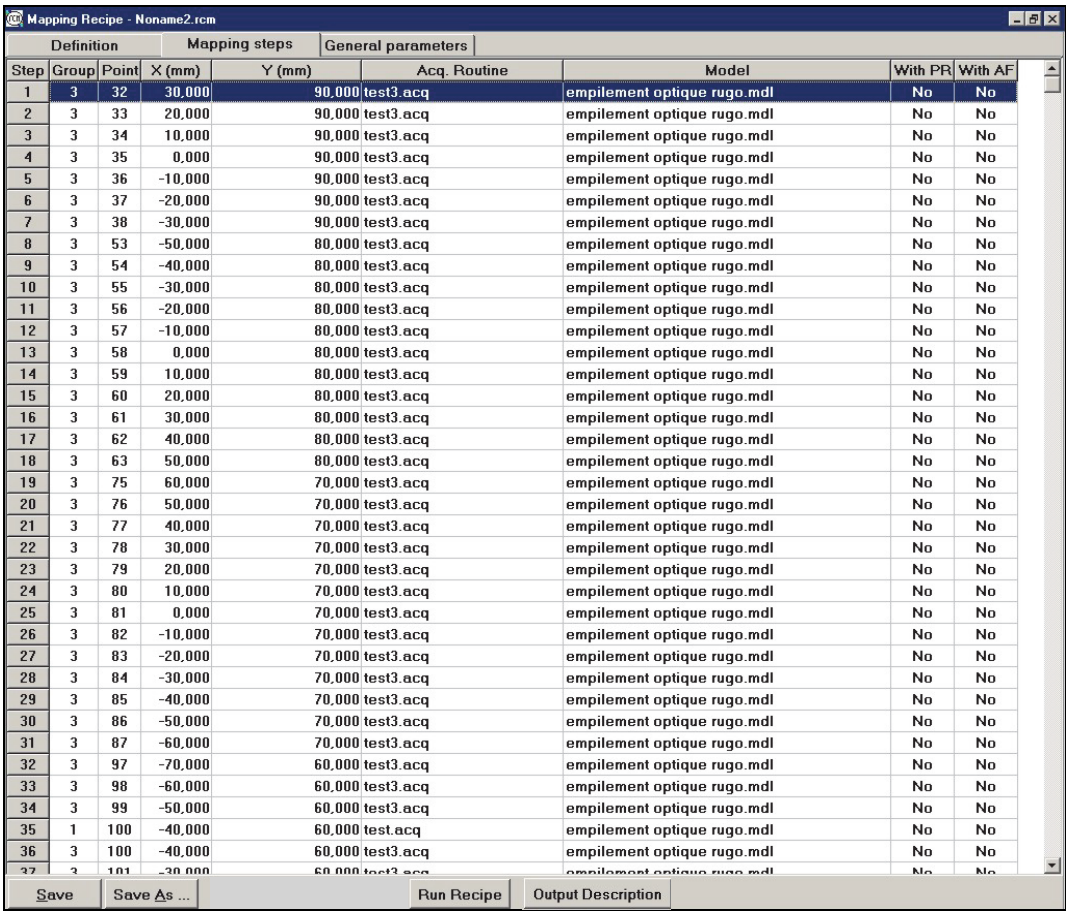

Mapping Steps creation diagram

The figure 18-2 shows an example: three groups have been created with three different acquisition routines on adapted models.

- **deactivate PR option:** the Pattern Recognition (PR) option is a powerful system which detects and marks, in real time, the specific points on a substrate. If the grid has been created from the Pattern Recognition system, the user can deactivate this feature.

- **deactivate AF option:** some systems include an autofocus device. This device can be deactivated.

The ***Output Description*** button allows the user to set the parameters and graphics presentation once the complete measurement and fit have been performed. This Output Description can be set before the process and, of course, edited after the process. The ***Output Description*** feature is detailed in the paragraph "Output Description feature" on page 264.




| Step | Group | Point | X (mm) | Y (mm) | Acq. Routine | Model | With PR | With AF |
|---|---|---|---|---|---|---|---|---|
| 1 | 3 | 32 | 30,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 2 | 3 | 33 | 20,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 3 | 3 | 34 | 10,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 4 | 3 | 35 | 0,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 5 | 3 | 36 | -10,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 6 | 3 | 37 | -20,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 7 | 3 | 38 | -30,000 | 90,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 8 | 3 | 53 | -50,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 9 | 3 | 54 | -40,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 10 | 3 | 55 | -30,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 11 | 3 | 56 | -20,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 12 | 3 | 57 | -10,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 13 | 3 | 58 | 0,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 14 | 3 | 59 | 10,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 15 | 3 | 60 | 20,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 16 | 3 | 61 | 30,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 17 | 3 | 62 | 40,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 18 | 3 | 63 | 50,000 | 80,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 19 | 3 | 75 | 60,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 20 | 3 | 76 | 50,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 21 | 3 | 77 | 40,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 22 | 3 | 78 | 30,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 23 | 3 | 79 | 20,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 24 | 3 | 80 | 10,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 25 | 3 | 81 | 0,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 26 | 3 | 82 | -10,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 27 | 3 | 83 | -20,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 28 | 3 | 84 | -30,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 29 | 3 | 85 | -40,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 30 | 3 | 86 | -50,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 31 | 3 | 87 | -60,000 | 70,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 32 | 3 | 97 | -70,000 | 60,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 33 | 3 | 98 | -60,000 | 60,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 34 | 3 | 99 | -50,000 | 60,000 | test3.acq | empilement optique rugo.mdl | No | No |
| 35 | 1 | 100 | -40,000 | 60,000 | test.acq | empilement optique rugo.mdl | No | No |
| 36 | 3 | 100 | -40,000 | 60,000 | test3.acq | empilement optique rugo.mdl | No | No |

**Figure 18-3**

The figure above lists all of the grid points selected to the measurement process. The following information is available from left to right:

- **Steps**

  The step lists the measurement points using a chronological order.

- **Group**

  The previously created groups (1st group, 2nd group, 3rd group etc...) are listed in front of each measurement point.

Clicking on the ***Group*** button will generate a group sorting from the first to the last. This could be useful if the selected acquisition routines are very different and require a long adjustment time. Once one acquisition routine is set, all of the related mapping points are measured and processed. It speeds the entire measurement process.

- **Point**

  By default, the numbered measurement points are displayed in the incremental order. This order is more suited if a same acquisition routine (or close to) is used to perform the measurements. Results will be delivered faster.

If the list is sorted by groups, click on the *Point* button.

- **X (mm), Y(mm)**

  For each measurement points, the coordinates are displayed.

- **Acq. Routine**

  The acquisition routine which will be used with the related measurement point is displayed in this column.

- **Model**

  The model used with each measurement point is displayed in this column.

- **With PR**

  The grids which have been built with the Pattern Recognition (PR) system are saved with a specific label. This column informs the user if the selected grid has been created using a Pattern recognition system.

- **With AF**

  This column informs the user if the selected grid has been created with the AutoFocus (AF) optional device.

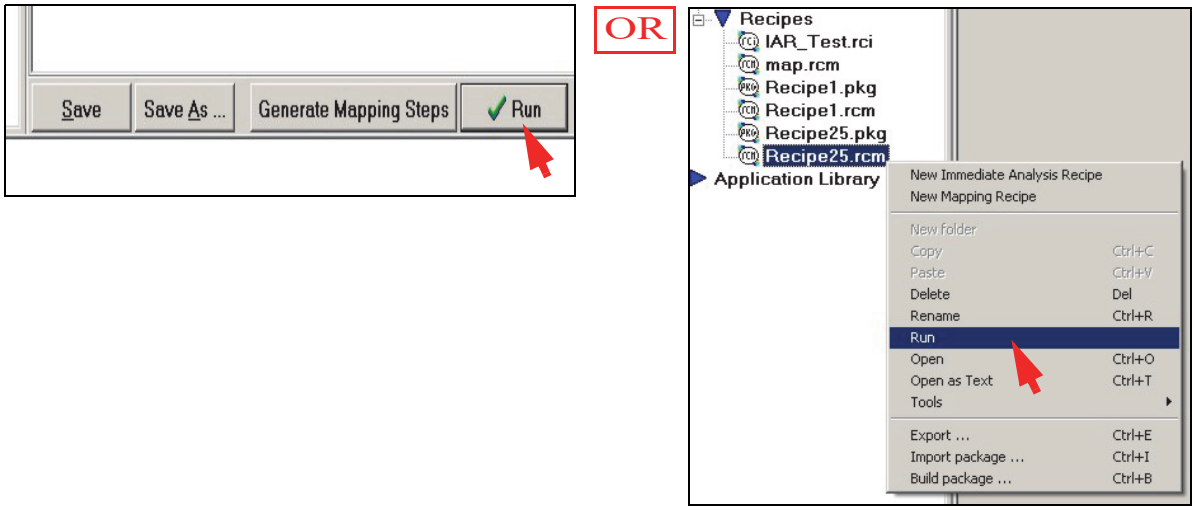
- **Save/Save As ...**

  The Mapping Recipe (name.rcm) configuration can be saved by clicking on the button located on the bottom left hand side of the screen.

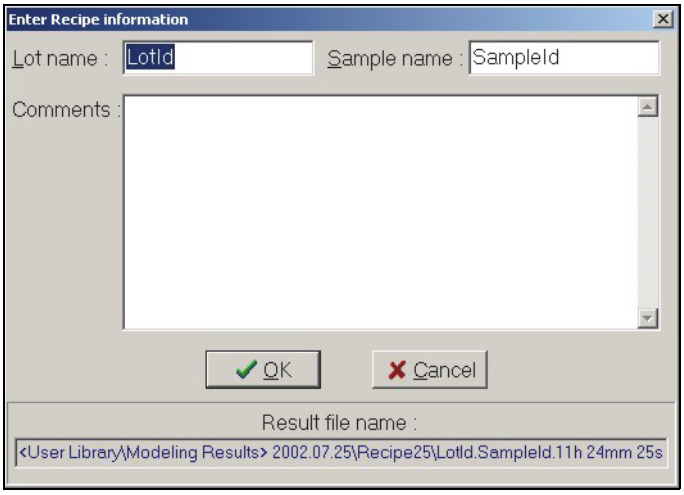
### 18.2.3 Running the Recipe

To start the Recipe process, click on the ***Run*** button located on the bottom of the screen. If the button is greyed, that means that the ***Mapping Steps*** has not been yet generated.

It is also possible to run a Recipe process directly from the Recipe file (RCM extension) listed in the TreeView. In this case, right-click on the file then select the ***Run*** choice.




The following screen is displayed:




Enter the ***Lot*** name, the ***Sample Name*** and various comments about the Recipe. The Recipe results are saved using a specific chained format. This format can be set from the ***File name formats*** screen (see "File name formats" on page 15).

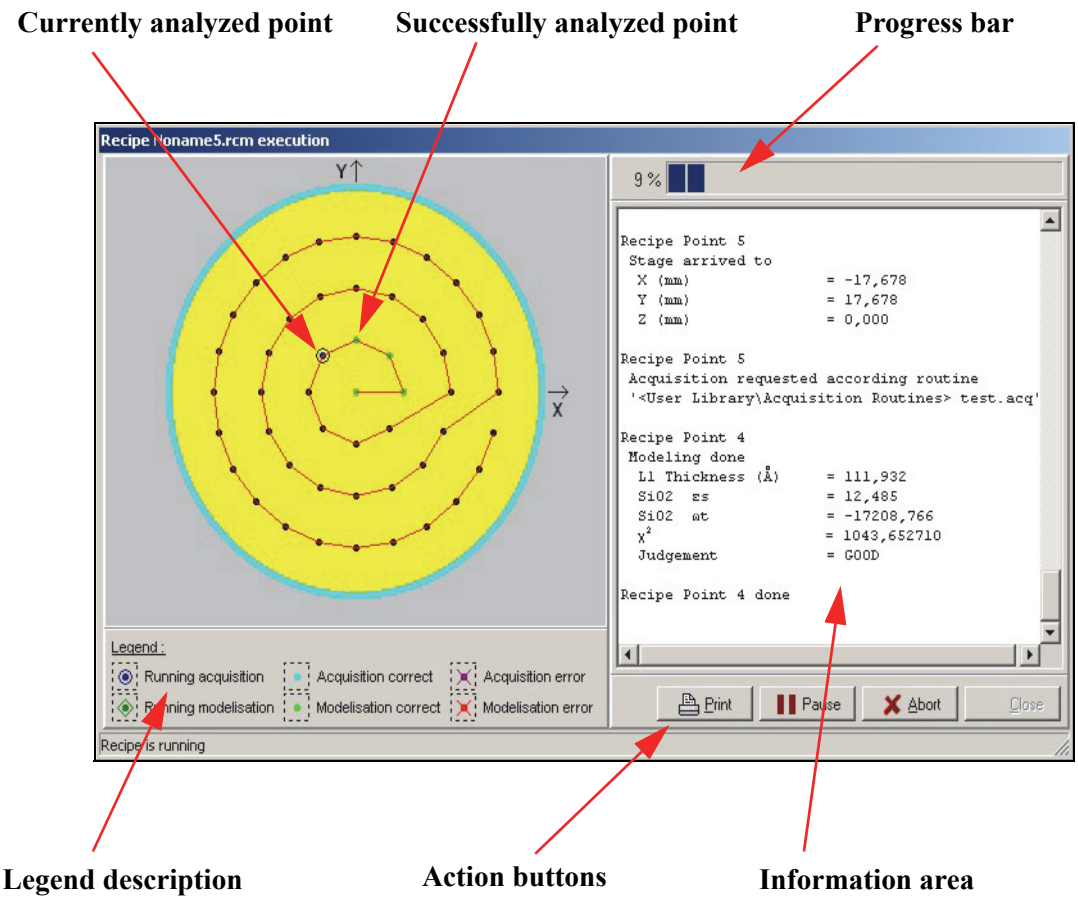
By default the following format will be used:

**User Library\Modeling Results\Year.Month.Day\Recipe name\LotId.SampleId.HourMinSec.**

Click ***OK*** to validate the screen.

Three screens will be displayed (if they have been activated from the ***File name formats*** screen (see "File name formats" on page 15).

The ***Show Recipe progress*** screen shows the progression of the processed points. The screen displays also the primary information about the recipe points condition.




Once a point has been aimed by the System, the Software activates the Acquisition routine. At this time, a new screen is opened and displays the real time acquisition parameters. The following screen is an example of such screen:

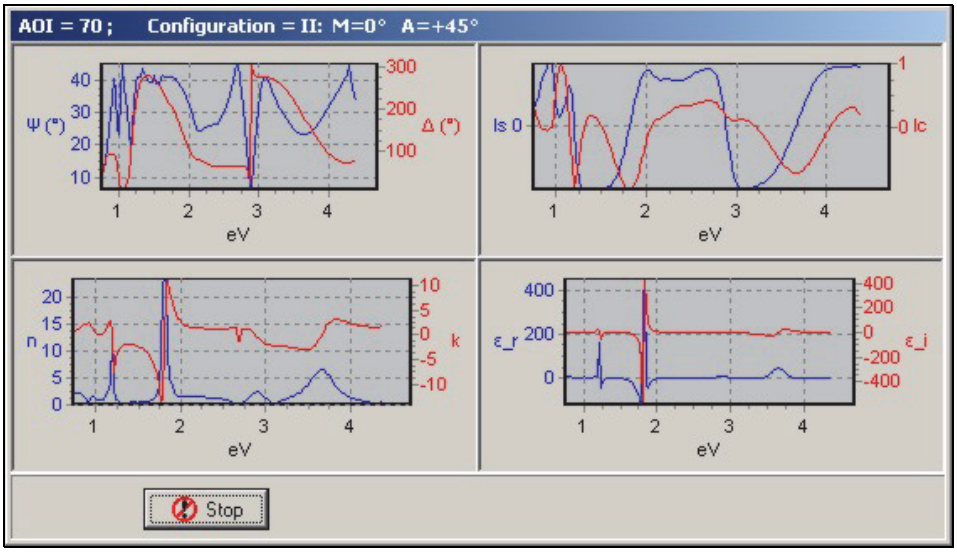



**Figure 18-4 Acquisition progress screen**

Once the acquisition has been performed, the related screen is closed. Then, the fitting screen is displayed and real time fitting process is run. As for the acquisition screen, the fitting screen is closed at the end of the process.

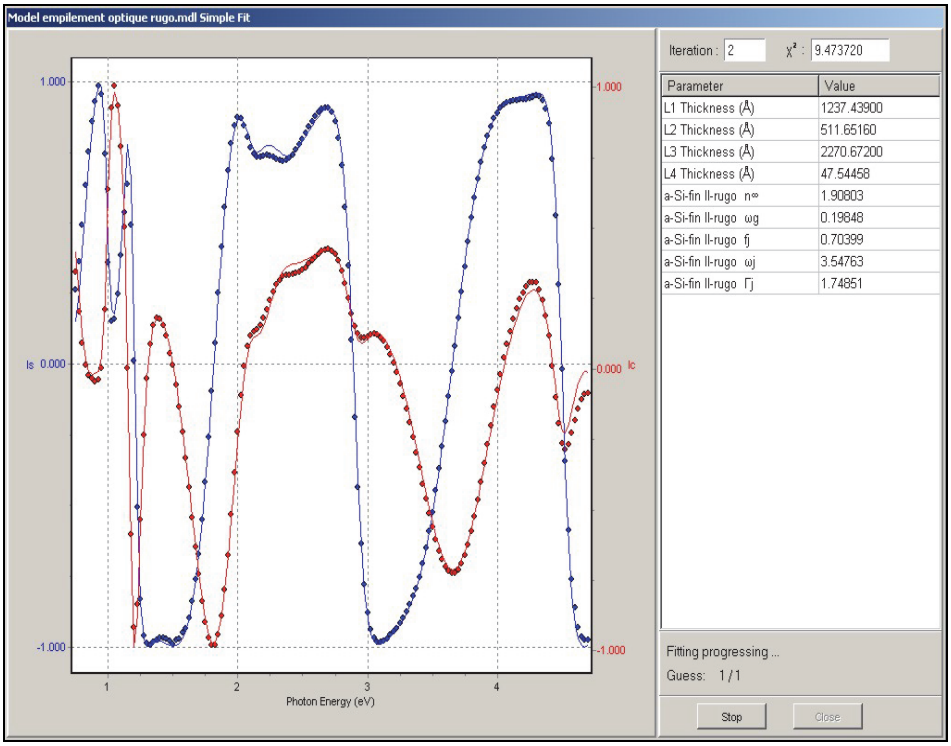
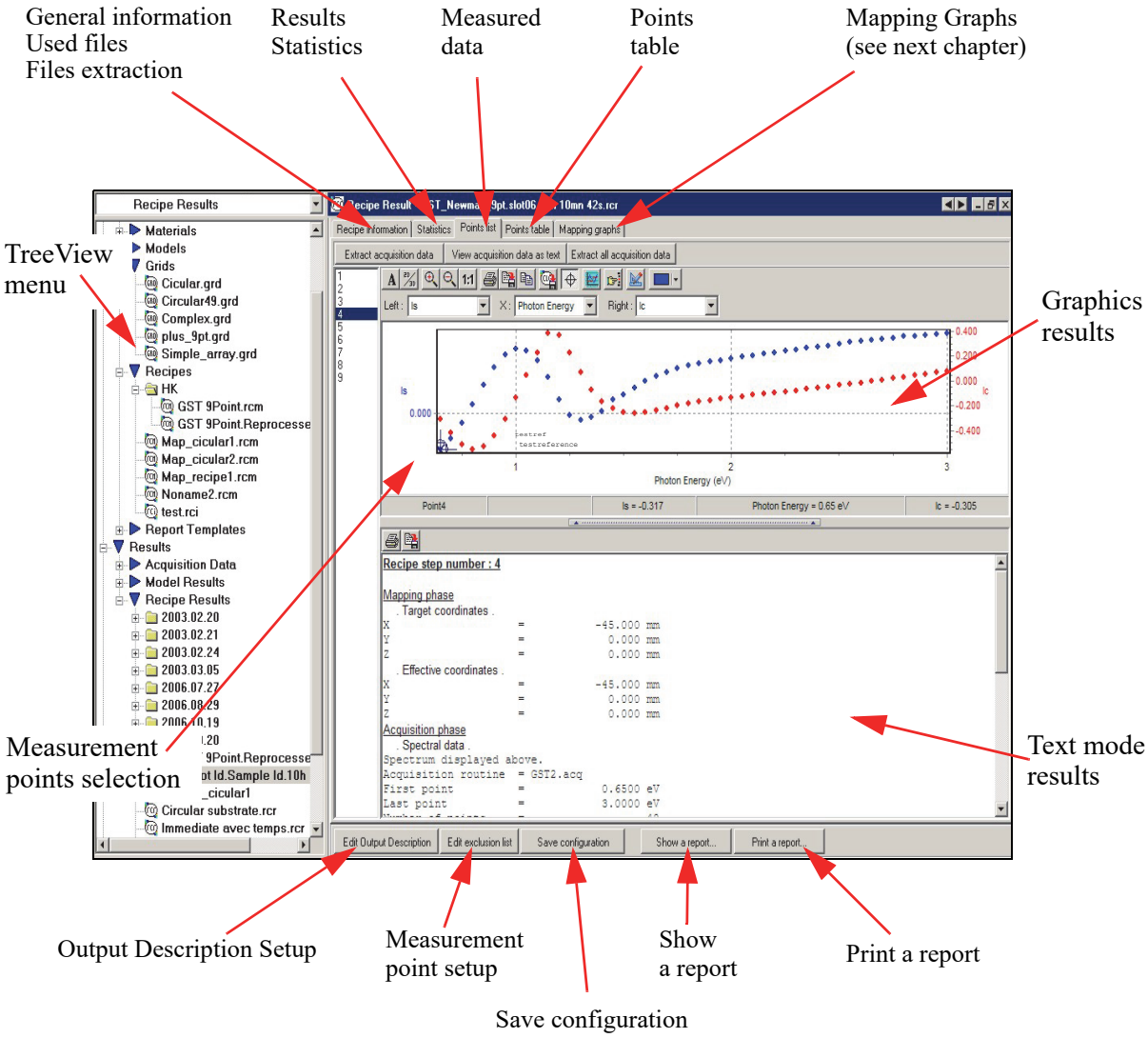



**Figure 18-5 Fitting progress screen**

The Software activates the XY Stage to the next point to analyze. At the end of the mapping process, the Recipe screen progress stops and the ***Close*** button becomes active.

The results are saved in the ***Modeling Results*** folder located in the TreeView menu. Double-click on the file to open it to visualize the results.

The results screen compiles all of the informations which have been used and which have been produced during and after the Recipe process.



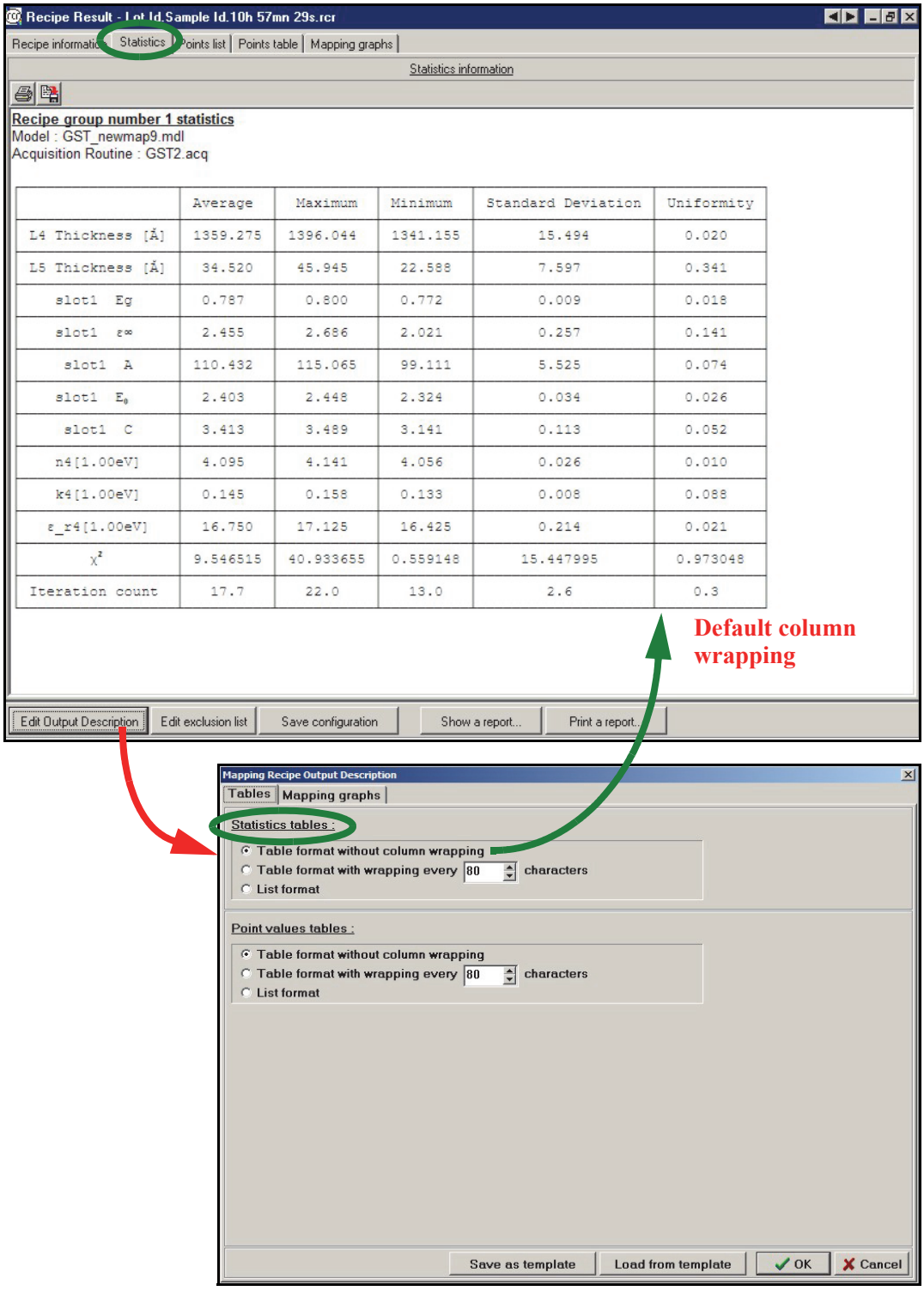

## 18.2.4 Output Description feature

The **Output Description** feature allows the user to set the parameters and graphics presentation once the complete measurement and fit have been performed. This Output Description button can be found on every Recipe main screen. The related parameters can be set before the process and, of course, edited after the process.

- **What is it possible to set ?**
  - The Statistics tables and list formats,
  - The Trends (for Immediate Recipe) and Mapping (for Mapping Recipe) Graphs,

When an Immediate Analysis or Mapping Recipe is launched the results are displayed in a 5-tab screen. Each tab concerns a specific parameters class. The tables, lists and graphics representation can be defined by the user.

Figure 18-6, “Tables setting Output Description”, page 265 shows an example of «***Table format without column wrapping***». This format has no limit on width; if the parameters results are long it should be necessary to limit the width (limiting characters) by using the second options.




| | Average | Maximum | Minimum | Standard Deviation | Uniformity |
|---|---|---|---|---|---|
| L4 Thickness [Å] | 1359.275 | 1396.044 | 1341.155 | 15.494 | 0.020 |
| L5 Thickness [Å] | 34.520 | 45.945 | 22.588 | 7.597 | 0.341 |
| slot1 Eg | 0.787 | 0.800 | 0.772 | 0.009 | 0.018 |
| slot1 $\varepsilon\infty$ | 2.455 | 2.686 | 2.021 | 0.257 | 0.141 |
| slot1 A | 110.432 | 115.065 | 99.111 | 5.525 | 0.074 |
| slot1 $E_0$ | 2.403 | 2.448 | 2.324 | 0.034 | 0.026 |
| slot1 C | 3.413 | 3.489 | 3.141 | 0.113 | 0.052 |
| n4[1.00eV] | 4.095 | 4.141 | 4.056 | 0.026 | 0.010 |
| k4[1.00eV] | 0.145 | 0.158 | 0.133 | 0.008 | 0.088 |
| $\varepsilon\_r4$[1.00eV] | 16.750 | 17.125 | 16.425 | 0.214 | 0.021 |
| $\chi^2$ | 9.546515 | 40.933655 | 0.559148 | 15.447995 | 0.973048 |
| Iteration count | 17.7 | 22.0 | 13.0 | 2.6 | 0.3 |

**Figure 18-6 Tables setting Output Description**

Figure 18-7, “Graphs setting Output Description”, page 266 shows how to add a graph which is not displayed by default and how to change the representation: size, color, orientation ...

05/20/2016

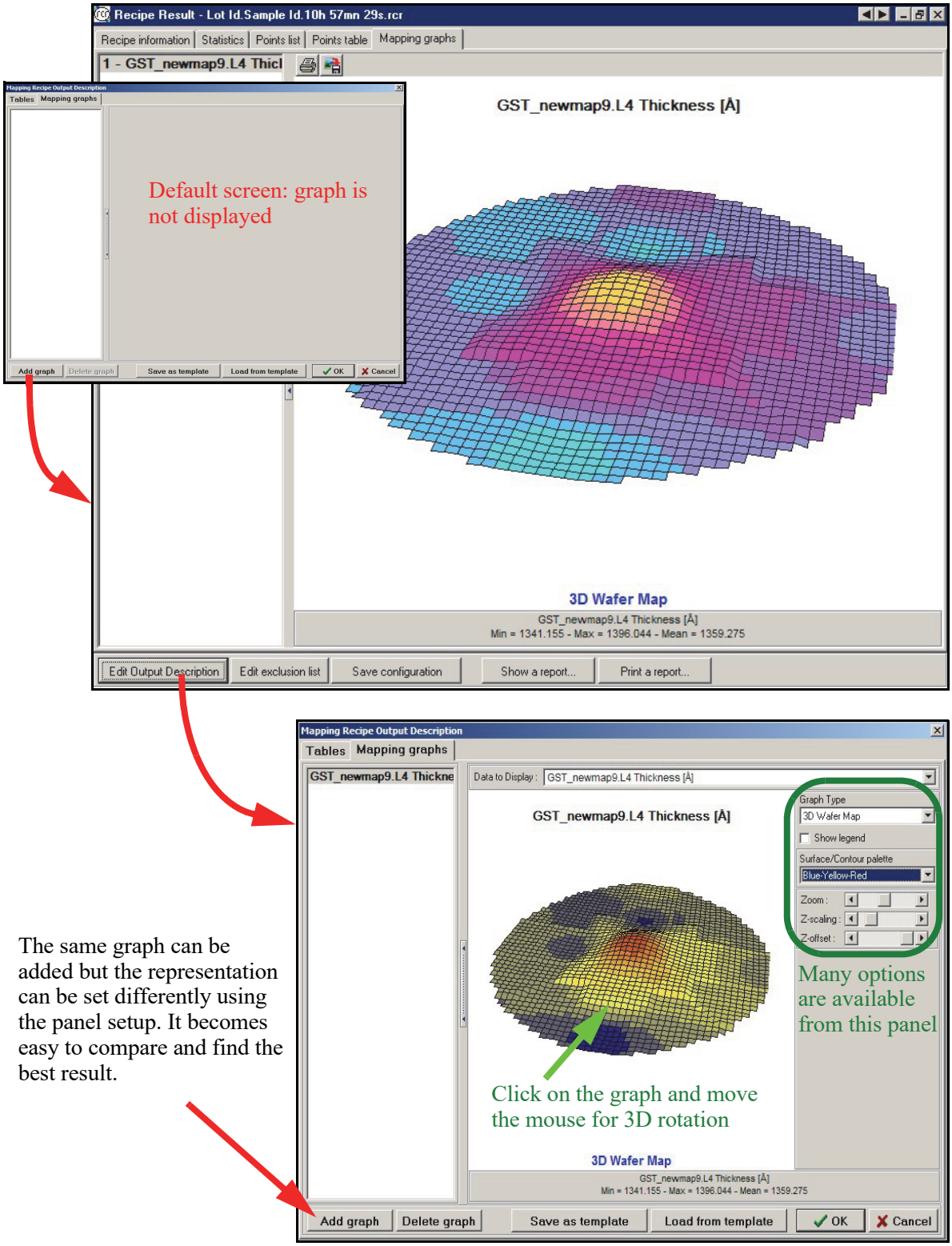


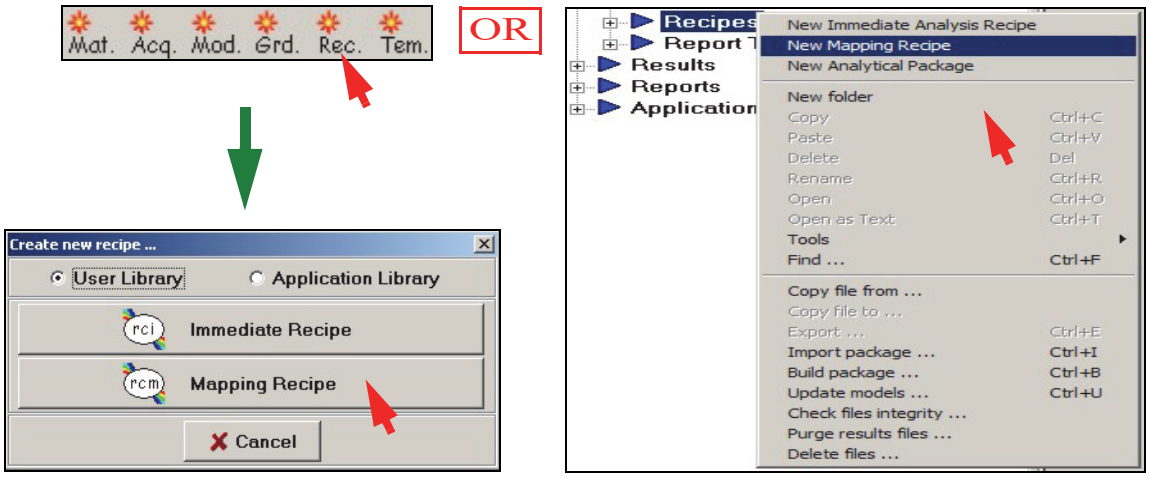

**Figure 18-7 Graphs setting Output Description**

# 18.3 Mapping Recipe using a defined complex grid

When a complex grid has been previously defined (see "Using Complex Grid" on page 244), the next step is to perform acquisitions and fittings on this grid. This chapter explains how to create a recipe file, which will use one or several acquisition routines and models. Thus, by opening the created recipe file, a fitting can be launched with a specific acquisition routine and model. All the hardware adjustments and process steps are performed automatically. At the end, the results are saved into a file.

## 18.3.1 Accessing

From the TreeView menu, right-click on the ***Recipes*** node and choose the ***New Mapping Recipe*** option.




The following screen will be then displayed

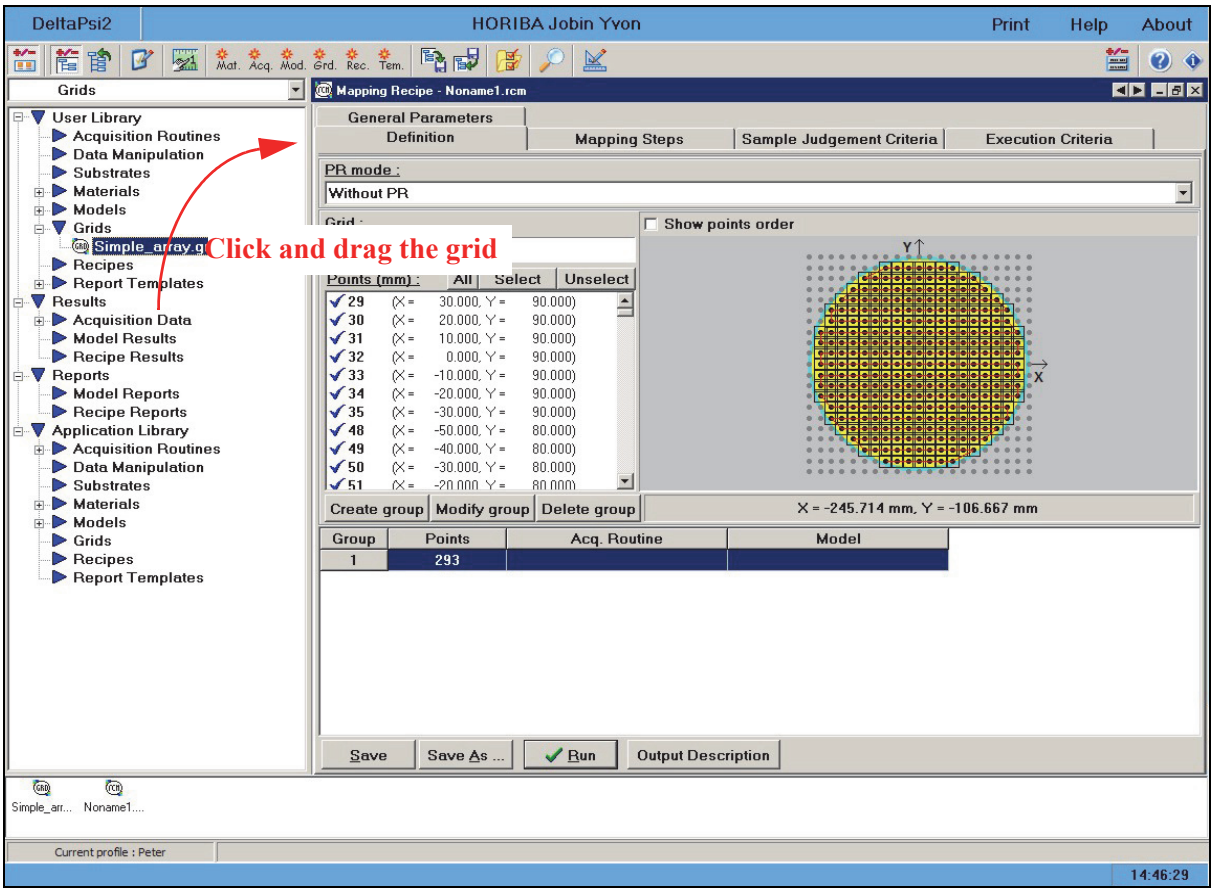


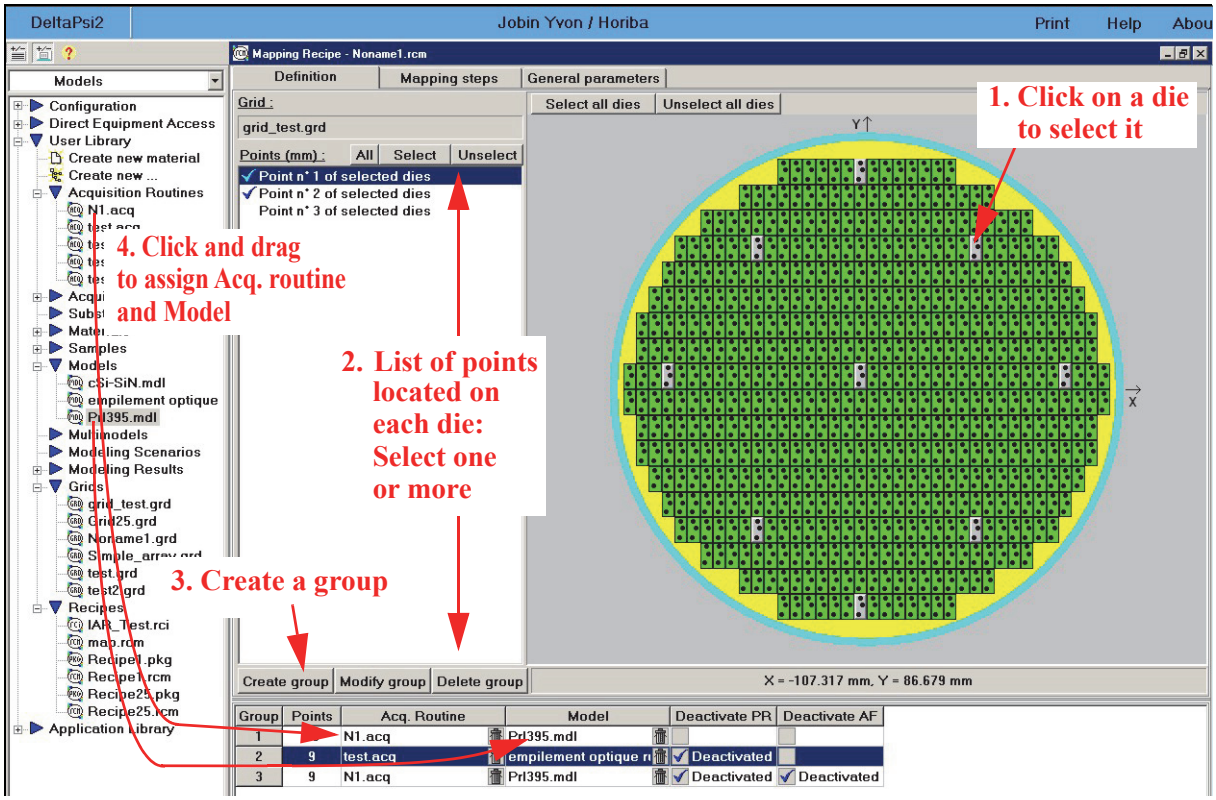

**Figure 18-8 Complex grid activation**

Open the ***Grids*** node, click and drag the grid from which the mapping steps will be created.

### 18.3.2 Mapping Steps creation

The procedure described below will show how to create and launch an automated process over many points located on one or many dies of a substrate.

As soon as a grid is loaded, all of the dies points are graphically displayed. Your choice is to select the measurement dies, then the point(s) of the die(s) to create a group. Each group must be then assigned to an acquisition routine with the adapted built model. The following screen shows an example of the upper part of the mapping recipe. Follow the steps shown on the figure 18-9.



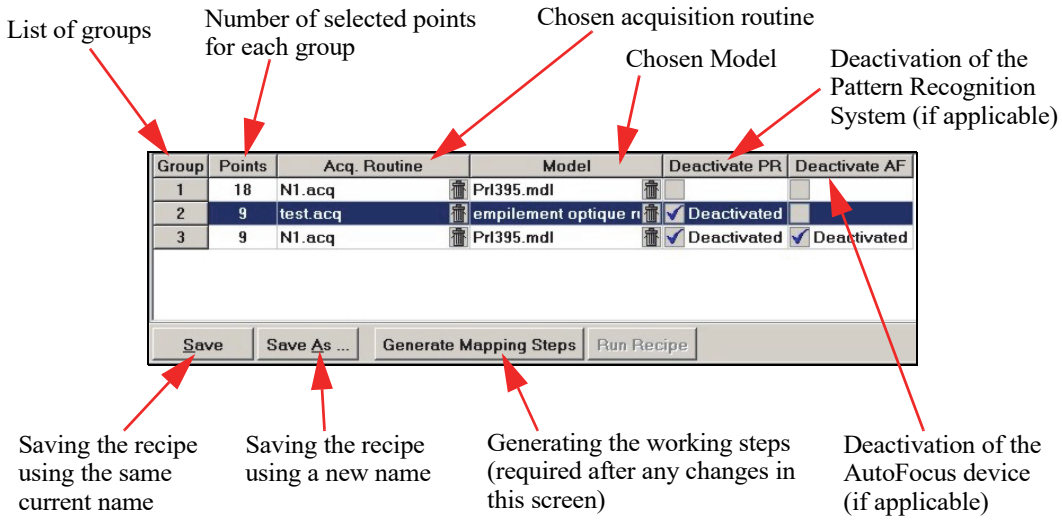
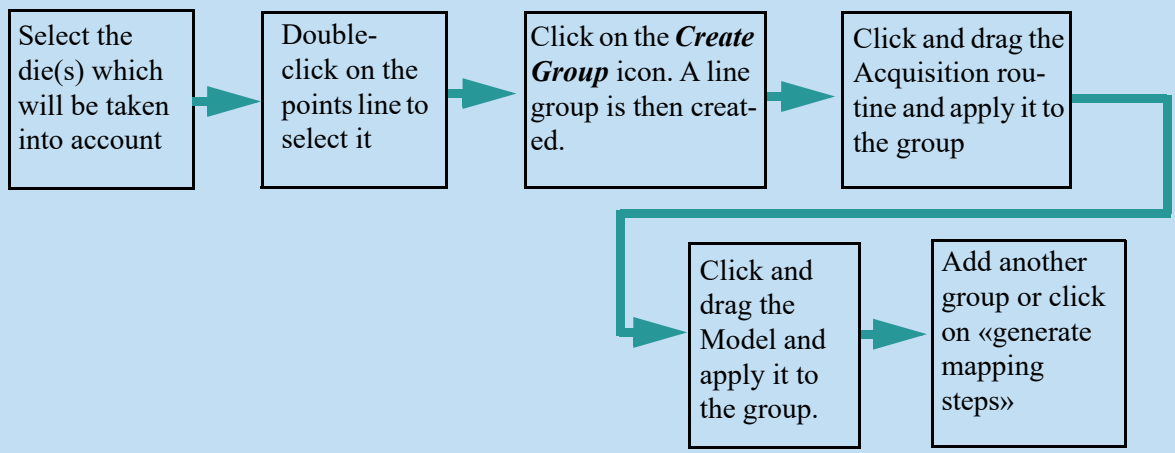




**Figure 18-9 Creating groups on complex grids**

05/20/2016




Mapping Steps creation diagram

The figure 18-9 shows an example: three groups have been created with three different acquisition routines on adapted models.

- **deactivate PR option:** the Pattern Recognition (PR) option is a powerful system which detects and marks, in real time, the specific points on a substrate. If the dies grid has been created from the Pattern Recognition system, the user can deactivate this feature.

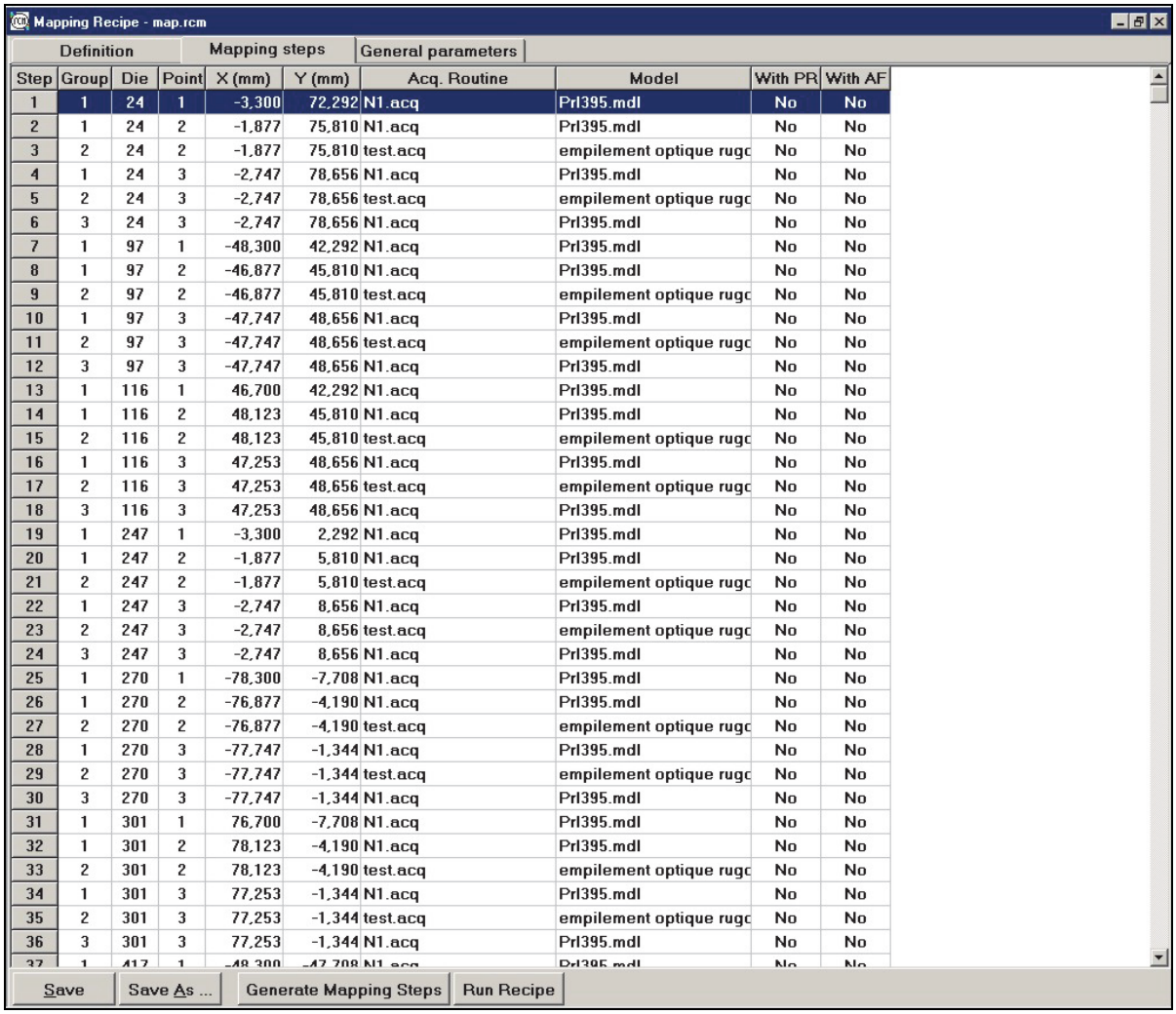
- **deactivate AF option:** some systems include an autofocus device. This device can be deactivated.

Once the recipe is prepared, save it using a judicious name.

As a rule, the dies located on a substrate are identical. The points located on each die can be different.

A saved Recipe can be re-edited later.

Click on the Generate Mapping Steps button to generate the Mapping Steps. The following Mapping Steps screen will be displayed.




| Step | Group | Die | Point | X (mm) | Y (mm) | Acq. Routine | Model | With PR | With AF |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | 24 | 1 | -3,300 | 72,292 | N1.acq | Prl395.mdl | No | No |
| 2 | 1 | 24 | 2 | -1,877 | 75,810 | N1.acq | Prl395.mdl | No | No |
| 3 | 2 | 24 | 2 | -1,877 | 75,810 | test.acq | empilement optique rugc | No | No |
| 4 | 1 | 24 | 3 | -2,747 | 78,656 | N1.acq | Prl395.mdl | No | No |
| 5 | 2 | 24 | 3 | -2,747 | 78,656 | test.acq | empilement optique rugc | No | No |
| 6 | 3 | 24 | 3 | -2,747 | 78,656 | N1.acq | Prl395.mdl | No | No |
| 7 | 1 | 97 | 1 | -48,300 | 42,292 | N1.acq | Prl395.mdl | No | No |
| 8 | 1 | 97 | 2 | -46,877 | 45,810 | N1.acq | Prl395.mdl | No | No |
| 9 | 2 | 97 | 2 | -46,877 | 45,810 | test.acq | empilement optique rugc | No | No |
| 10 | 1 | 97 | 3 | -47,747 | 48,656 | N1.acq | Prl395.mdl | No | No |
| 11 | 2 | 97 | 3 | -47,747 | 48,656 | test.acq | empilement optique rugc | No | No |
| 12 | 3 | 97 | 3 | -47,747 | 48,656 | N1.acq | Prl395.mdl | No | No |
| 13 | 1 | 116 | 1 | 46,700 | 42,292 | N1.acq | Prl395.mdl | No | No |
| 14 | 1 | 116 | 2 | 48,123 | 45,810 | N1.acq | Prl395.mdl | No | No |
| 15 | 2 | 116 | 2 | 48,123 | 45,810 | test.acq | empilement optique rugc | No | No |
| 16 | 1 | 116 | 3 | 47,253 | 48,656 | N1.acq | Prl395.mdl | No | No |
| 17 | 2 | 116 | 3 | 47,253 | 48,656 | test.acq | empilement optique rugc | No | No |
| 18 | 3 | 116 | 3 | 47,253 | 48,656 | N1.acq | Prl395.mdl | No | No |
| 19 | 1 | 247 | 1 | -3,300 | 2,292 | N1.acq | Prl395.mdl | No | No |
| 20 | 1 | 247 | 2 | -1,877 | 5,810 | N1.acq | Prl395.mdl | No | No |
| 21 | 2 | 247 | 2 | -1,877 | 5,810 | test.acq | empilement optique rugc | No | No |
| 22 | 1 | 247 | 3 | -2,747 | 8,656 | N1.acq | Prl395.mdl | No | No |
| 23 | 2 | 247 | 3 | -2,747 | 8,656 | test.acq | empilement optique rugc | No | No |
| 24 | 3 | 247 | 3 | -2,747 | 8,656 | N1.acq | Prl395.mdl | No | No |
| 25 | 1 | 270 | 1 | -78,300 | -7,708 | N1.acq | Prl395.mdl | No | No |
| 26 | 1 | 270 | 2 | -76,877 | -4,190 | N1.acq | Prl395.mdl | No | No |
| 27 | 2 | 270 | 2 | -76,877 | -4,190 | test.acq | empilement optique rugc | No | No |
| 28 | 1 | 270 | 3 | -77,747 | -1,344 | N1.acq | Prl395.mdl | No | No |
| 29 | 2 | 270 | 3 | -77,747 | -1,344 | test.acq | empilement optique rugc | No | No |
| 30 | 3 | 270 | 3 | -77,747 | -1,344 | N1.acq | Prl395.mdl | No | No |
| 31 | 1 | 301 | 1 | 76,700 | -7,708 | N1.acq | Prl395.mdl | No | No |
| 32 | 1 | 301 | 2 | 78,123 | -4,190 | N1.acq | Prl395.mdl | No | No |
| 33 | 2 | 301 | 2 | 78,123 | -4,190 | test.acq | empilement optique rugc | No | No |
| 34 | 1 | 301 | 3 | 77,253 | -1,344 | N1.acq | Prl395.mdl | No | No |
| 35 | 2 | 301 | 3 | 77,253 | -1,344 | test.acq | empilement optique rugc | No | No |
| 36 | 3 | 301 | 3 | 77,253 | -1,344 | N1.acq | Prl395.mdl | No | No |
| 37 | 1 | 417 | 1 | -48,300 | -47,708 | N1.acq | Prl395.mdl | No | No |

**Figure 18-10 Complex Grid Mapping steps**

The figure 18-10 lists all of the dies points selected to the measurement process. The following information is available from left to right:

- **Steps**

  The step lists the measurement points using a chronological order.

- **Group**

  The previously created groups (1st group, 2nd group, 3rd group etc...) are listed in front of each measurement point.

  Clicking on the ***Group*** button will generate a group sorting from the first to the last.

  In this case, the ***Point*** column is also sorted and has priority on the ***Die*** column.

  This could be useful if the selected acquisition routines are very different and require a long adjustment time. Once one acquisition routine is set, all of the related mapping points are measured and processed. It speeds the entire measurement process.

The following table describes the priority sorting:

**Die –> Point –> Group**

**Point –> Group –> Die**

**Group –> Point –> Die**

- **Die**

  This column shows the number assigned to each die of the Complex Grid. The counting starts from the upper left hand side of the substrate.

- **Point**

  By default, the numbered measurement points are displayed in the incremental order. This order is more suited if a same acquisition routine (or close to) is used to perform the measurements. Results will be delivered faster.

If the list is sorted by groups, click on the *Point* button. See above the priority sorting.

- **X (mm), Y(mm)**

  For each measurement points, the coordinates on the related die are displayed.

- **Acq. Routine**

  The acquisition routine which will be used with the related measurement point is displayed in this column.

- **Model**

  The model used with each measurement point is displayed in this column.

- **With PR**

  The grids which have been built with the Pattern Recognition (PR) system are saved with a specific label. This column informs the user if the selected Complex Grid has been created using a Pattern recognition system.

- **With AF**

  This column informs the user if the selected grid has been created with the AutoFocus (AF) optional device.

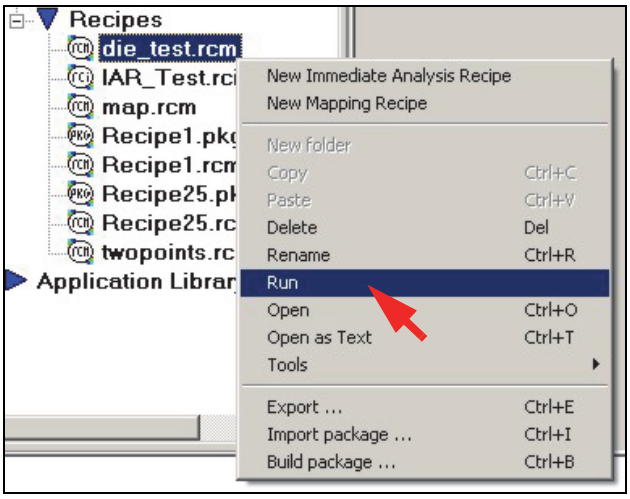
- **Save/Save As ...**

  The Mapping Recipe (name.rcm) configuration can be saved by clicking on the button located on the bottom left hand side of the screen.

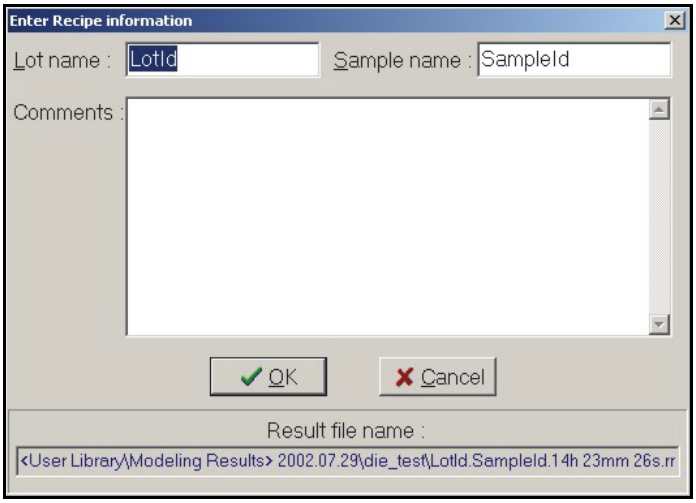
### 18.3.3 Running the Recipe

Once the mapping steps have been built and the measurement points correctly assigned, the Recipe process can be performed.

To start the Recipe process, right-click on the Recipe file (RCP extension) and select the ***Run*** choice.




The following screen is displayed:




Enter the ***Lot*** name, the ***Sample Name*** and various comments about the Recipe. The Recipe results are saved using a specific chained format. This format can be set from the ***File name formats*** screen (see "File name formats" on page 15).

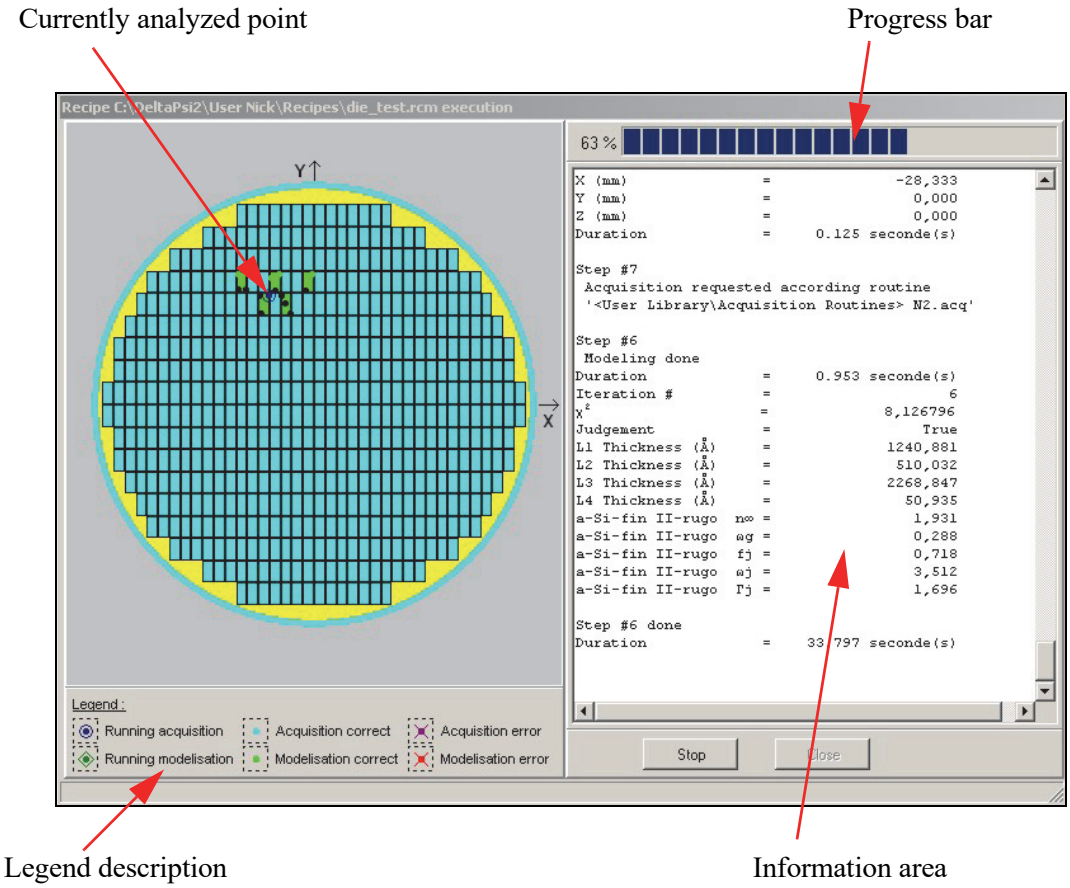
By default the following format will be used:

**User Library\Modeling Results\Year.Month.Day\Recipe name\LotId.SampleId.HourMinSec.**

Click ***OK*** to validate the screen.

Three screens will be displayed (if they have been activated from the ***File name formats*** screen (see "File name formats" on page 15).

The ***Show Recipe progress*** screen shows the progression of the processed points. The screen displays also the primary information about the recipe points condition.




Once a point has been aimed by the System, the Software activates the Acquisition routine. At this time, a new screen is opened and displays the real time acquisition parameters. The following screen is an example of such screen:

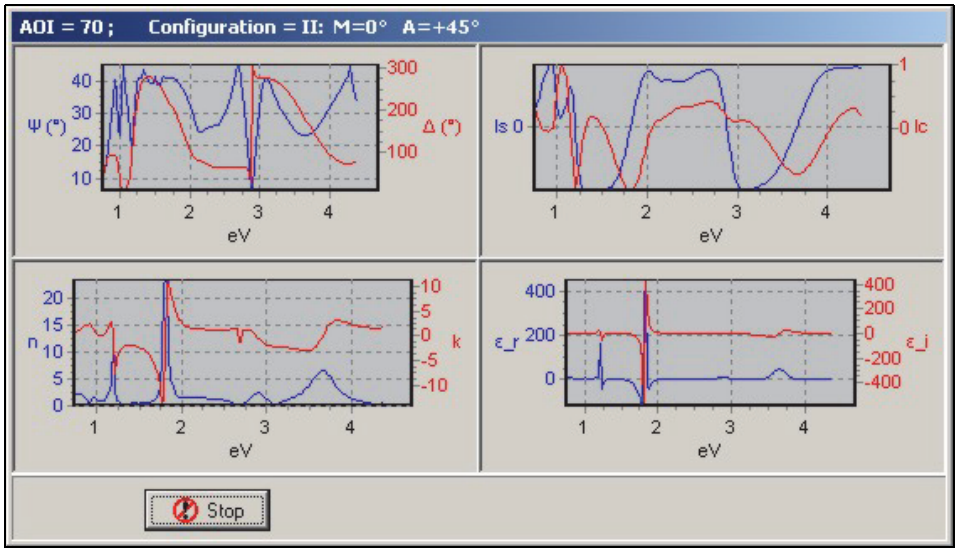



**Figure 18-11 Acquisition progress screen**

Once the acquisition has been performed, the related screen is closed. Then, the fitting screen is displayed and real time fitting process is run. As for the acquisition screen, the fitting screen is closed at the end of the process

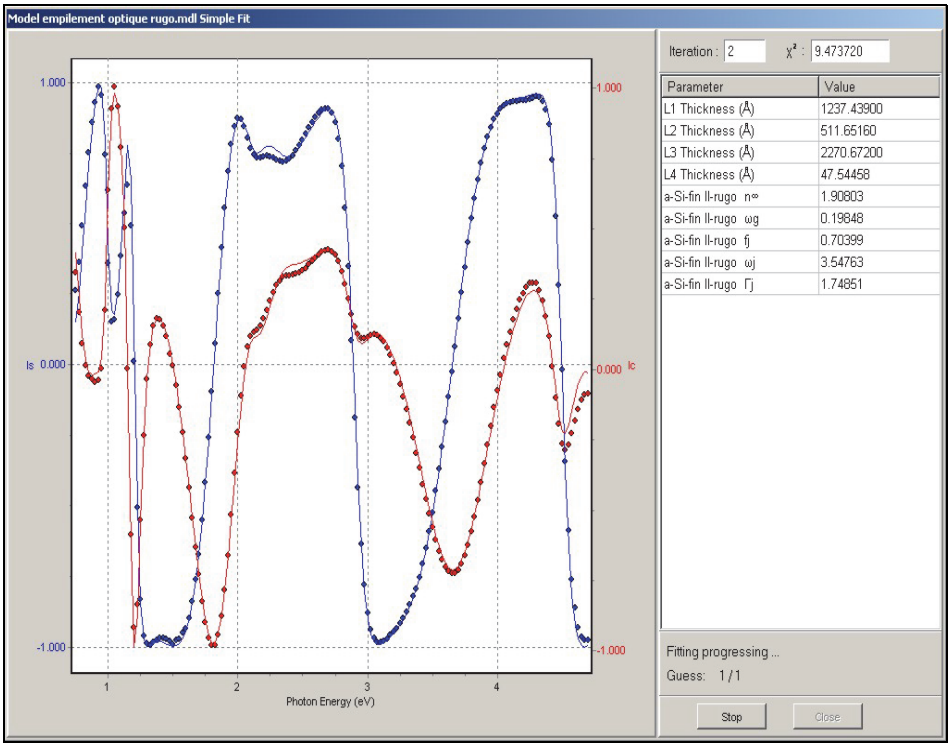



**Figure 18-12 Fitting progress screen**

The Software activates the XY Stage to the next point to analyze. At the end of the mapping process, the Recipe screen progress stops and the ***Close*** button becomes active.

The results are saved in the ***Recipe Results*** folder located in the TreeView menu. Double-click on the file to open it to visualize the results.

The linked information screen includes numbered steps. Clicking on each step will display the related date. Right-clicking on the step will allow a data file extraction. Thus, the acquired data of each step can be extracted in a file (see figure 18-13).

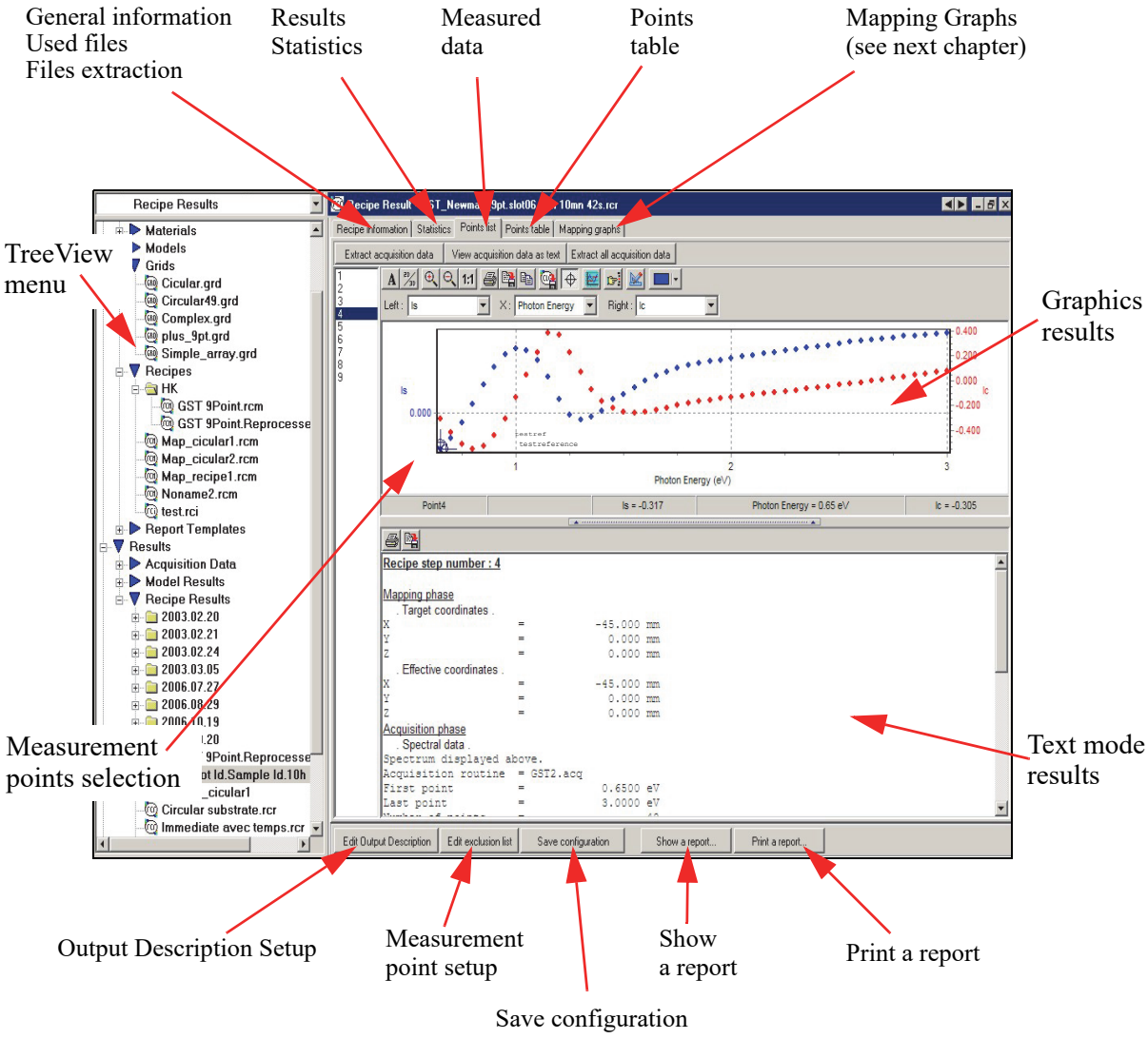



**Figure 18-13 Recipe Results screen**

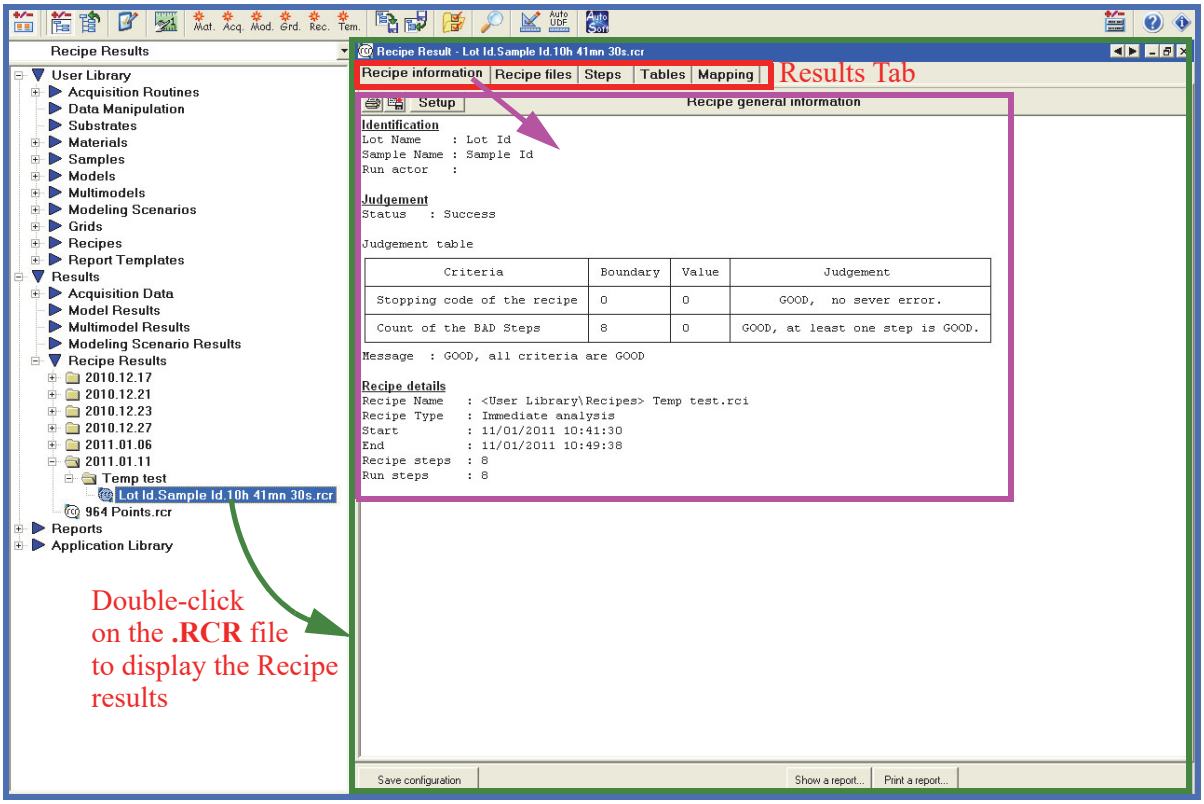
### 18.3.4 Output Description feature

See detailed description in the chapter **"Output Description feature" on page 264**.

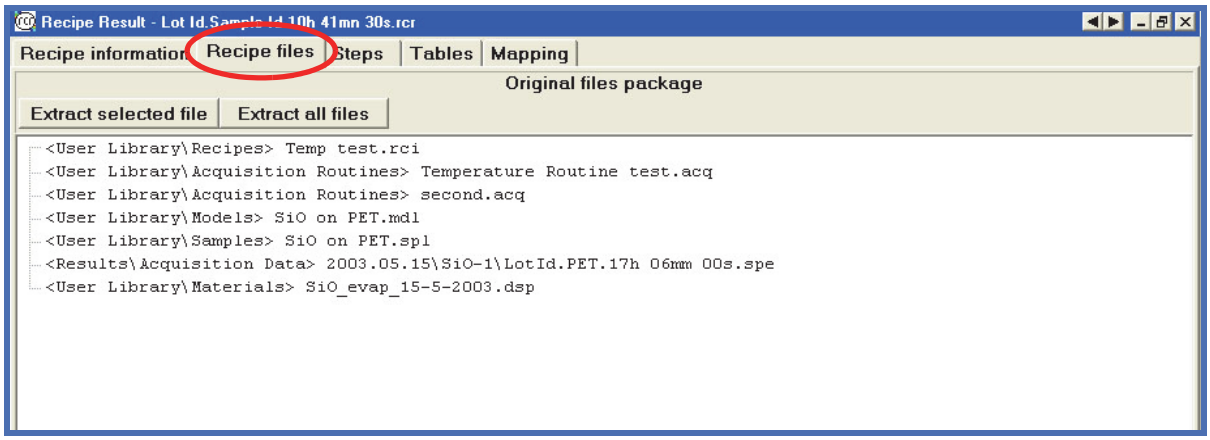
# 19 Reading Recipe Results files (.RCR)

As soon as a **Results Recipe** file has been created, the parameters can be consulted. Double-click on the .RCR file to display these results. Several parameters windows are available. The Result tab, located on the top allows the user to navigate along the results.

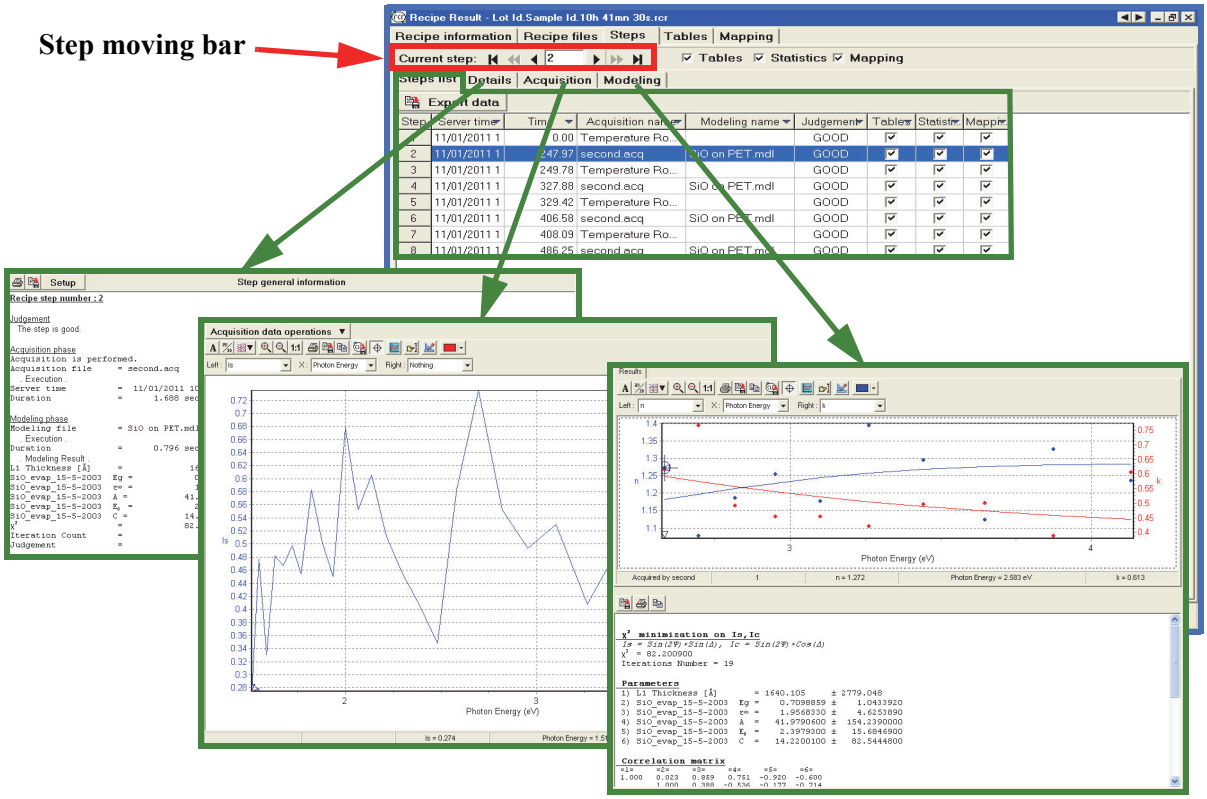
**Recipe information**: the **Recipe information** window groups the general information about the Recipe analysis conditions.



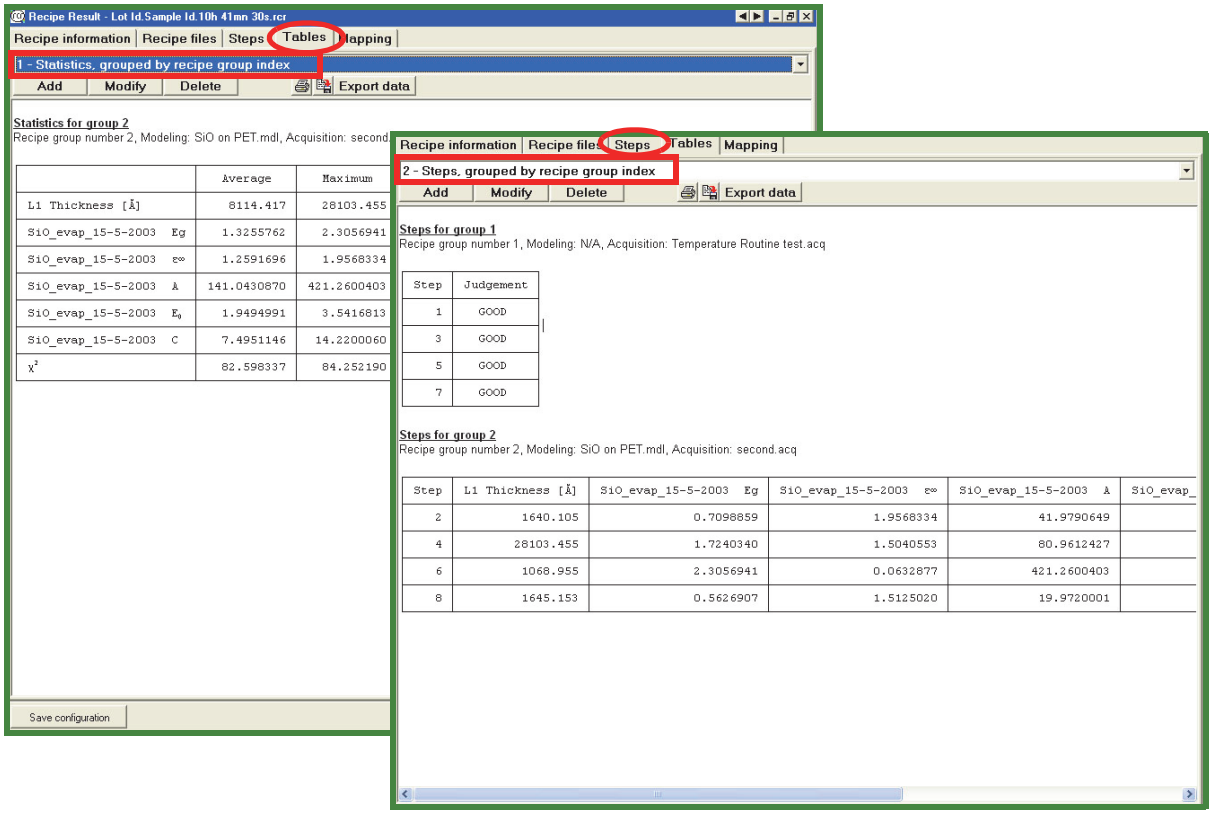

**Recipe files**: the Recipe files window lists all the files used by the Recipe.



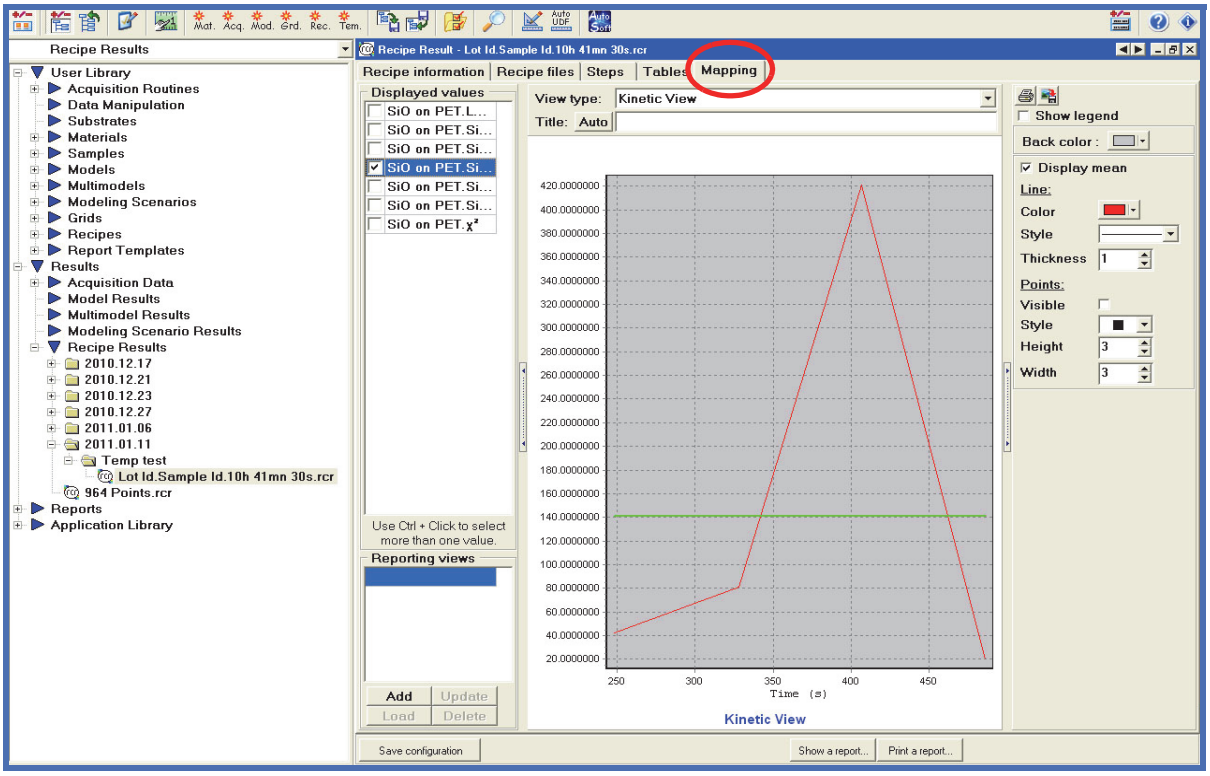
**Steps**: the ***Steps*** tab shows the complete information by steps from the steps sequencing to the sample ***Acquisition*** and ***Modeling***.




**Tables**: the **Tables tab** displays the results by step and by parameter and also statistics according to the previous entered criteria.



**Mapping**: the **Mapping** tab shows a 3D presentation of the Recipe. Of course, a real mapping with many acquisition points will render the overall view more usable.



# 20 Reporting

The Reporting feature is the final step of a DeltaPsi2 session. It allows to select useful information, organize it and generate it for the final report.

Two steps are required to edit a report:

1 Create a template. Once created, it will be used each time to create the same layout for the report.

From the Model screen, launch the reporting feature to generate the reporting files.

## 20.1 Creating Template

DeltaPsi2 allows the user to create different types of templates according to the needs or the company graphics chart. To create a template, follow the steps described below:

- From the ***User Library*** node, select the ***Report Templates*** choices.

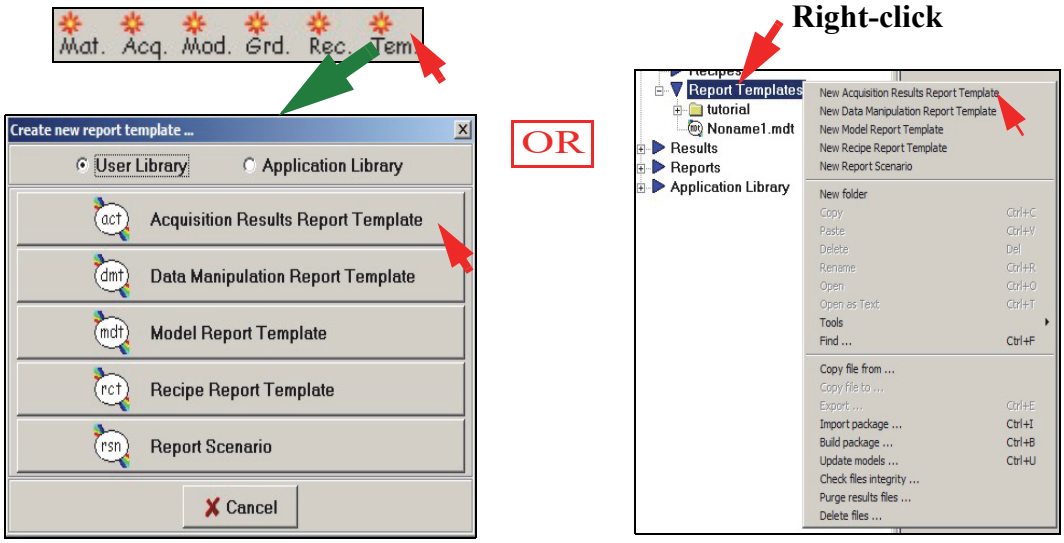



**Figure 20-1**

As shown above, different categories of ***Report Templates*** are available. If fact, each category has its own set of parameters which could be useful to export to a template.

The ***Report Scenario*** can chain and group several categories of report template.

## 20.2 Template Setup

The Template setup is exactly the same for all of the template categories, only the parameters to display are different.

- Click on the ***Acquisition Results Report Template*** button; the following reporting screen will be then displayed:

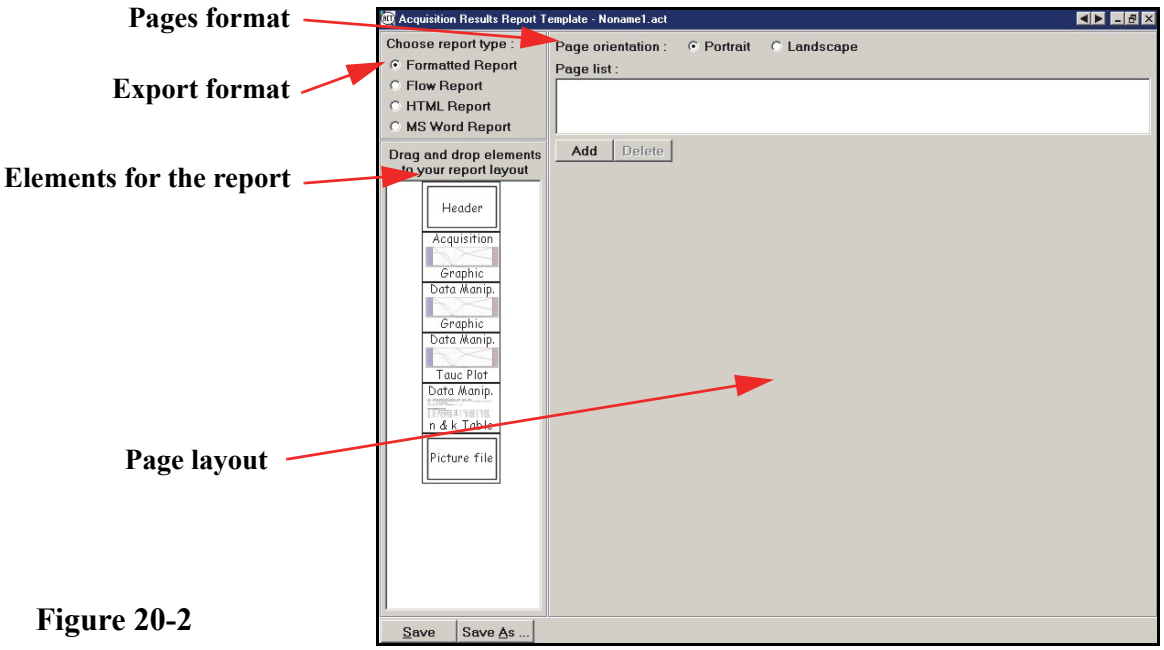



**Figure 20-2**

- Choose first the export format. This format is important and depends on the use of the report. Four formats are available:

---

**Formatted Report:** This is a proprietary HORIBA Scientific format. That means that no other softwares are required to print a report.

The procedure below explains step by step how to create a ***formatted*** template.

**1** Select the ***Formatted Report*** type from the upper left hand side of the screen,

Select the page orientation option: Portrait or Landscape

Click on the Add button to create a new page

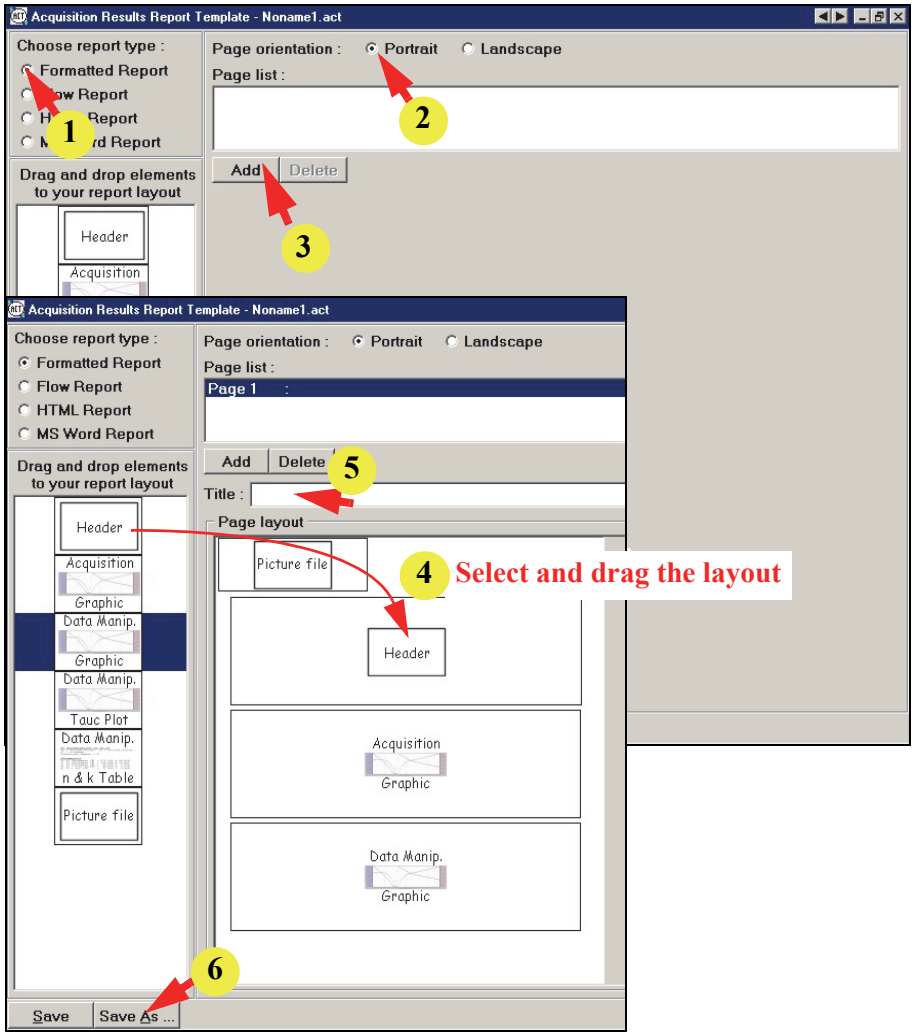
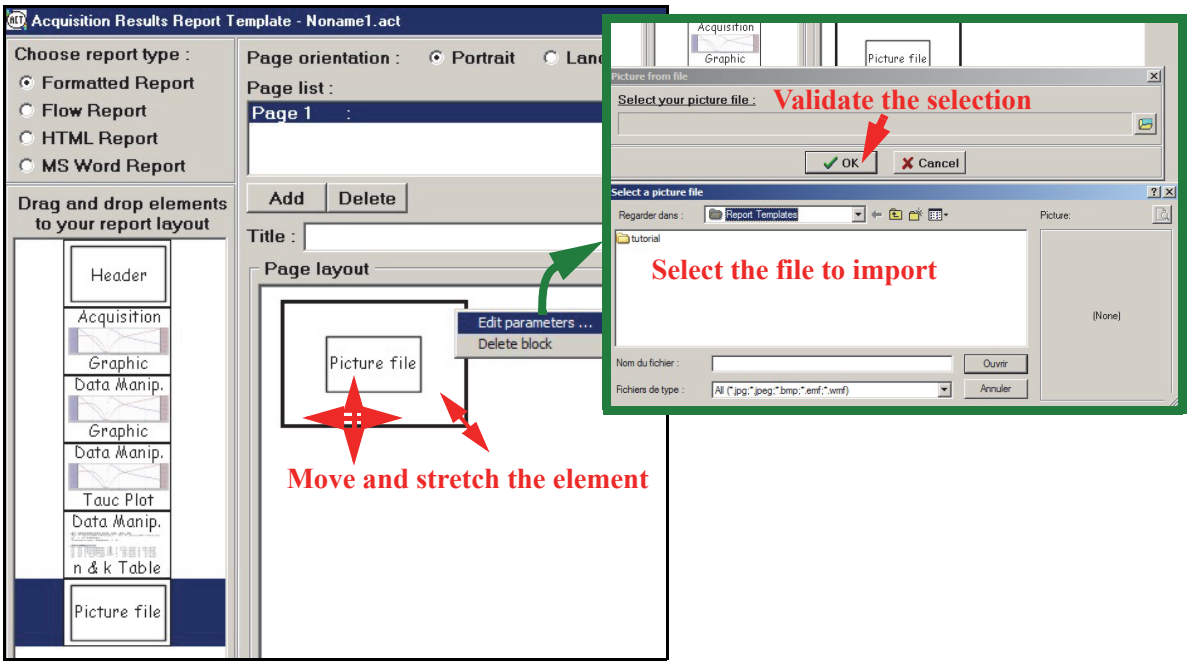



**Figure 20-3 Building formatted report**

Select and drag the element to the page layout, then using the mouse stretch this element over the page layout.

The elements can be placed everywhere on a page and even superimposed.

The special ***Picture File*** element can contain an image. Accepted graphics format files are JPG, BMP, WMF and EMF. To insert a file in the Picture File element, right-click on the element and select the file.




Enter the page title. This title will be printed on the top of the page,

Click on ***Add*** (see step 3) to add a new page then proceed as above. Add as many pages as necessary to create the report,

Click on the ***Save As*** button to save the built report type.

## How to use the created report template ?

See the chapter "Generating a report" on page 285

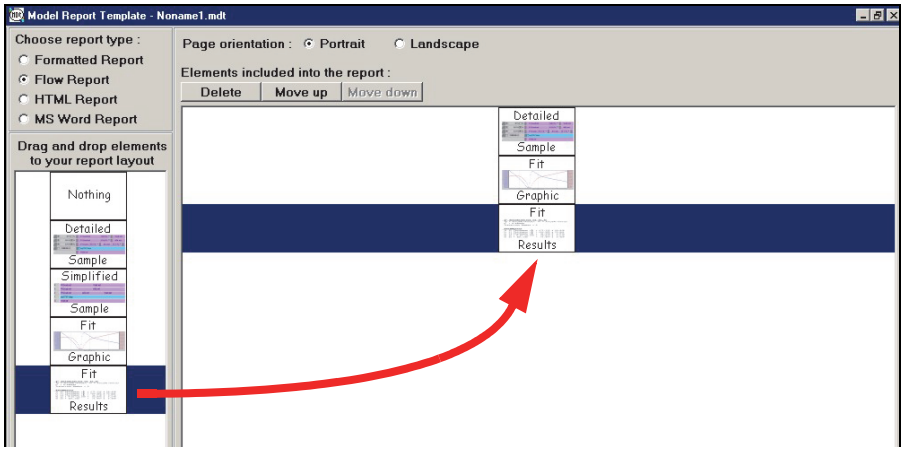
---

**Flow Report:** As the Formatted Report, the Flow Report is a HORIBA Scientific proprietary format. That means that no other software is required to print a report. Comparing to the ***Formatted Report*** type, the ***Flow Report*** is less versatile: the elements cannot be stretch or superimposed on the page layout.

**1** Select the ***Flow Report*** type from the upper left hand side of the screen,

Select the page layout ***Portrait*** or ***Landscape***,

As shown on the figure below, select, drag and drop the report layouts.



**HTML Report:** This format will generate an HTML page description which can be useful for Web publishing. Moreover, this export format can convert the figures to JPEG format. The procedure below shows how to create this type of report:

**1** Choose the HTML Report and the following screen will be displayed:

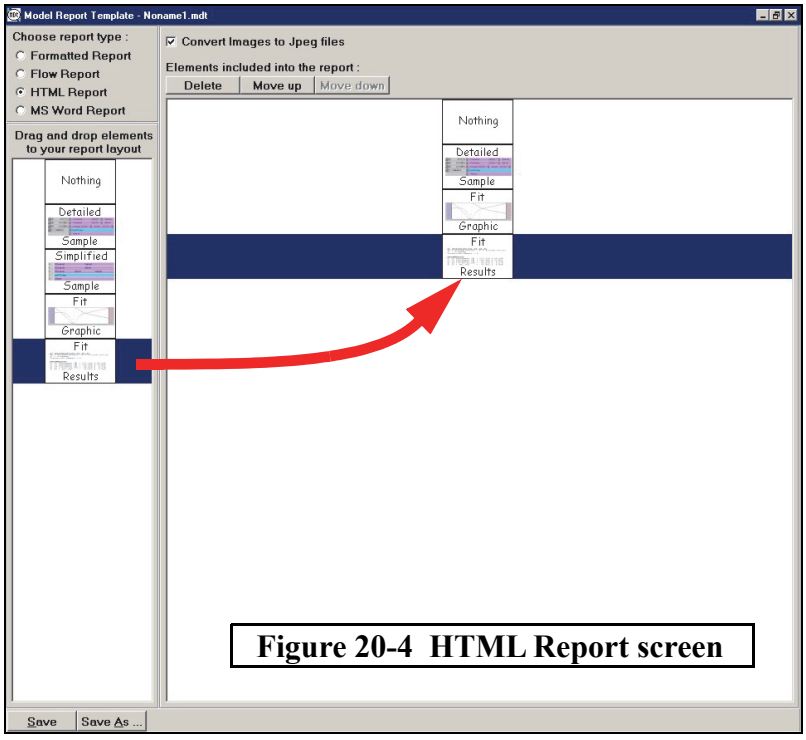



**Figure 20-4 HTML Report screen**

Check in the «***Convert images to Jpeg files***»: the .Jpeg format is usually the standard format for HTML files, otherwise the .WMF will be used. The .WMF format is one of the most elaborate graphics formats but, compared to the Jpeg format, it can be limited in that only the Windows® Internet Explorer can be used to open the HTML files linked to .WMF figures. The others Internet browsers do not accept this format.

Using the mouse, drag and drop the predefined graphic elements to the ***Page Layout*** screen. The ***Delete*** and ***Move up*** buttons help the user to change the layout.

The special ***Picture File*** element can contain an image. Accepted graphics format files are JPG, BMP, WMF and EMF. To insert a file in the Picture File element, right-click on the element and select the file.

**3** As soon as the report is ready, click on the ***Save As*** button and enter the name of the new template. If an existing template has been opened to be modified, click on the ***Save*** button.

To use the created templates, follow the chapter "Generating a report" on page 285.

**MS Word® Report:** This export format is useful for users who own the Microsoft Word® software. This export format is certified for Microsoft Word® 97 and 2000. Due to multiple Word® versions over the Word, we cannot guarantee a good result on every Word® version. On the export phase, DeltaPsi2 will open and manage the Word software layout.

Once the ***MS Word Report*** is selected, the following screen will be displayed:

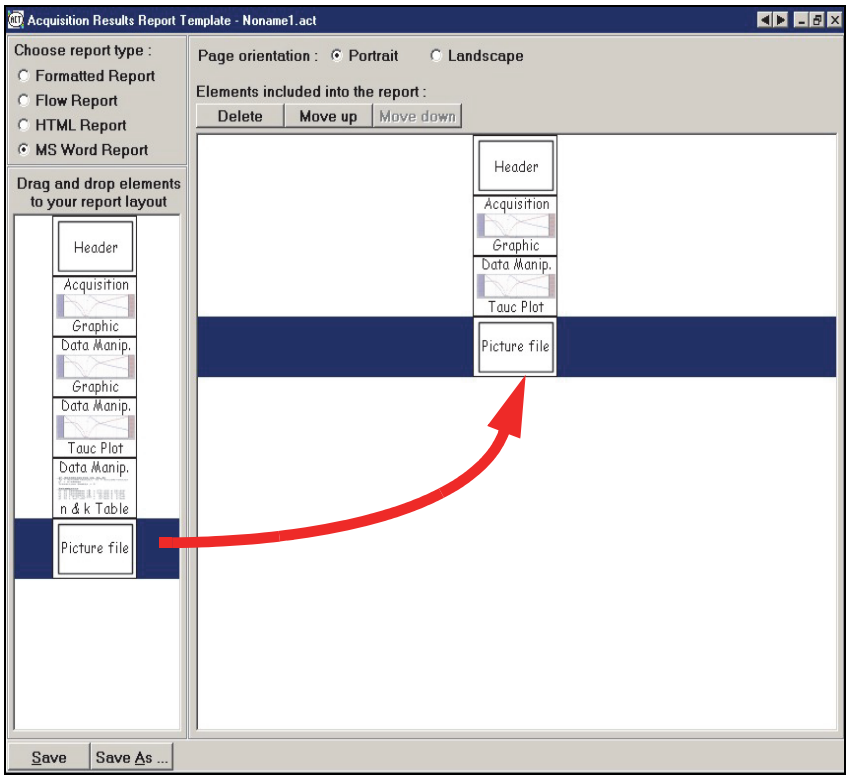



**Figure 20-5 MS Word® Report**

**1** Choose the page orientation: ***Portrait*** or ***Landscape.***

Using the mouse, drag and drop the predefined graphic elements to the ***Page Layout*** screen. The ***Delete*** and ***Move up*** buttons help the user to change the layout.

The special ***Picture File*** element can contain an image. Accepted graphics format files are JPG, BMP, WMF and EMF. To insert a file in the Picture File element, right-click on the element and select the file.

As soon as the report is ready, click on the ***Save As*** button and enter the name of the new template. If an existing template has been opened to be modified, click on the ***Save*** button.

## 20.3 Generating a report

The Report creation feature can be activated from the main Model screen. Here below is an example of such of screen:

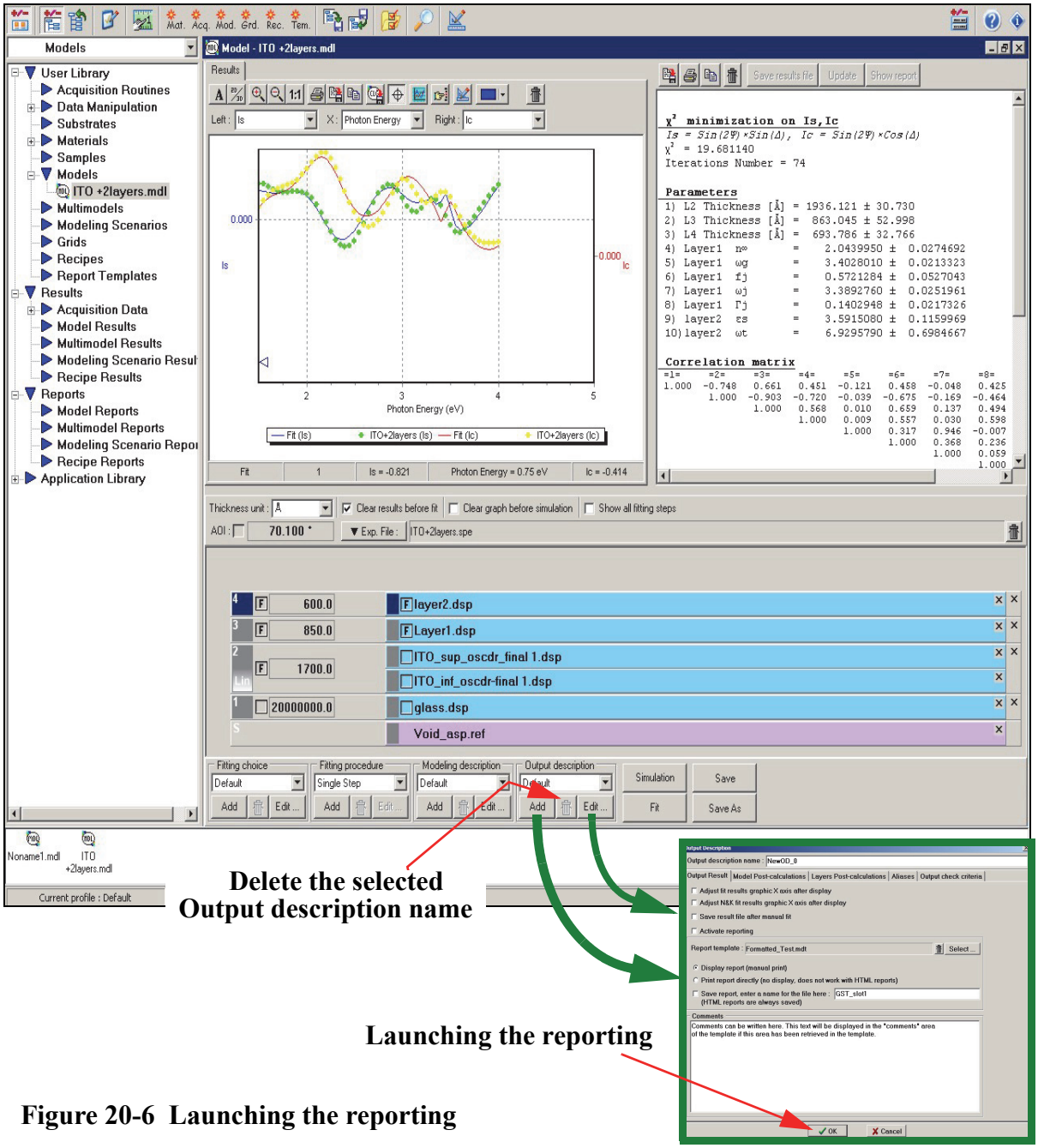



**Figure 20-6 Launching the reporting**

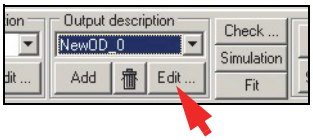
On the ***Model*** screen, the part labelled Output description concerns the reporting section.

Click on the Edit button to open the Reporting setup screen. Enter or check the options according to the description shown on the figure below:

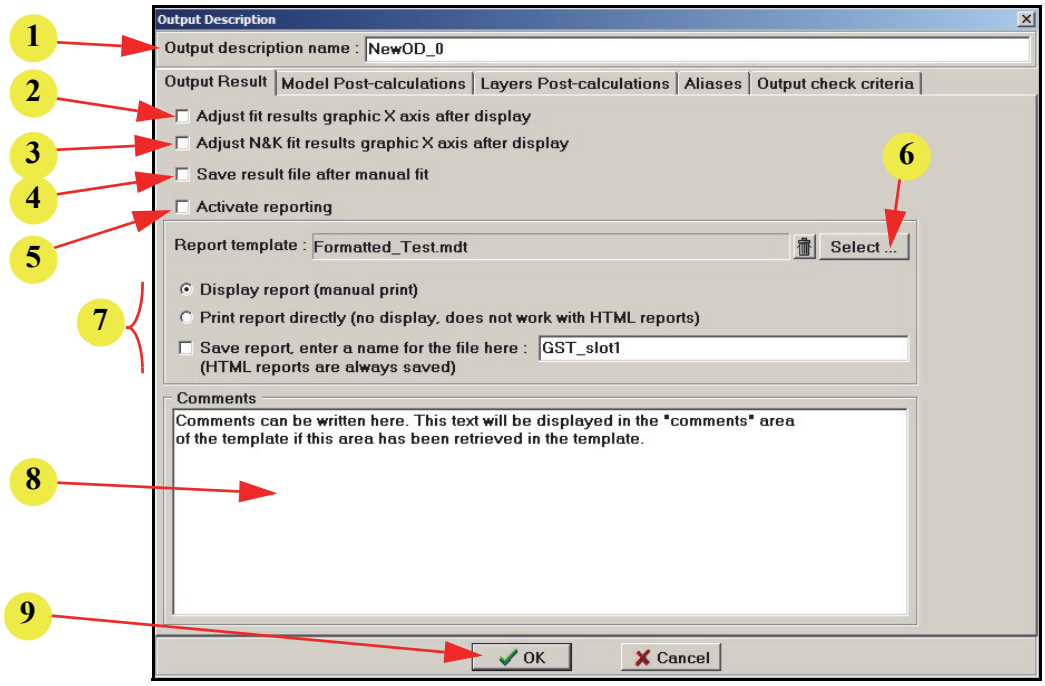
### 20.3.1 How to use the created report template ?

The previously created report template can only by activated from the general Modeling screen. When a model has been prepared and ready to be fitted, the user can activate the ***Reporting*** feature. The procedure below explains how to activate it.

This procedure will only show how to activate the created template. In fact, the ***Output Description*** screen includes two important additional tabs which are described in the chapter "Advanced Model Post-calculation features" on page 206. On the bottom the main Modeling screen, click on the ***Edit button*** of the ***Output Description*** part.




4 The following ***Output Description*** screen is then displayed:




1 ***Output Description name***: Enter a name which will memorize the parameters entered in this screen.

2 If a Customize X-axis layout has been set (see figure 3-3, page 36), the Fit results will keep the same layout. If the box is check-in, this customize layout will be ignore, and the Fit results will display the full scale.

3 Same as above but for the n&k fit results screen.

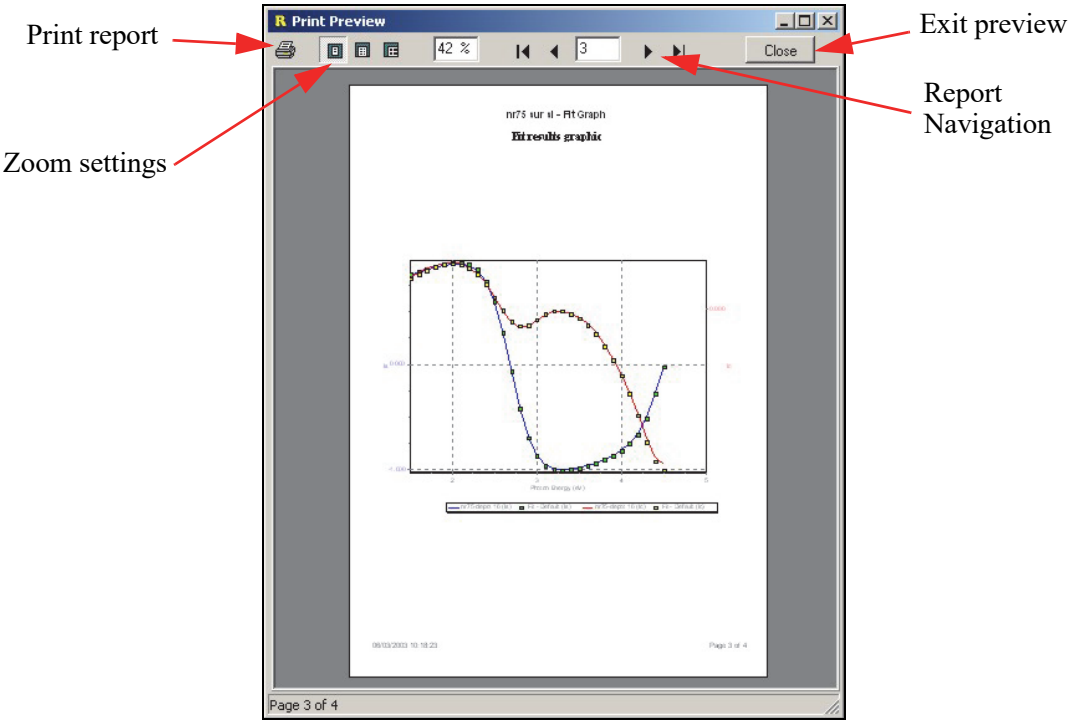
4 If checked-in, every manual fit results will be saved.

5 ***Activate reporting***: check in to activate the reporting feature.

6 Select the ***report template*** which has been previously built.

7 ***Output reporting options***: three different outputs can be selected:

1 The ***Display report*** option will open a window and the report can be previewed before printing. The figure below shows an example of a ***Print Preview*** screen:




2 The ***Print report*** option will print immediately the report using the current printer installed from the Windows® operating system.

3 The ***Save report*** option will create a file which can be printed later. The file name must be entered and the file will be stored in the ***Reports*** folder of the TreeView. Then, double-click on this file to open the report ***Print Preview***. The HTML reports are always saved.

**8** Enter the comments in this area. If the report template has been prepared with a comments area, the entire text will be retrieved in the report.

**9** Click ***OK*** to validate the Output description screen.

Once the ***Output Description*** parameters screen has been setup, the report will be automatically created after a Fitting process.

**CAUTION**: If an MS Word® template has been selected to create the report, **the Word® application must be closed**. DeltaPsi2 will open it by itself. If the ***Save report*** has been clicked on, the generated report will automatically be saved in the ***Report*** node with the .DOC extension.

**NOTICE:** the HTML format can only be saved. The ***Display*** and ***Print*** features are not yet implemented.

# 21 Kinetic Measurements

## 21.1 General

The kinetics acquisition allow the study of rates of Physical/Chemical processes. Physical/Chemical kinetics includes investigations of how different experimental conditions can influence the speed of a reaction and yield information about the reaction's mechanism and transition states, as well as the construction of mathematical models that can describe the characteristics of a physical/chemical reaction.

DeltaPsi2 software includes the complete kinetics feature from the acquisition to the real time modeling with end point.

### *Requirements*

The Kinetics Feature is available on UVISEL Series and MM-16 Series ellipsometers. This feature can also be used on Auto SE Series from DeltaPsi2 Software.

### *Working Method*

Kinetic feature can be performed at several levels depending on your goal and your System availability.

- If you want to obtain results as soon as possible and your System is ready to be used, follow the steps below:

  -"Acquisition Routine" on page 290,

  -"Model Creation" on page 297,

  -"Kinetic Recipe" on page 303.

- Another working solution is to perform acquisitions on the fly on one or several samples, then, later, manipulate the Models and apply the fitting process. Excepting the acquisition step, the other processes can be done on another computer (without the System connected). In this case follow the following steps:

  -Acquisition Routine and acquisitions on the fly (see "Acquisition Routine" on page 290),

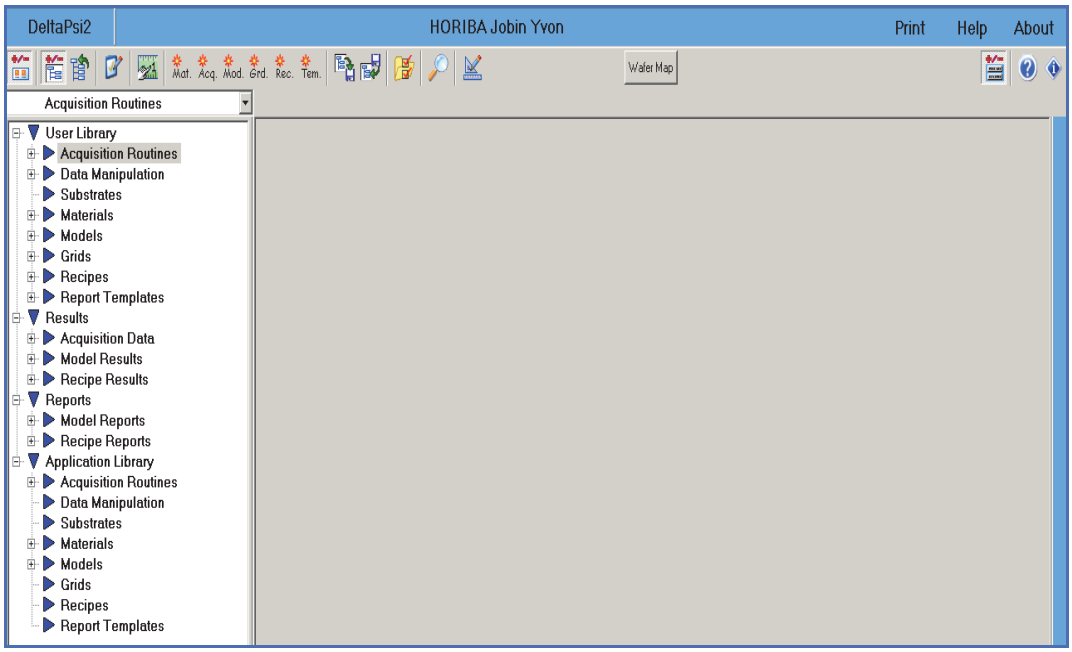
  -Model Creation with experimental files import and fitting process. See "Model Creation" on page 297 then "Fitting Process" on page 304.

- If you want to prepare your project on a standalone computer, you can simulate the Ψ and Δ result values according to a layer deposition thickness rate. See "Simulation" on page 303.

# 21.2 Acquisition Routine

When a sample must be analyzed with some physical/chemical changes, it becomes very important to define the time scale events. This chapters will detail how to prepare an acquisition routine for a fast and reliable results.

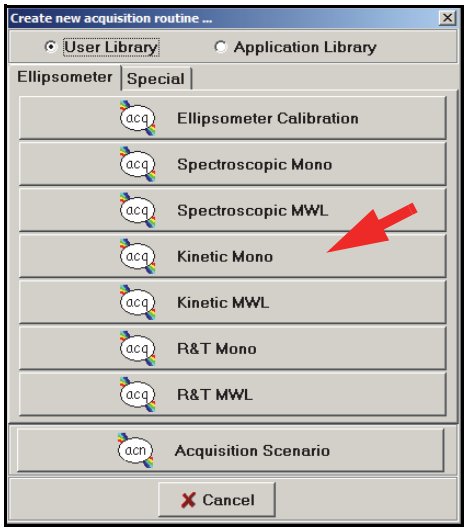
- **Run the DeltaPsi2 Software. The following screen will be displayed:**




- **Kinetic Acquisition can be launched from several locations in the DP2:**

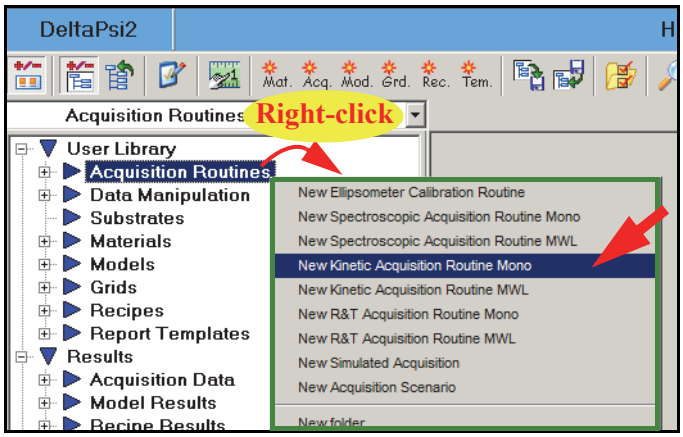
## For UVISEL with Monochromator

From the Main Menu, click on the ***Acq*** icon  , then select the ***Kinetic Mono*** (monochromator) choice.




**OR**

From the TreeView list, right-click on the ***Acquisition Routines***, located in the User Library. Then select the ***New Kinetic Acquisition Routine Mono*** choice.



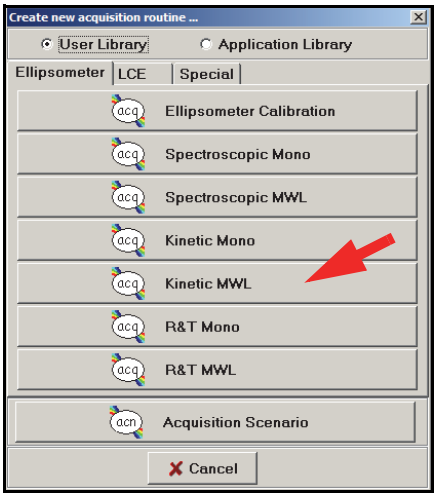

05/20/2016

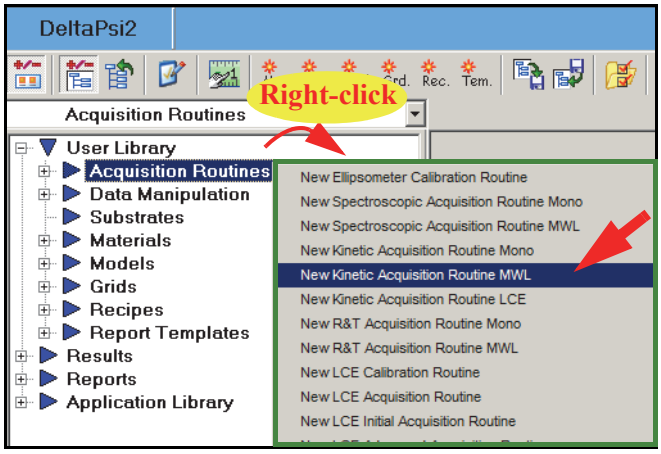
## For UVISEL with MWL

From the Main Menu, click on the ***Acq*** icon  , then select the ***Kinetic MWL*** (**M**ulti**W**ave-**L**ength) choice.

**OR**

From the TreeView list, right-click on the ***Acquisition Routines***, located in the User Library. Then select the ***New Kinetic Acquisition Routine MWL*** choice.



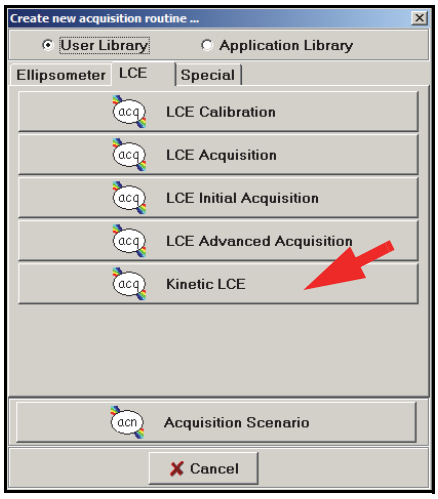



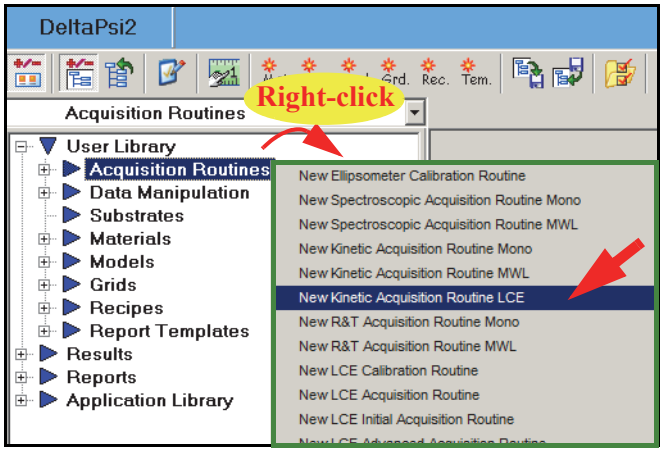

## For MM-16 Ellipsometer

From the Main Menu, click on the ***Acq*** icon  , then select the ***Kinetic LCE*** (**L**iquid **C**rystal **E**llipsometer = MM-16) choice.

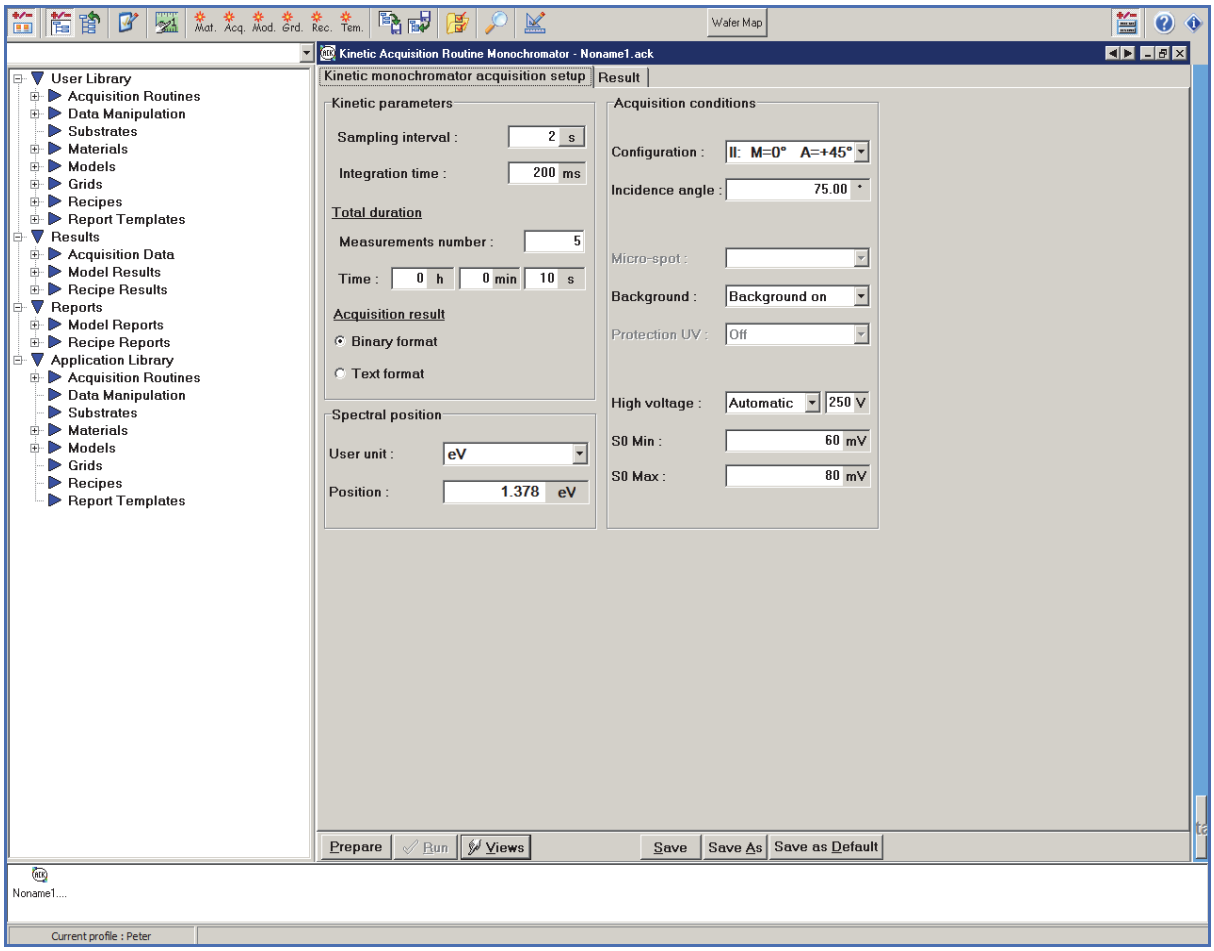
**OR**

From the TreeView list, right-click on the ***Acquisition Routines***, located in the User Library. Then select the ***New Kinetic Acquisition Routine LCE*** choice.



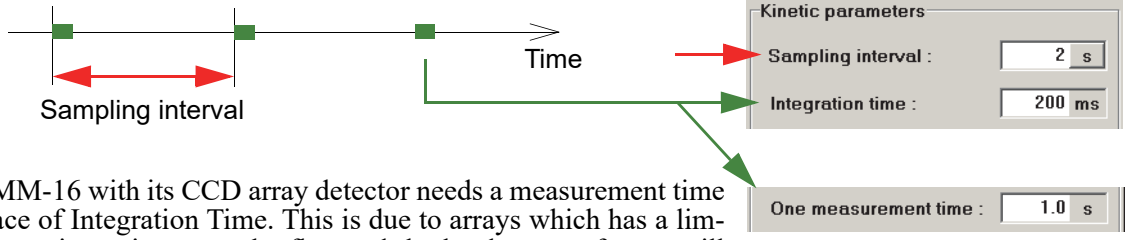
- **The following Kinetic Acquisition screen will be displayed:**




This configuration screen includes all of the parameters required to build the acquisition sequence. We will detail each of them.

05/20/2016

**Kinetic Parameters**

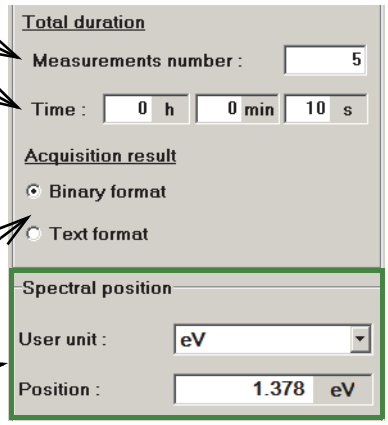



The MM-16 with its CCD array detector needs a measurement time in place of Integration Time. This is due to arrays which has a limited capacity to integrate the flux and the hardware+software will calculate and optimize the integration time and the accumulation time according to the sample.

Enter here the number of measurement you want to perform

**OR** enter here the total time of the experiment

**NOTICE: All the screens are interactive; for example if 5 measurements are chosen, the total time will be adjusted according to the Sampling interval and the Integration time.**




By default, **Binary** result format is the highest precision format as it records the full 32-bit floating point data.
The **Text** result format uses the precision entered in the ***Observables precision*** located in the ***Software options*** part.



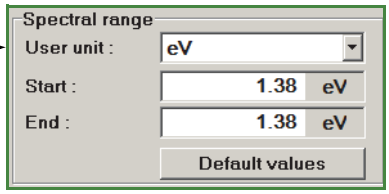

***Spectral Coverage setup:***

➠ For UVISEL with Monochromator

Using a monochromator, kinetic acquisition are only possible for one wavelength. The user unit (the choices are eV, cm-1, nm, A, µm). Select a wavelength within Instrument coverage.

➠ For UVISEL with MWL

The MWL or MultiWaveLength spectrograph can simultaneously analyze several wavelengths (up to 32). See the wavelengths values list in the ***HV values*** Tab. Enter the Start/End wavelengths.



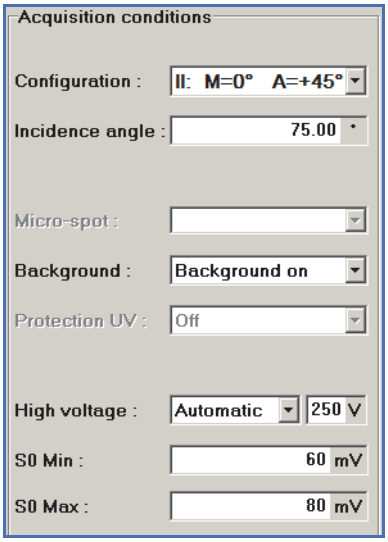

➠ For MM-16

The used Spectral Coverage is always the maximum range, as the detector is a CCD chip. Thus there is no range selection.

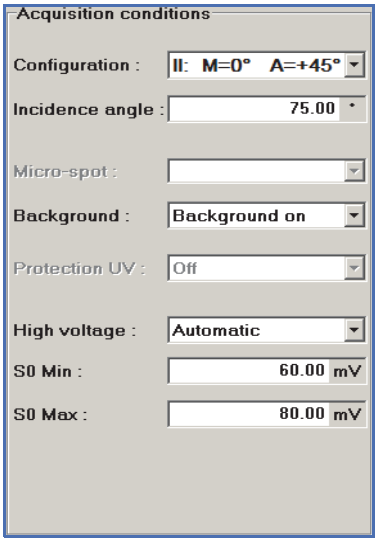
05/20/2016

## Acquisition Conditions

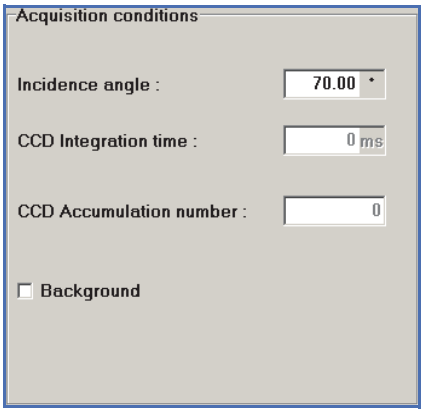
**UVISEL with Monochromator**




**UVISEL with MWL**



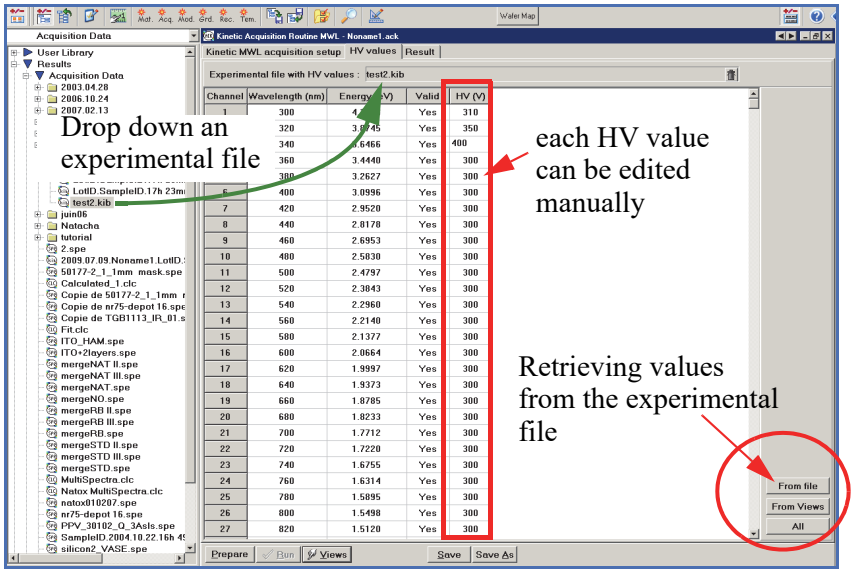

**For MM-16**




| | |
|---|---|
| **Configuration** | Choose the appropriate configuration choice. For Kinetic acquisition where the time is very important, the "no merge" method is used by default. The user can choose the Configuration I (M=0°, A=0°), II (M=0°, A=+45°) or III (M=-45°, A=+45°).<br>By default, the Configuration II is used. |
| **Incidence angle** | Enter the incidence angle: angle determined between the Modulator, the sample and the Analyzer.<br>For information:<br>- Measurements for most semiconductors are usually made around 70°-75°,<br>- Measurements on transparent materials are made at around 60°. |
| **MicroSpot** | The beam diameter can be set on the **Micro-Spot** sub-unit. The choices are 1200, 250, 120 and 60 µm. |

*05/20/2016*

| | |
|---|---|
| **Background** | A high background level can mask weak features and reduce the dynamic range of the Real Time Control.<br><br>**Before applying the «background» feature, be sure that the offsets calibration procedure has been performed. This calibration should be performed at least once a year or in case of important changes of the system, e.g. PMT replacement.**<br><br>**Background off:** performs a noise background measurement before an acquisition. This noise background will be subtracted from each data point acquisition.<br><br>**Background on:**<br><br>If the Background box is checked, for each data point, two acquisitions will occur. The system will close the shutter located on the light source path and perform an acquisition. A signal plus a dark acquisition will be performed with the shutter open. The dark acquisition will be subtracted from the signal plus the dark acquisition. This technique minimizes the effect of any drift conditions that may occur. |
| | **Automatic Background:**<br><br>If the Offsets calibration has been performed, a results file (background.ini) has been saved on the hard disk. If an acquisition is launched using this mode, the [$S0_{background\ noise}$ (HV)] will be subtracted from the acquisition data. |
| **Protection UV (UV Option)** | Choice is ON or OFF<br><br>**ON:** activates a filter to protect the UV sensitive samples. |

| | |
|---|---|
| **High Voltage** | **UVISEL with Monochromator:**<br>The user has the choice between two High Voltage modes:<br>• **Fixed**<br>Enter the fixed high voltage value in the related field (located on the right). During the acquisition, the high voltage value will not be changed. This is necessary when fast kinetic acquisition is performed.<br>• **Automatic**<br>The software performs an automatic feedback control on this relation: HV = f(S0) using a calibrated HV curve and the S0 measured value. The High Voltage Field set the **minimal** HV Value.<br>**UVISEL with MWL:**<br>As MWL includes many PMTs, a good idea is to fix the High Voltage values for each PMT to avoid inconsistent acquisition signal when fast kinetic acquisition is performed.<br>These fixed High Voltage values can be adjusted manually or retrieved from an existing experimental file. The figure below shows the window from which this action can be done.<br>  <br> |
| **S0 Min / Max** | Choose a S0 range (minimum and maximum) in which the High Voltage does not change. |
| **CCD Integration Time** | For MM-16 only. When the Kinetic acquisition is setup, the CCD Integration Time is just an information field which becomes active only after the ***Prepare*** step. The integration time is automatically calculated according to the kinetic timing. |
| **CCD Accumulation number** | For MM-16 only. Information field. According to the kinetic timing and the CCD Integration Time, the ***Accumulation number*** will be automatically calculated. |

- **There are several buttons on the acquisition bottom screen:**


The classical "Save" buttons to save an acquisition configuration for further use.

Switch to the Real Time (Views) Acquisition screen: it allows to verify the acquisition level on a sample.

Become active when the ***Prepare*** button has been activated. Pressing the ***Run*** button will activate the kinetic acquisition program.

Once the Acquisition Configuration parameters has been completed, click on this ***Prepare*** button. This action is available only for kinetic because the software tests the hardware and retrieves the real Integration Time value and prepare the experiment to run the kinetic acquisition to minimize the spare time.

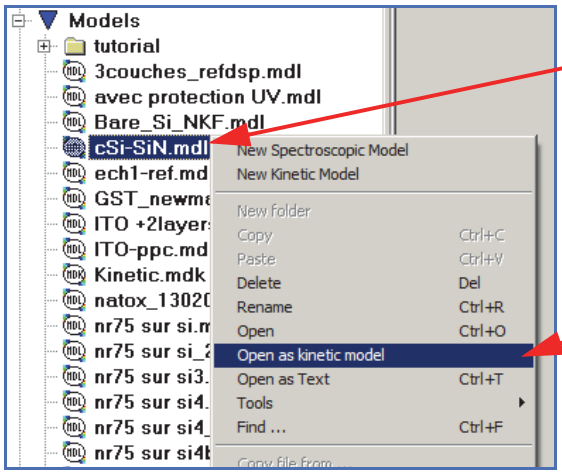
Clicking on the ***Run*** button will launch the Acquisition measurement. The created file is called **Experimental file** and will be recorded in the **Result\Acquisition data** folder with a **.kib** extension.

## 21.3 Model Creation

This chapter describes how a kinetic model is created.

### *Conversion to Kinetic*

The structure of the Kinetic Model defers from the standard spectroscopic. If a spectroscopic model must be used in a kinetic measurement, the file must be converted. The procedure is very simple and described on the figure below:




Right-click on the file to convert

Select this choice to perform the conversion

**NOTICE**: the conversion can also be performed from kinetic to spectroscopic model. Procedure is the same.

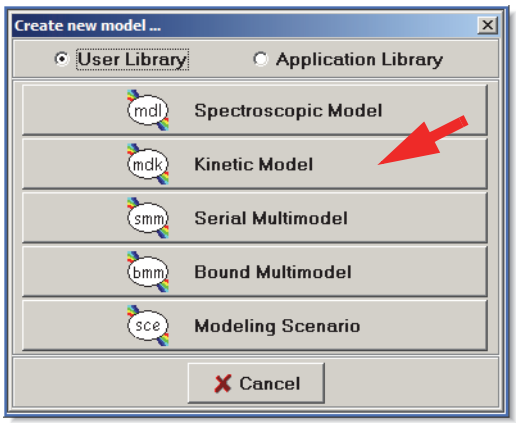
05/20/2016

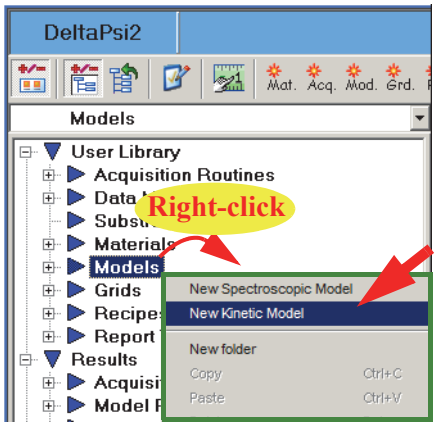
## *Getting Started*

From the Main Menu, click on the ***Mod.*** icon  , then select the ***Kinetic Model*** choice.

**OR**

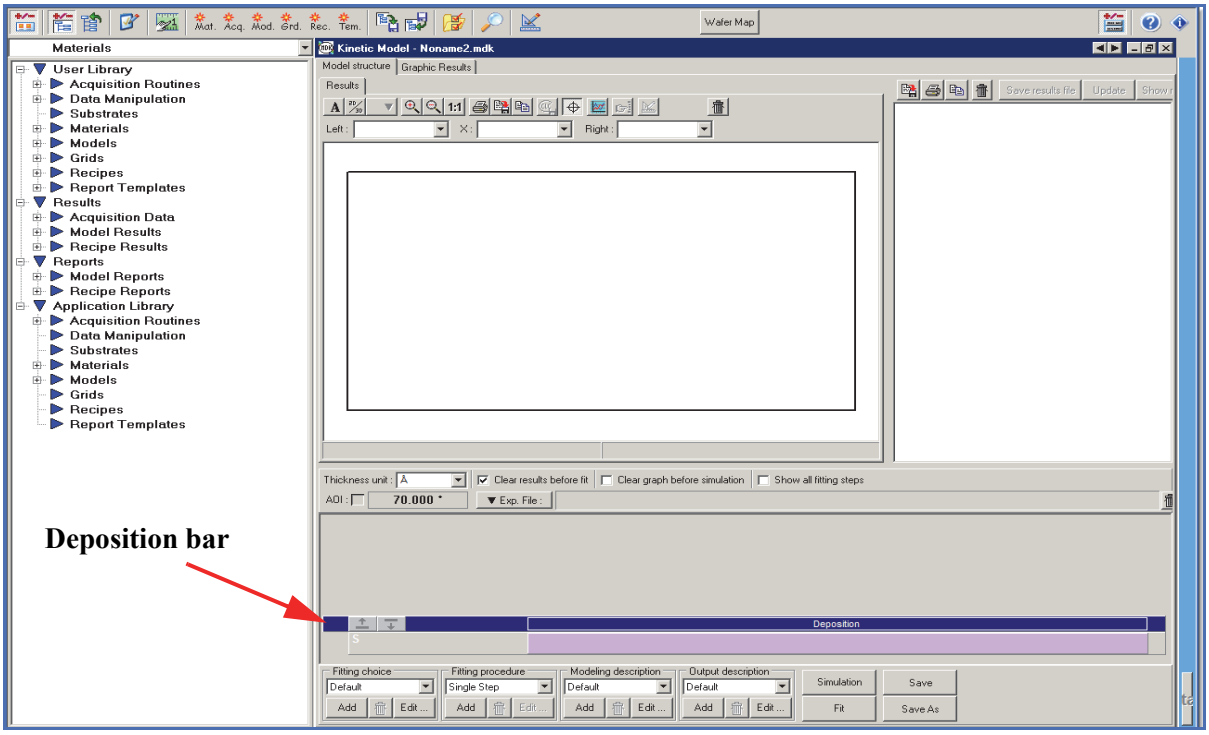
From the TreeView list, right-click on the ***Models*** folder, located in the User Library. Then select the ***New Kinetic Model*** choice.



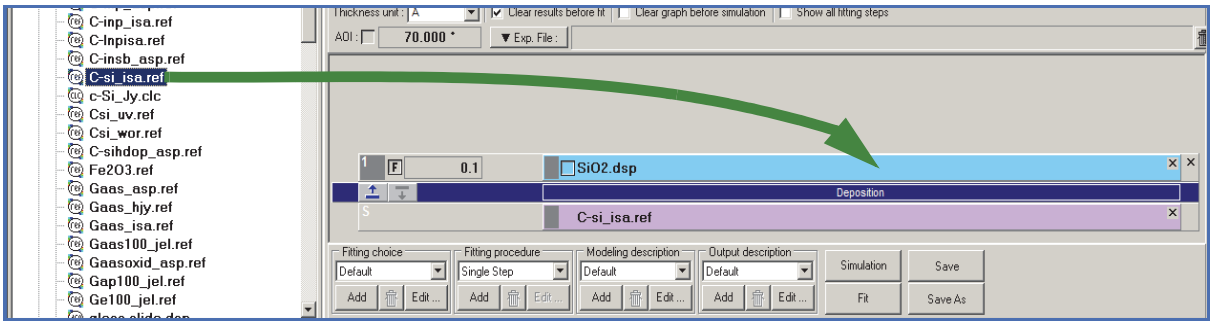

## *Setup*

The following start screen will be displayed. We can notice that, excepting the ***Deposition*** bar, there is no difference with the other Model Creation profiles.



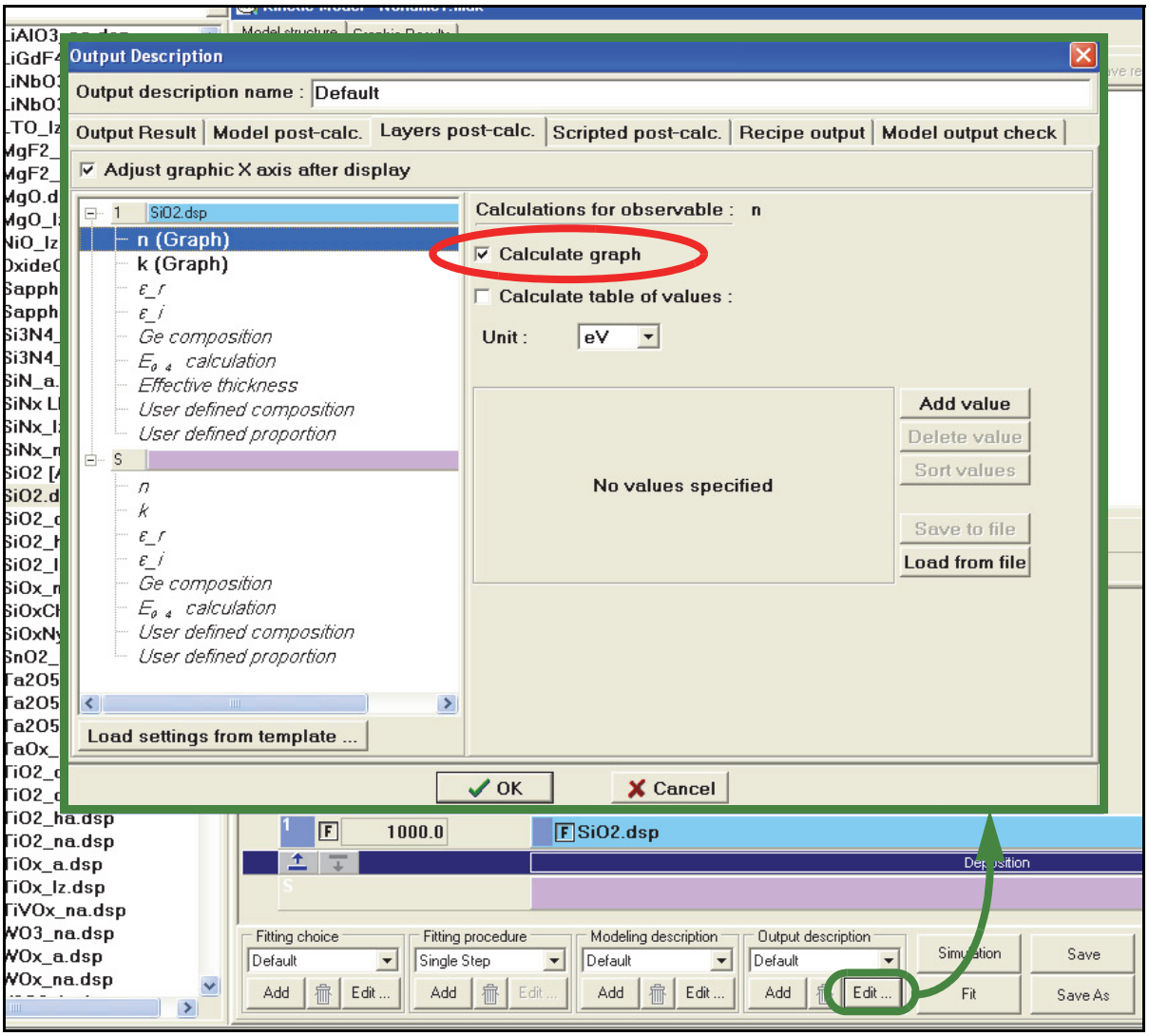

05/20/2016

- **The Kinetic Model can be composed of a substrate, several fixed layers and a layer with a changing thickness which must always be on the top. As for the others models, drag the material over the substrate and then the layer**




- **Output Description Setup**

  Before to continue the explanation of the kinetic feature, verify the Output Description conditions. These conditions are important because it could impact on the availability of parameters. For example, to display the n&k parameters on the final output, it is important to check the **n and k** as ***Calculate graph*** (see the figure below).




The ***Deposition*** bar can be moved through the layers. Actually, only one layer can be fitted as a deposition; the figure above shows this working structure. Future upgrade will extend the use the layer(s) etching.

05/20/2016

The figure below shows the deposition configuration.

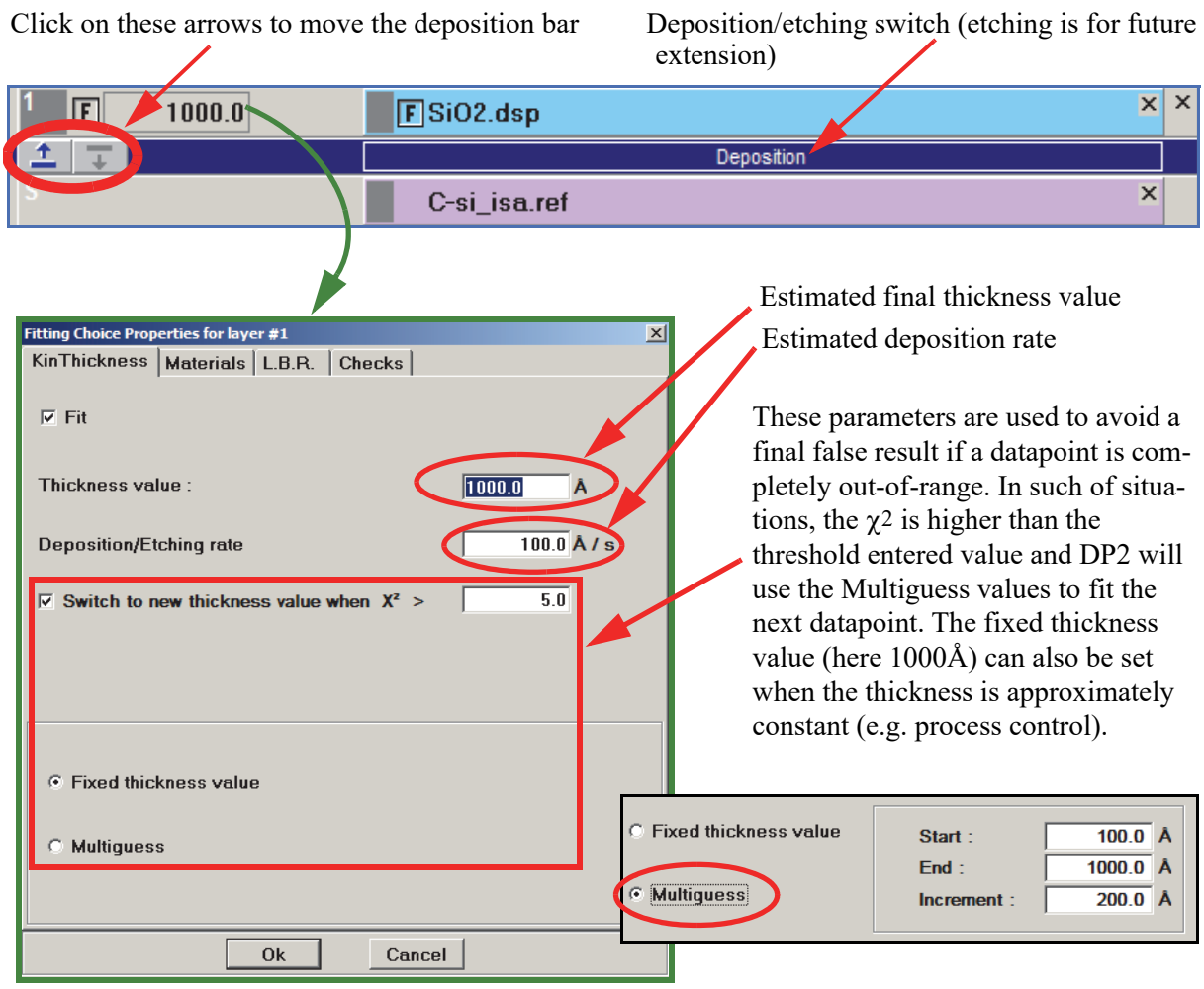
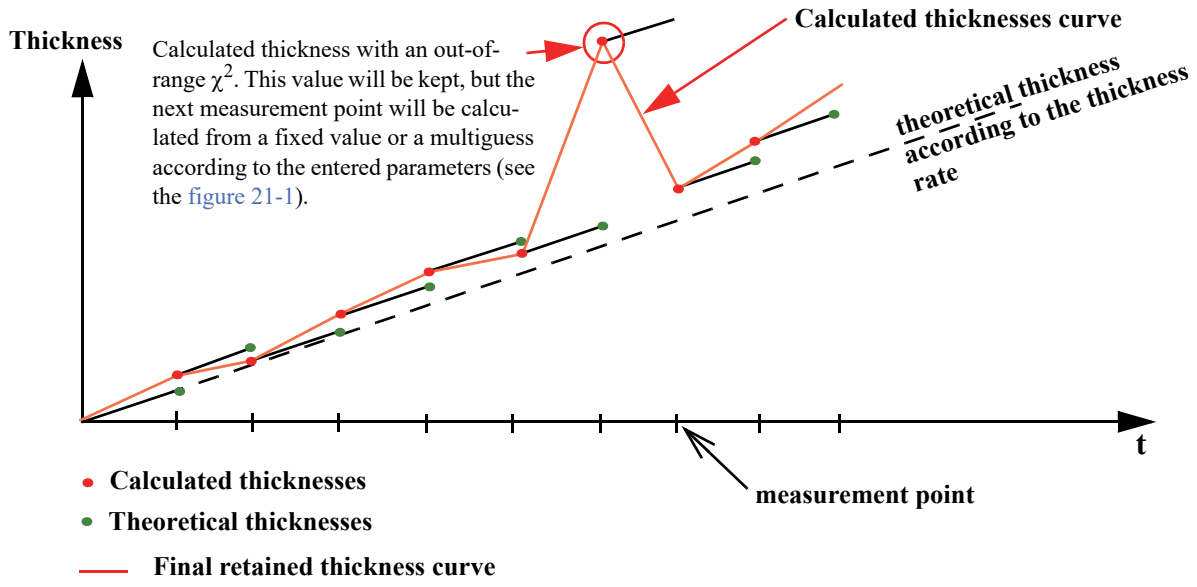



**Figure 21-1 Fitting choice properties**

The following figure shows an example of a kinetic measurement.

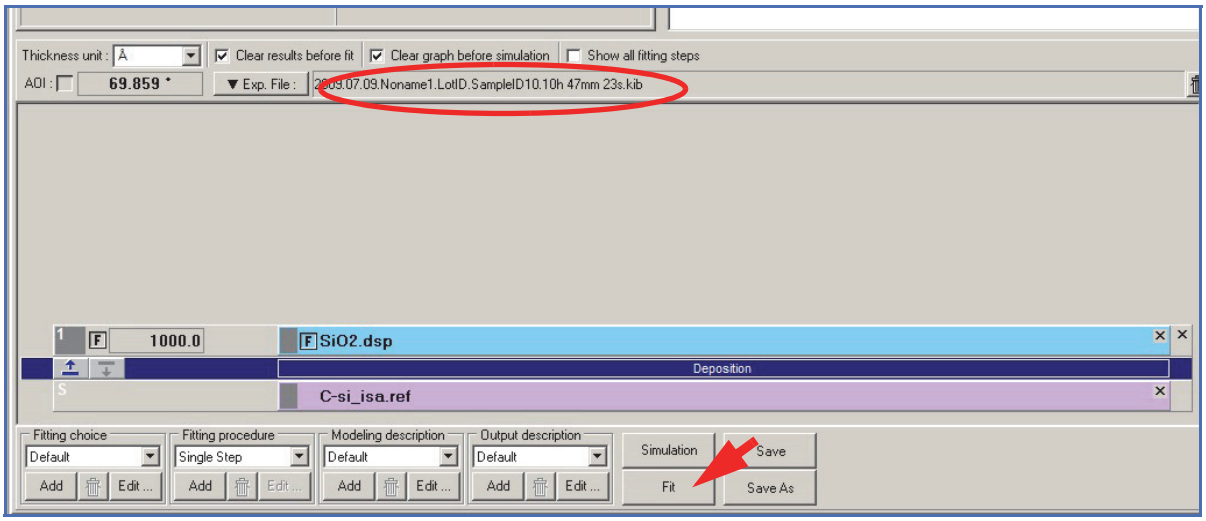



**IMPORTANT**: The «**Multiguess thickness value**» will be chosen for a growing thickness (deposition).

The «**Fixed thickness value**» will be chosen when the thickness is approximately constant during the kinetic measurement (e.g. process control..).

- **Once the Model has been built and fitting choice properties defined, it becomes possible to fit it using the following two methods:**

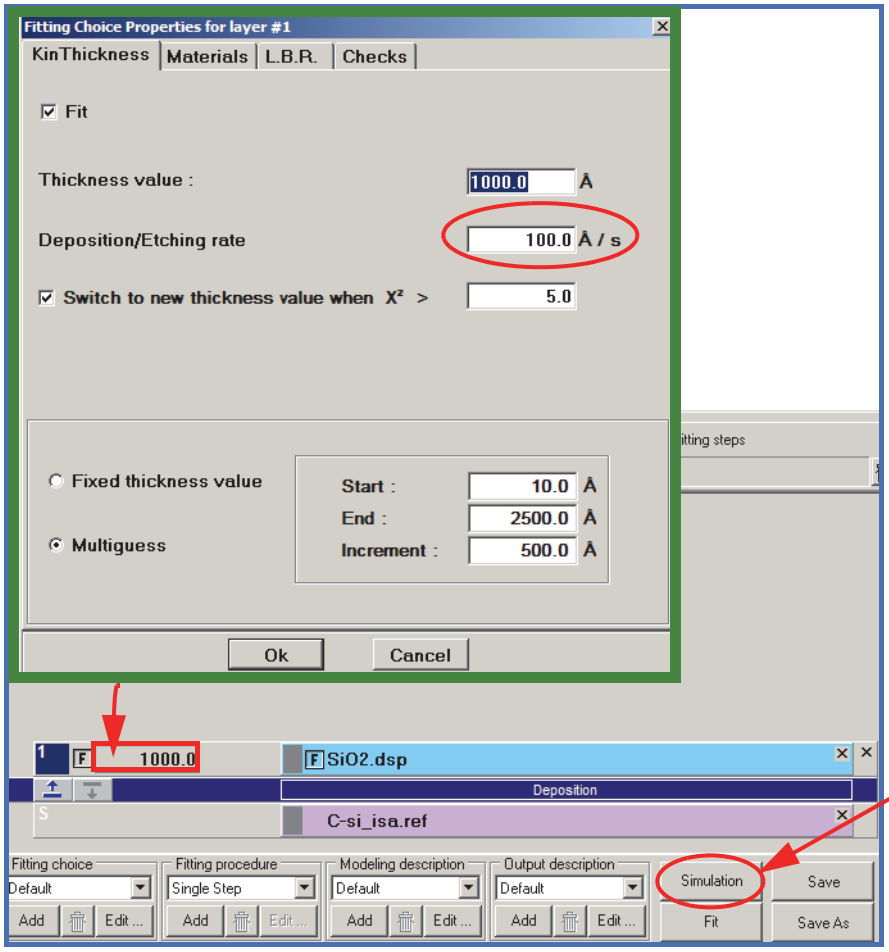
1 **From the Model Kinetic Structure screen, with an Experimental file: click on the Fit button to launch the Fitting process,**




2 **From the Kinetic Recipe by inserting the Kinetic Model with the Acquisition Routine:** see "Kinetic Recipe" on page 303.

## 21.4 Simulation

When no experimental file is available or if the Model structure needs to be prepared before a real acquisition, the Simulation feature can be used. To perform a Kinetic Simulation, enter a ***Deposition Rate*** value as explained on the figure below.




Launching Simulation

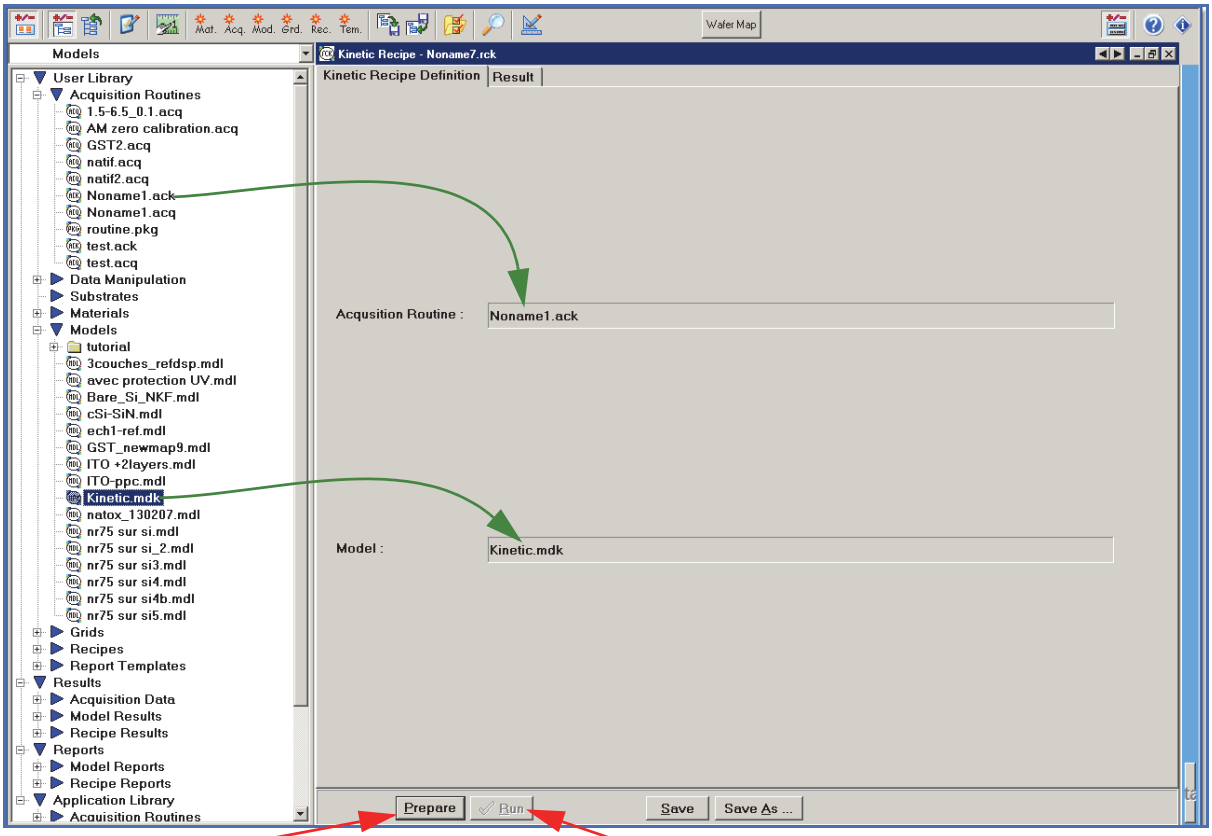
Right-click on the deposition layer thickness value to display the parameters related to the layer. From this window, the deposition rate can be entered.
Validate the entered value, then click on the ***Simulation*** button to launch the complete process. The graphics results are automatically displayed in the "***Graphic Results***" tab.

## 21.5 Kinetic Recipe

The Recipe feature performs the complete acquisition and fitting process on the fly. That means that an acquisition routine will be applied by respecting the strict defined timing and the fitting process will be automatically processed using the multitasking computer mode.

Before launching the Recipe feature, the Acquisition Routine should already be created (see chapter "Acquisition Routine" on page 290) and the Model should be built (see "Model Creation" on page 297)

### *Launching*




Click on the ***Prepare*** button to synchronize software and hardware, and thus optimize them for the kinetic measurement

The ***Run*** button becomes active and launch the full acquisition with the fitting process.

### *Results*

Once the Recipe is launched, and as soon as a measure is ready, a popup window will be displayed with the current results. At the end of the process, the overall results are displayed in the ***Result*** tab.

## 21.6 Fitting Process

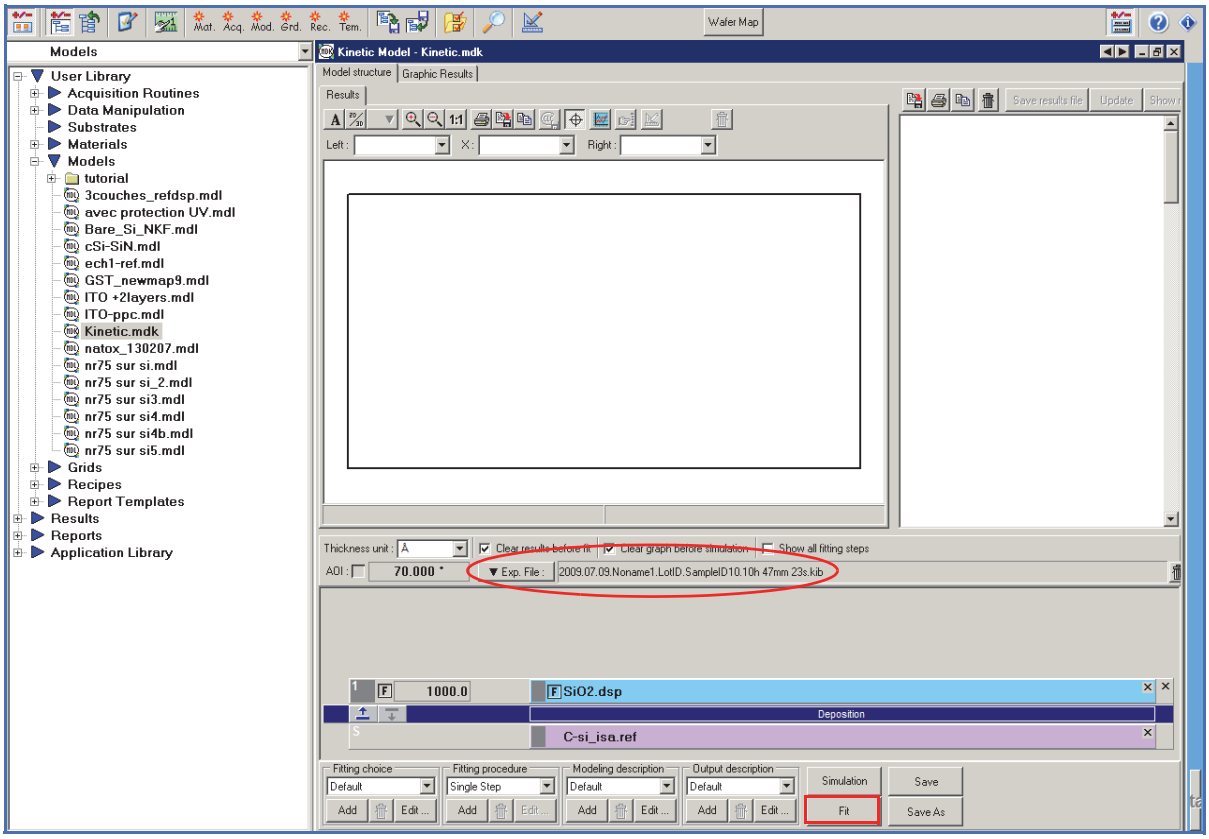
The fitting process is the final step to obtain the results. This final process can be launched automatically if you are using the Recipe feature but can also be launched manually from the main Model window. This chapter explains this last option.

- **Preliminary**

  The Experimental file (acquisition) has been already created,

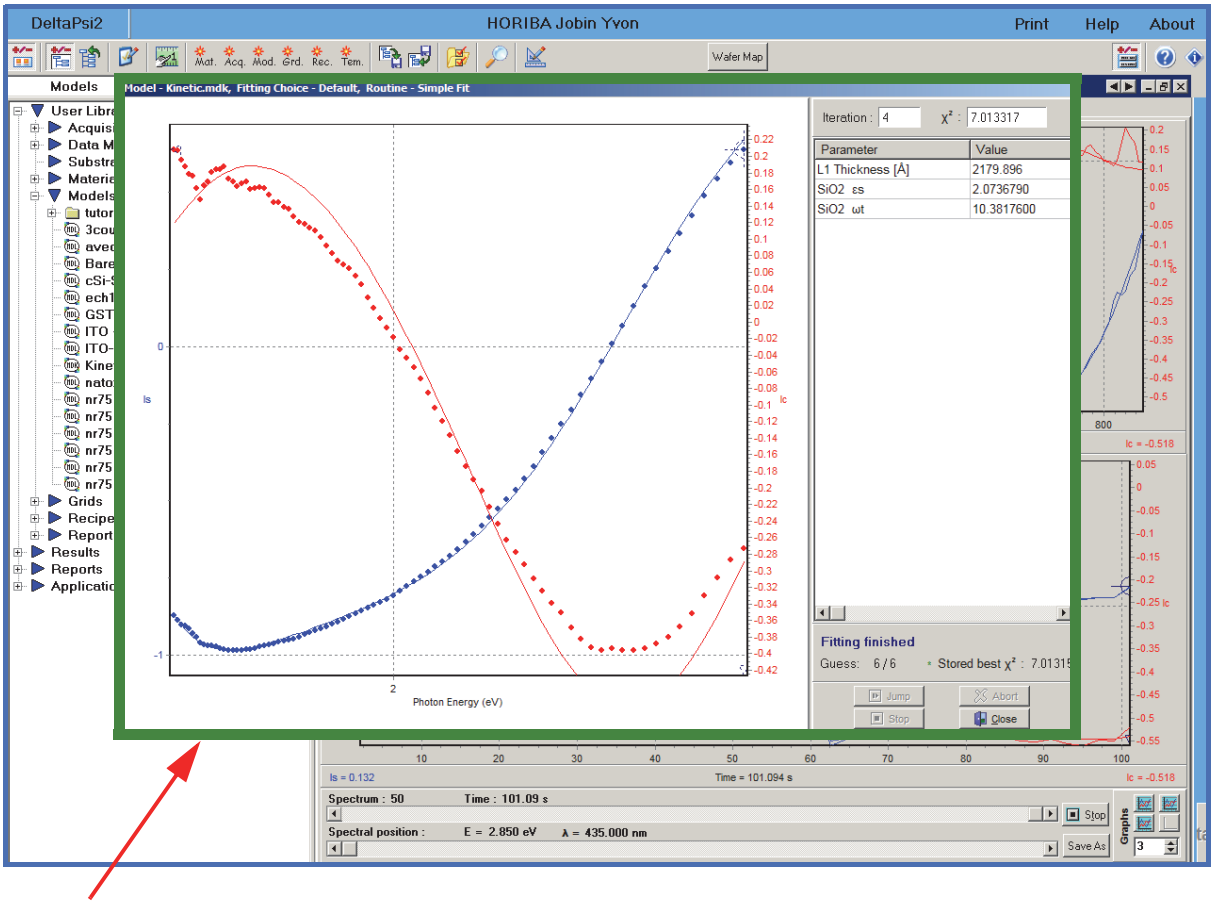
  The Model is created and ready to be used.

- **Run the main Model Windows (see "Model Creation" on page 297). The following window is then displayed:**

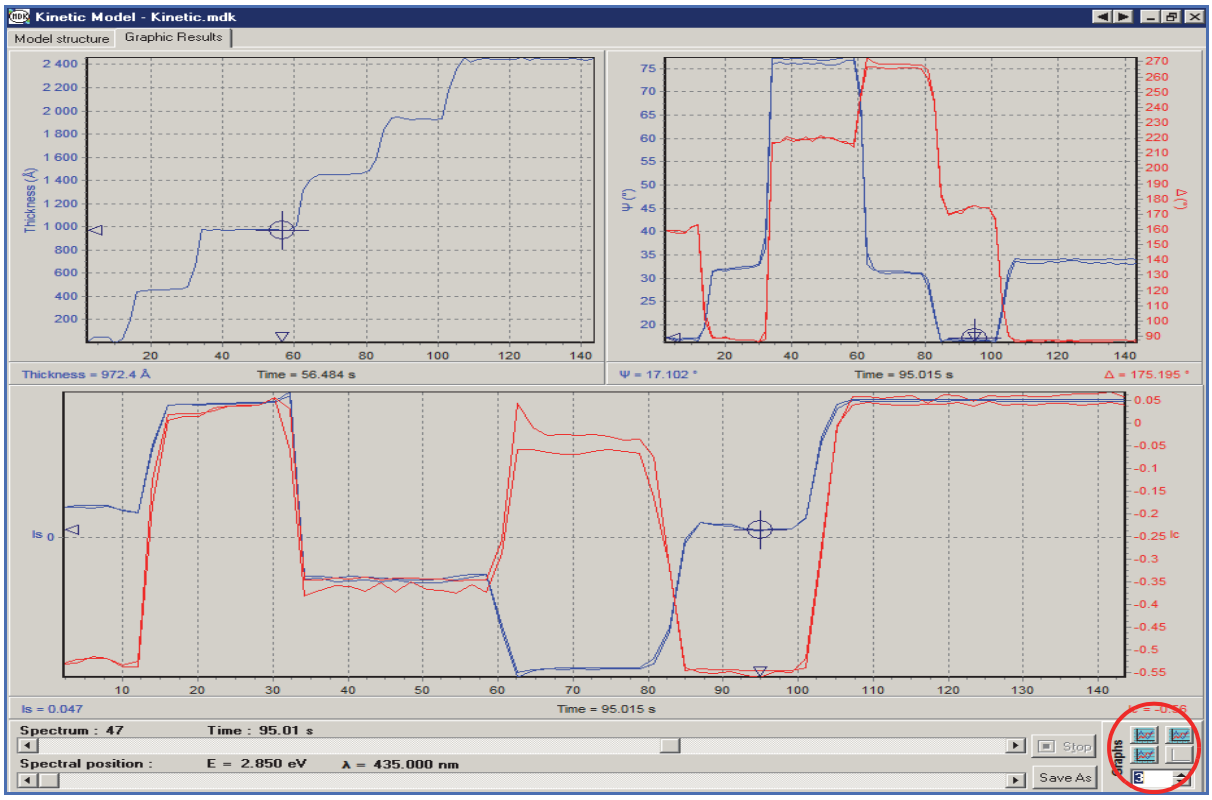



The above window shows a built Model with the Experimental file. The experimental file (red circled on the figure) has been previously created (see "Acquisition Routine" on page 290).

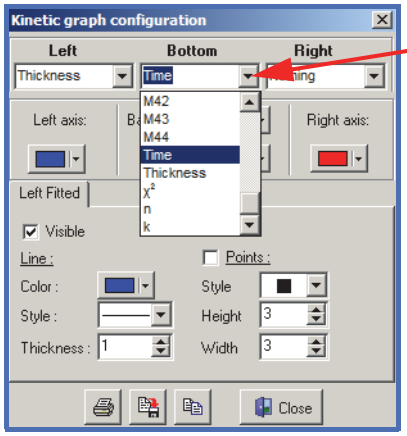
- **Click on the Fit button to run the fitting process. The real time process is then displayed in a pop-up window. Once the fit finds the best $\chi^2$, the full resulted parameters are displayed in the Model structure and the Graphics Results tabs. Figures below shows the displayed windows:**




Once the fitting process is launched a pop-up windows shows the real time process. Once the best $\chi^2$ is found, the final results are displayed. To see the detailed results, close this window. The following Graphic results screen will be displayed: up to 4 windows can be shown and configured at will.




On the bottom right corner, select the number of wished graphics windows, then click on one of the icons  to display the following ***Kinetic graph configuration*** screen:



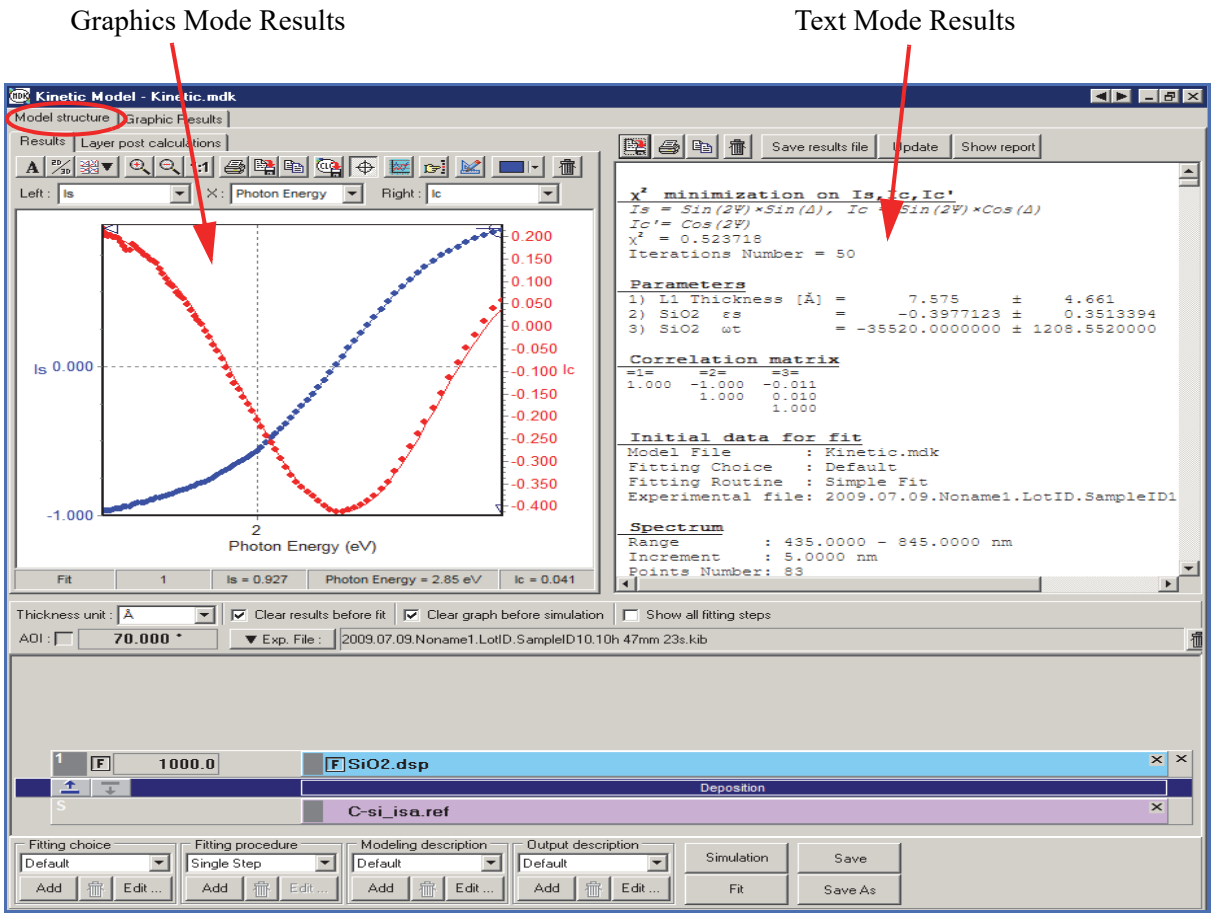

Select the Left-bottom-right axis units

Select the graphics colors and line styles

If some parameters or observables are not available, see **Output Description Setup** in the chapter "Setup" on page 298.

Once the graphs configuration have been made, you can studies them by using the time slider located below the graphs.

- **To visualize the text mode results, click on the Model Structure tab. The following screen will be displayed:**




- **All of the others feature, including the Report Template, are operational and identical to spectroscopic acquisition.**

### 21.6.1 Saving Kinetic results

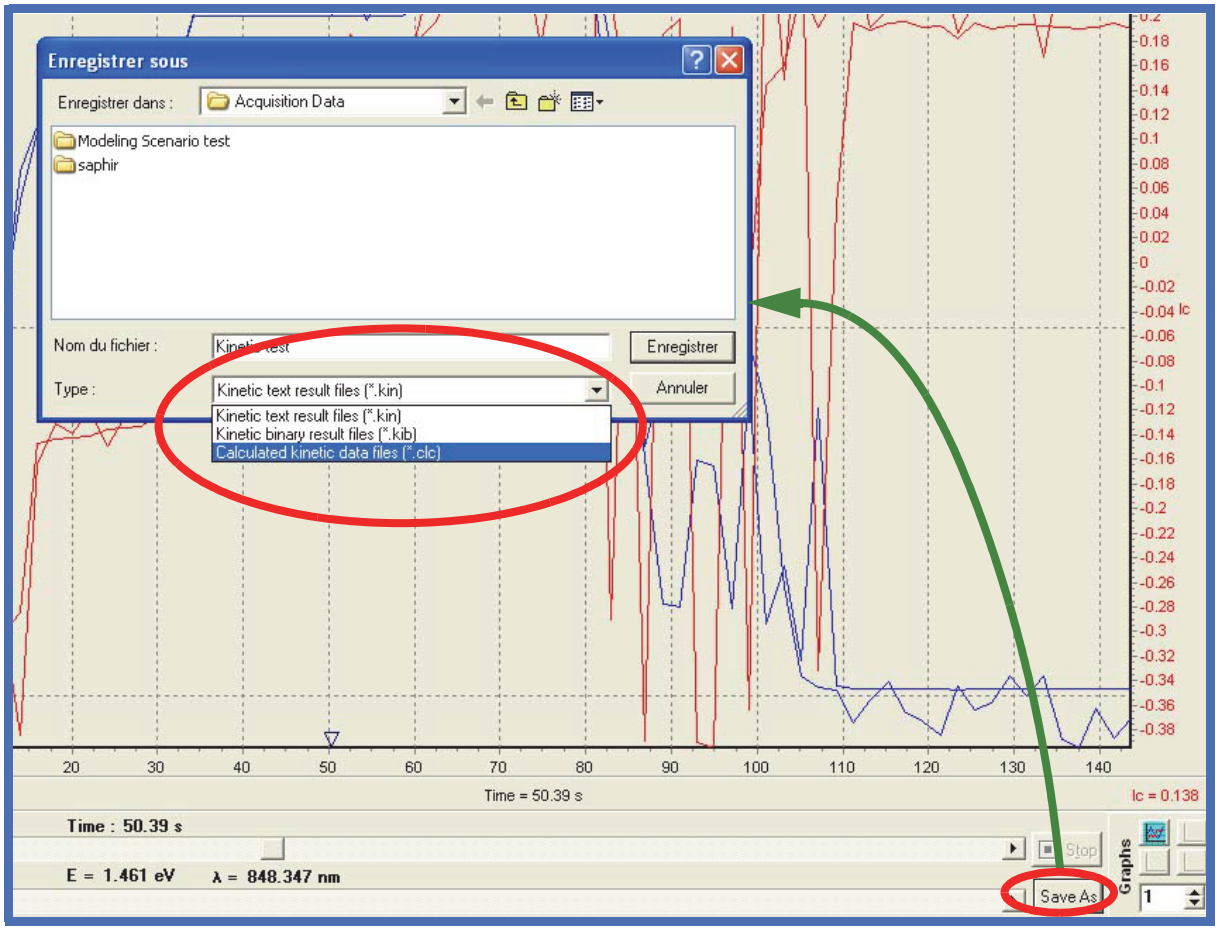
Kinetic results can be saved using 3 different formats:

**.kin** extension: this format arranges all the data according to each time increment. To visualize the data, right click on the file (from the TreeView) and select the ***Open as Text*** choice.

**.kib** extension: it is a binary file which cannot be modified or visualized. This file takes less size on the hard disk with a higher safety protection.

**.clc** extension: This format rearrange data using the CLC (CaLCulation) format. The data are organized in function of the wavelength. In other words, all the data are arranged for each wavelength increment. Additional Manipulations can be performed using the Data Manipulation feature (see "Data Manipulation" on page 117)

This format allows the Excel import for a data processing.


Enregistrer sous
Enregistrer dans :
Acquisition Data
Modeling Scenario test
saphir
Nom du fichier :
Type :
Kinetic text result files (*.kin)
Enregistrer
Annuler
Kinetic text result files (*.kin)
Kinetic binary result files (*.kib)
Calculated kinetic data files (*.clc)
Time = 50.39 s
Ic = 0.138
Time : 50.39 s
E = 1.461 eV
λ = 848.347 nm
Stop
Save As
Graphs

# 22 Checking files

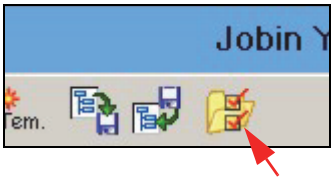
DeltaPsi2 includes a powerful files checking feature. The advanced properties described in this chapter are recommended only for expert users. The user can precisely analyze the files properties and will able to answer the following types of questions:

- **Which file is linked to a recipe, a model etc....**
- **Which file is not linked,**
- **Which file is corrupted,**
- **Which file can be deleted...**

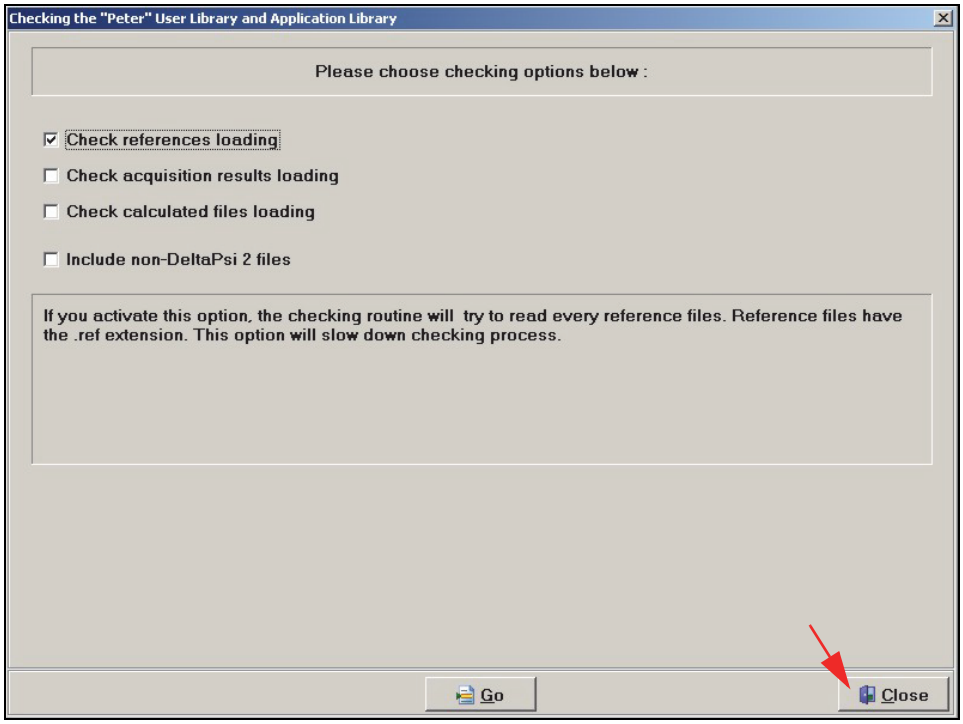
**NOTICE**: The checking procedure will only check the current user profile.

## 22.1 Accessing

Click the ***Check files integrity*** icon  located on the ***Button Bar.***




The following screen will be displayed:

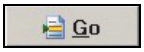



05/20/2016

The ***Checking*** screen shows additional selective options for analyzing and checking files. Each choice can be added to another for a complete (but time consuming!) files control.

The table below shows which format and how each format will be checked according to the selected checking options:

| Files formats | Checking Status |
|---|---|
| .REF | Listed or fully checked if the additional option «***Check references loading***» has been selected. |
| .SPE<br>.KIN | Listed or fully checked if the additional option «***Check acquisition results loading***» has been selected. |
| .CLC | Listed or fully checked if the additional option «***Check calculated files loading***» has been selected. |
| .PKG, .RCR<br>.MDR, .MMR<br>.SCR, .RAF | Listed but never fully checked. |
| .MDT, .RCT<br>.MMT, .SCT<br>.DSP, .ALY<br>.ACQ, .PPC<br>.SPL, .MDL<br>.SCE, .GRD<br>.RCI, .RCM<br>.UDF, .UDA<br>.BMM, .SMM<br>.DMN, .RSN<br>.ACT | Listed and always fully checked |
| .HTML, .DOC<br>.JPG, .WMF<br>.EMF, .TXT<br>.CSV, .RTF<br>and other extensions | Listed if the additional option «***Include non-DeltaPsi2 files***» has been selected, but these files cannot be fully checked |

Once the options have been chosen, click on the 

button to launch the process.

Figure 22-1 shows an example of a resulted checked screen. Many different properties are available from this screen and are detailed below.

**«Processed files» tab**

On the top, «radio» buttons allow the user to select the sorting list type:

- TreeView file names,

- group by file extensions,
- most corrupted files listed first and valid files sorting by TreeView order.

The listed files are organized in three columns:

- A check in selection which validates the Delete and close button: the selected file can be then deleted. Take care.
- A file name list with location.
- The status for each file:

  ***loaded*** means that the file has been fully checked and loaded properly,

  ***loaded, X error(s)*** means that the file has been fully checked and loaded properly but X linked files are missing or corrupted,

  ***reference by N file(s)*** means that the current selected file is linked and used by N other files. Click on a file to highlight it, the complete information will be then displayed on the bottom screen area. The linked or referenced files are also listed.

  It becomes easier to manage, control, test and verify the integrity of all of the files.

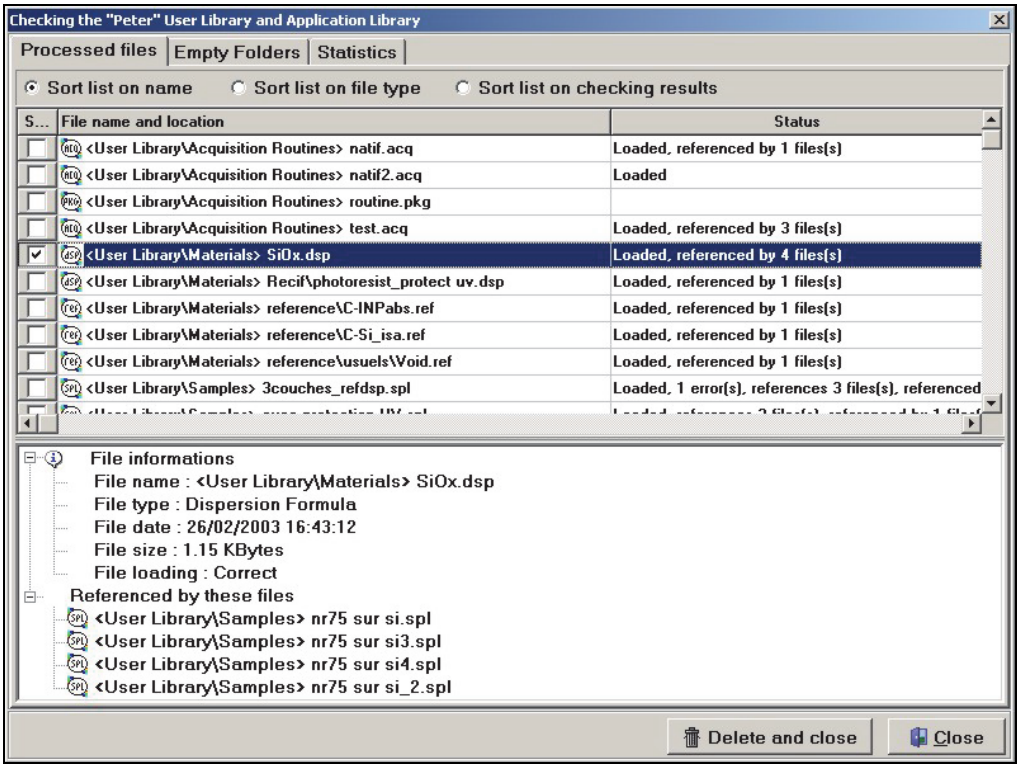



**Figure 22-1 Checked screen**

**«Empty Folders» tab**

Click on the Empty Folder tab to list the empty folder(s) founded during the checking process. This feature is useful to clean a DeltaPsi2 profile.

Figure 22-2 shows an example of an Empty Folders screen:

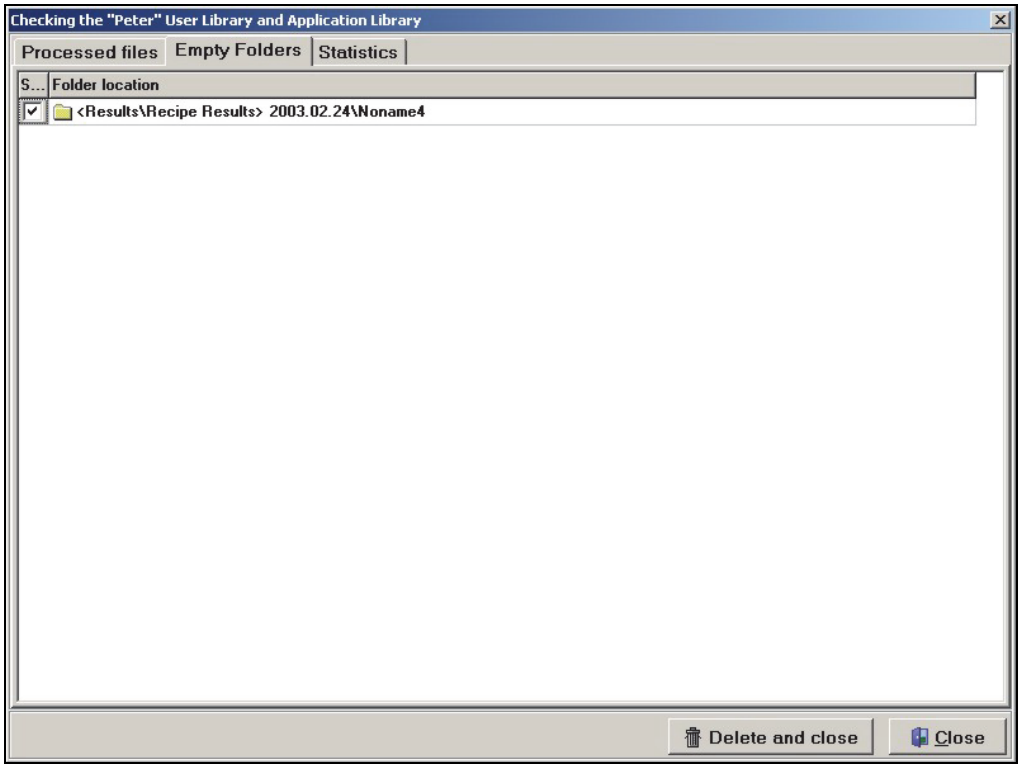



**Figure 22-2 Checking Empty Folders**

### «Statistics» tab

The ***Statistics*** tab summarizes the general checking results. Figure 22-3 shows an example of the displayed information.

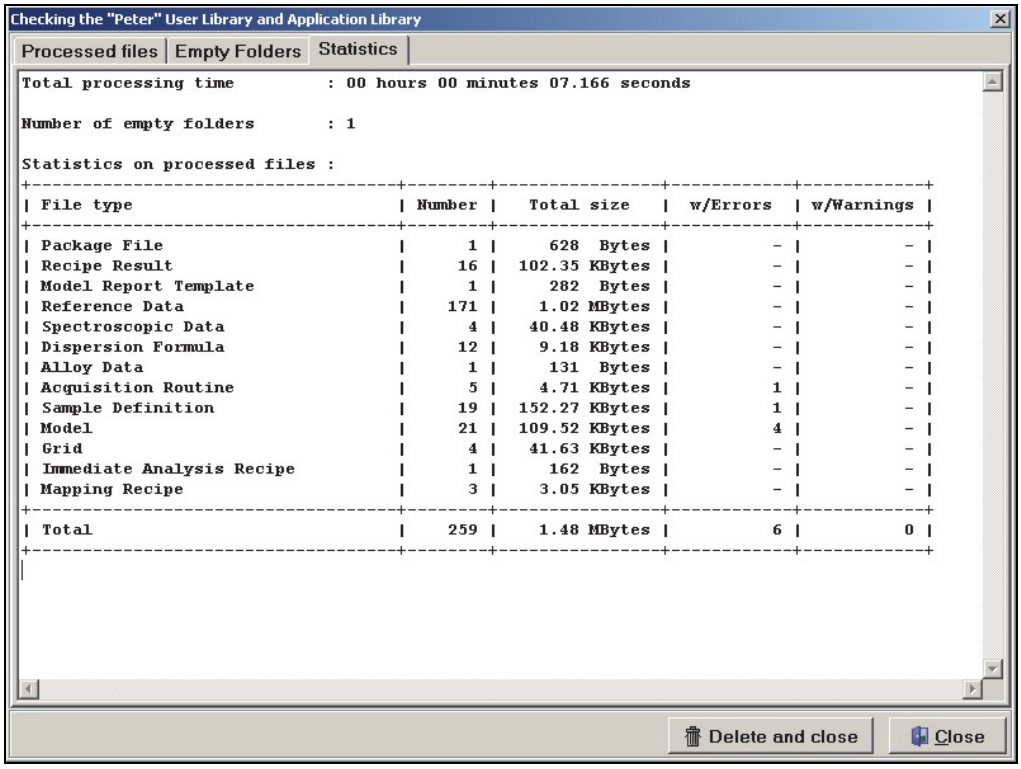



**Figure 22-3 Statistics**

# 23 Cooling/Heating Stage

## 23.1 Definition

Optionally, your Ellipsometer can be equipped with a Cooling/Heating Linkam T9X Stage.

DeltaPsi2 controls the Cooling/Heating Stage by using a **Temperature Control Routine** which can be fully arranged by the user. The Temperature start and final values, the increment as well as the ramp rate with the stabilization delay can be fully setup.

Before using the stage, a driver installation must be performed.

**Preliminary requirements**:

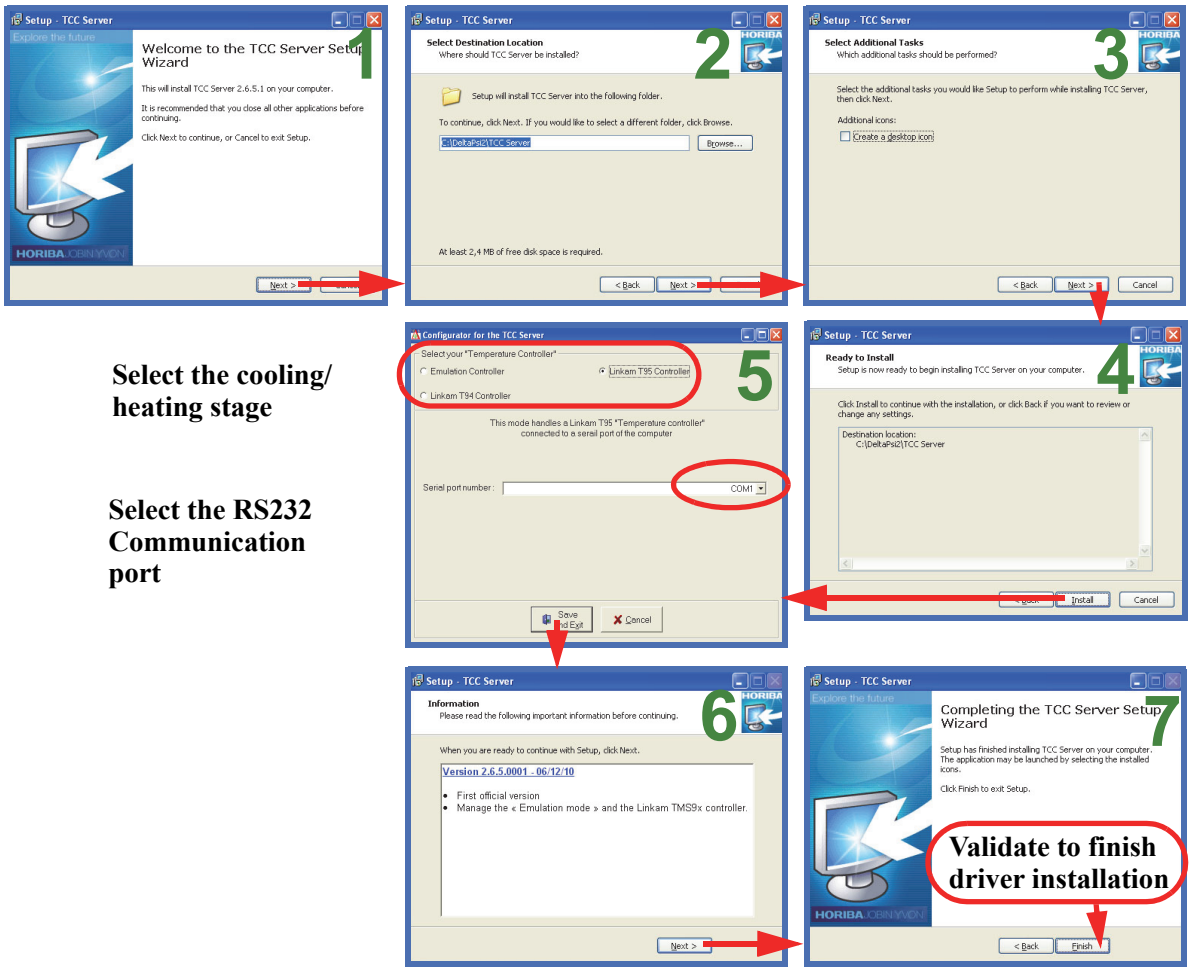
- **Your System must be compatible with this option,**
- **DeltaPsi2 Software must be already installed,**
- **RS232 communication port must be available on the PC Workstation and the Cooling/Heating Stage properly connected. Read the Hardware installation described in the Stage manual.**

## 23.2 Driver Installation

Two steps are required for installation. The driver is delivered on a CD with the cooling/heating stage.

**1st Step**: Locate the Setup ***TCC Server installation*** file 

on the CD, then double-click on it. Follow the procedure explained below:




**Select the cooling/ heating stage**

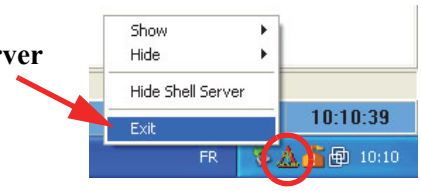
**Select the RS232 Communication port**

**2nd Step**:The new TCC Temperature Server must be added to the ShellServer. Follow the procedure described below:

1 Close DeltaPsi2 Software if it is running,

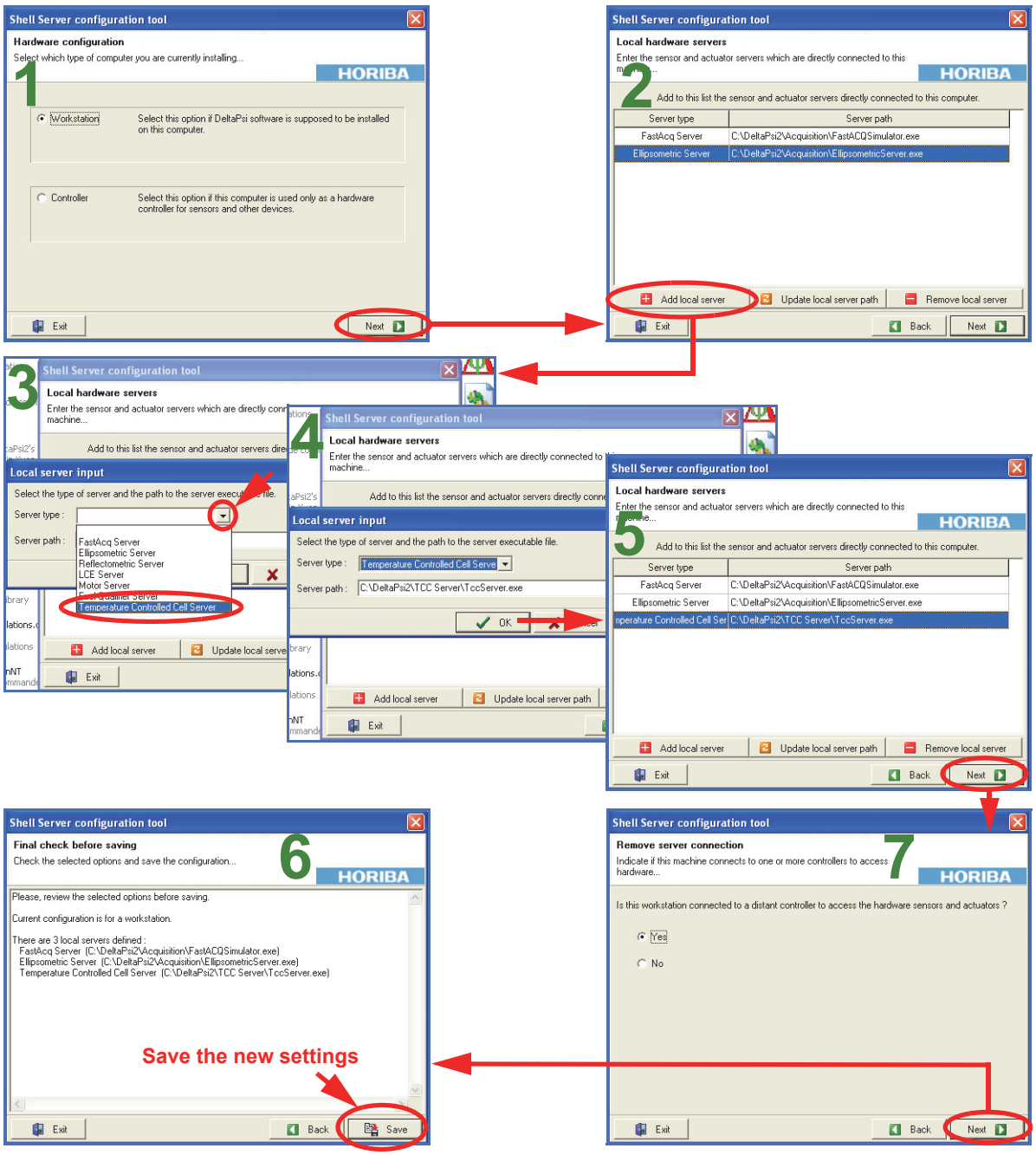
2 From the taskbar located on the bottom right hand side of the desktop window, right-click on the ***Shell Server*** icon, then select ***Exit***,

**Exiting Shell Server**




05/20/2016

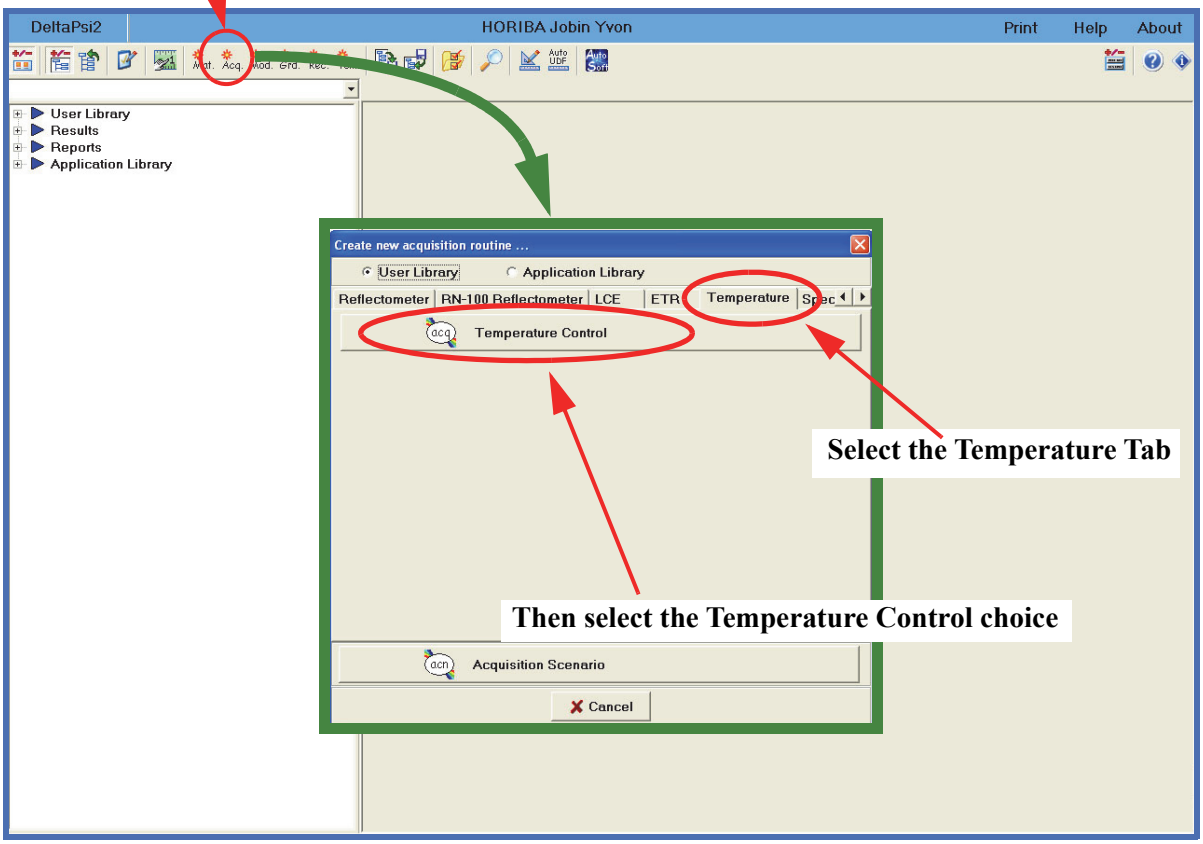
**3** From Windows Explorer, double-click on the **ShellServerConfig.exe** program located in the **C:\DeltaPsi2 folder**. Follow the procedure described below to add the TCC Server:



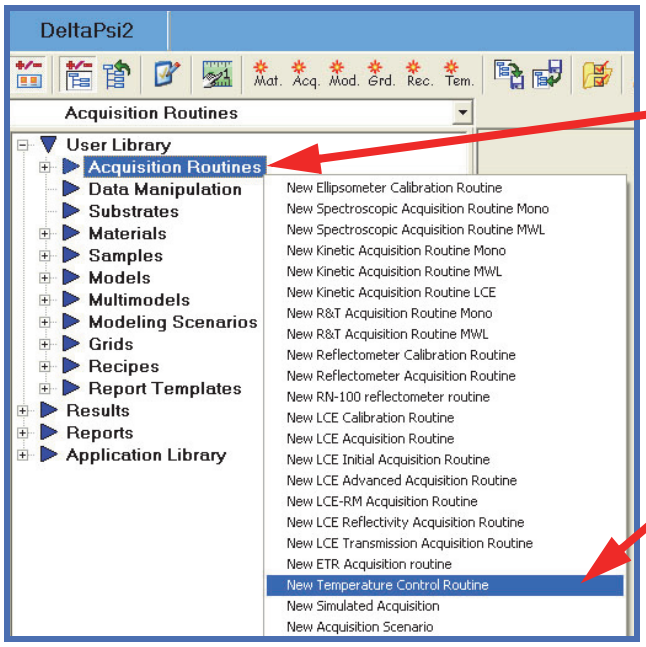

## 23.3 Accessing

There are two ways to access to the ***Temperature Control Routine*** window:

**Click on the *Acq* icon located on the Main icons bar**




**OR**



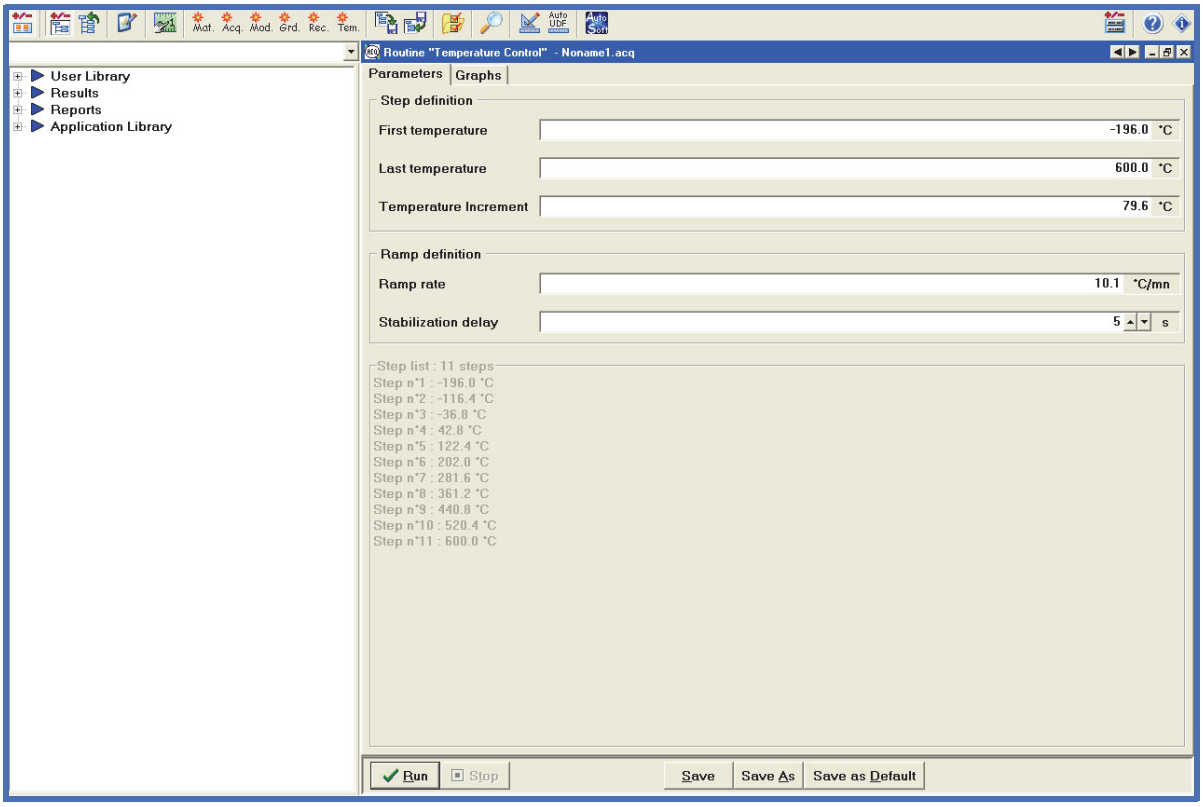

**Right-Click on the *Acquisition Routine* folder**

**Then, select the *New Temperature Control Routine* choice**

*05/20/2016*

## 23.4 Creating a Temperature Routine

The Temperature Control starting window will be displayed. The displayed default values shows the lower/upper temperature limit of the delivered Stage. The other parameters can be set according the measurement procedure.

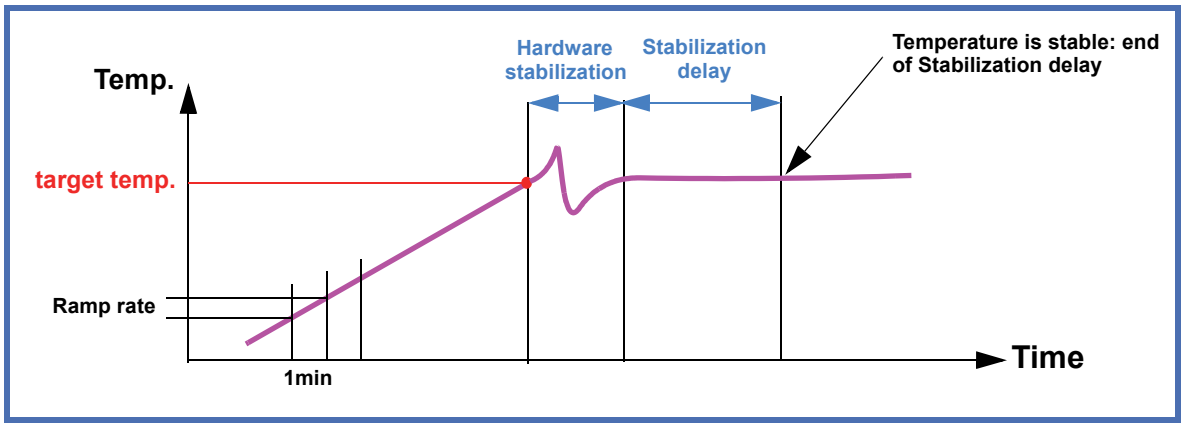



### 23.4.1 How to set the Routine

1. Enter the **temperature limits** of the experiment: first temperature is the lower, last temperature is the upper.
2. Enter the **Temperature Increment**: according to the entered value, the routine steps are automatically calculated below.
3. Enter the **Ramp rate**: this value limits the temperature growth calculated by minute. This parameter is necessary for some samples which cannot withstand a fast temperature variation. Please notice that 20°C/min. is the maximum Ramp rate (according to the hardware capabilities).
4. Enter the **Stabilization delay**: a high performance temperature stabilization circuitry has been integrated to the system, however the user can apply an additional scrutinizing. The temperature control scenario is organized in two steps:
   - As soon as the temperature is reached, the hardware regulation circuitry is activated and the target temperature tries to be stuck within 0.1°C.
   - DP2 keeps watch on it and as soon as the target temperature is reached within ±0.1°C, DP2 starts the Stabilization delay. During this period, if the temperature moves outside the ±0.1°C range around the target temperature, the Stabilization delay stops and will restart as soon as the target temperature will be

inside the ±0.1°C range. If, during the Stabilization delay, the target temperature remains stable inside the ±0.1°C, the Stabilization delay stops and the routine temperature step ends.

**5** The diagram below shows the temperature progression to a defined target temperature with the Stabilization delay.




### 23.4.2 How to use the Routine

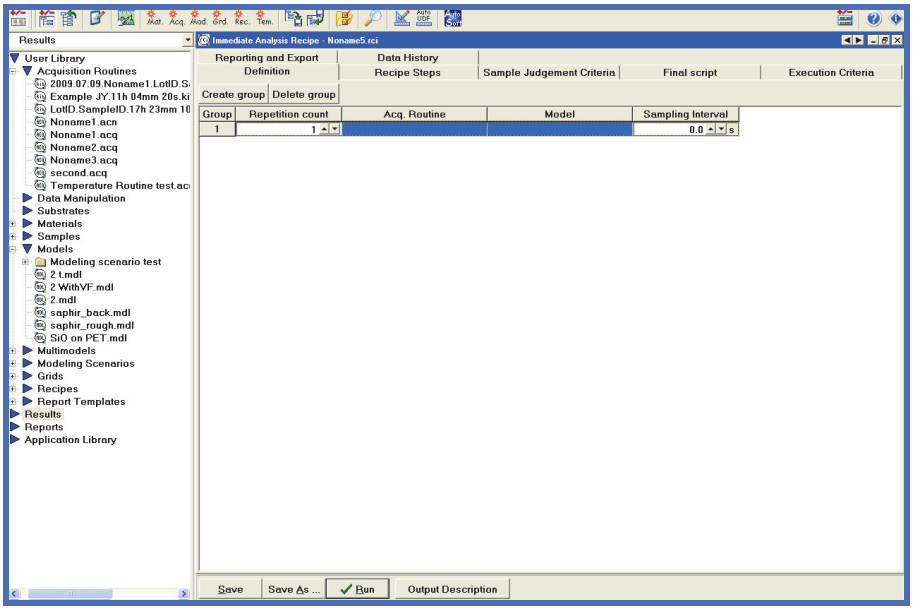
As shown above, the Temperature Routine includes a step for each programmed temperature. These steps will be used in the Recipes.

The Temperature Routine is programmed as a continuous loop. Each **Run** action moves to the next step.

## 23.5 Creating an Immediate Recipe with Temperature Control

In the previous chapter, we have seen how create a Temperature Routine. This routine is organized by steps. We will create now an Immediate Recipe which performs an acquisition at each temperature step. Follow the steps described below:

1 Create a new **Immediate Analysis Recipe**: see **"Immediate Analysis Recipe" on page 254.** The following screen will be displayed.




2 The aim is to perform an acquisition at each temperature step, which means that the Temperature Routine must be alternated with the Acquisition Routine.

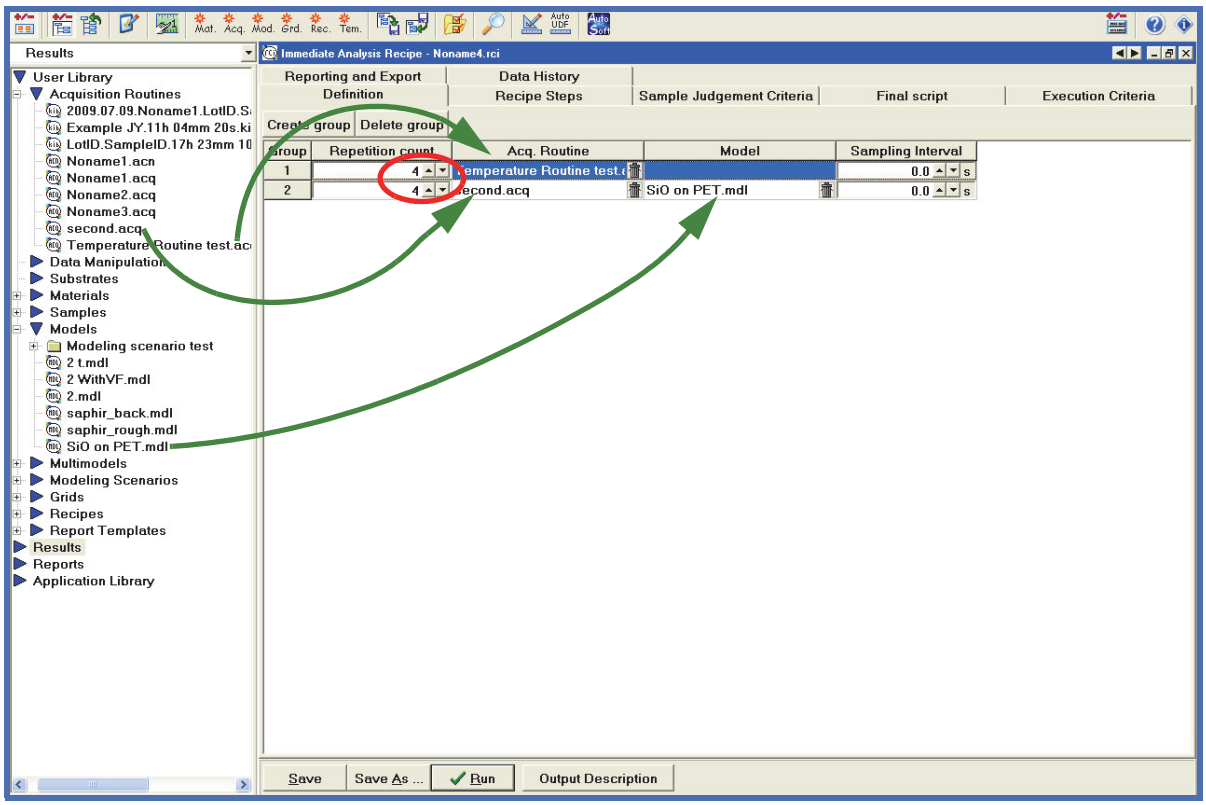
3 Click on the ***Create group*** button to add a new group.

4 From the TreeView, drag and drop the previously created **Temperature Routine** to the ***Acq. Routine*** column of the 1st group.

**NOTICE**: The **Temperature Routine** must always preceded the **Acquisition Routine** concerned by the temperature steps.

5 From the TreeView, drag and drop the **Acquisition Routine** and the **Model**.

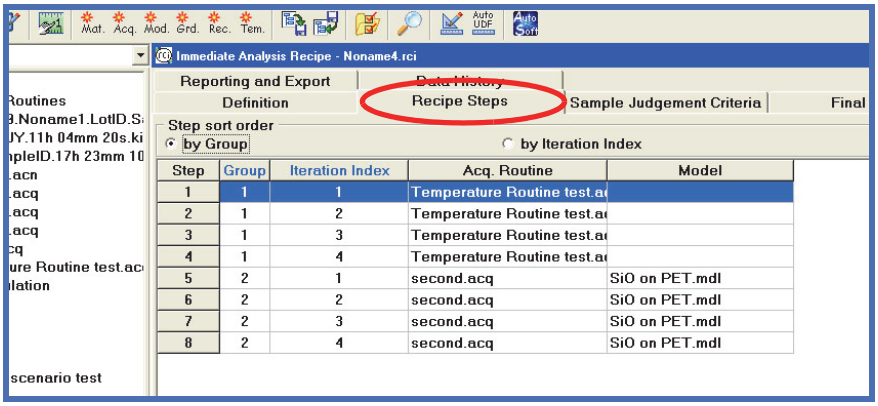
6 In our example, we have 4 temperature steps. If we want to really analyze the sample at each temperature step, we must change the ***Repetition count*** parameter to 4 for the both groups. The figure below shows




**TIP**: To avoid the continuous heating of the sample when the last step is reached, a good idea is to add a temperature routine step at the end of the analysis Recipe. Thus, after the last acquisition Routine, the last Temperature routine will be launched and will load the next Temperature step which will be in fact the first Temperature step (Temperature Routine steps are programmed to be running in continuous loop). The sample heating will automatically decrease to the temperature set in the first step.

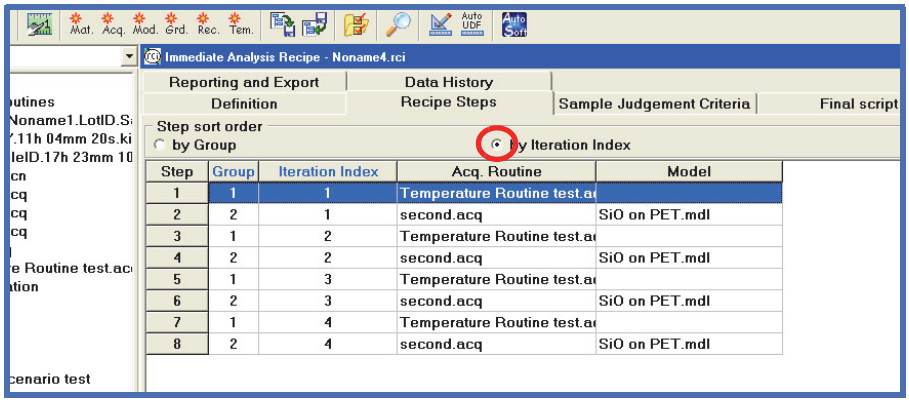
In practice, add one more Repetition Count for the **Temperature Routine** comparing to **Acquisition Routine**. In our example, we have not added a Temperature routine as a last step.

**7** Now we must arrange our analysis to alternate the routines. Click on the Recipe Steps tab located above the ***Acq. Routine*** column. We obtain the following window:




05/20/2016

As shown on the figure above, by default, the steps are listed by Group and the steps are not temperature/analysis alternated. Click on the «***by Iteration Index***» to reach the wished result. The following window is displayed:




We have an **Immediate Analysis Recipe** which contains alternated temperature/analysis steps.

**8** Click on the ***Save As*** button to save the Recipe.

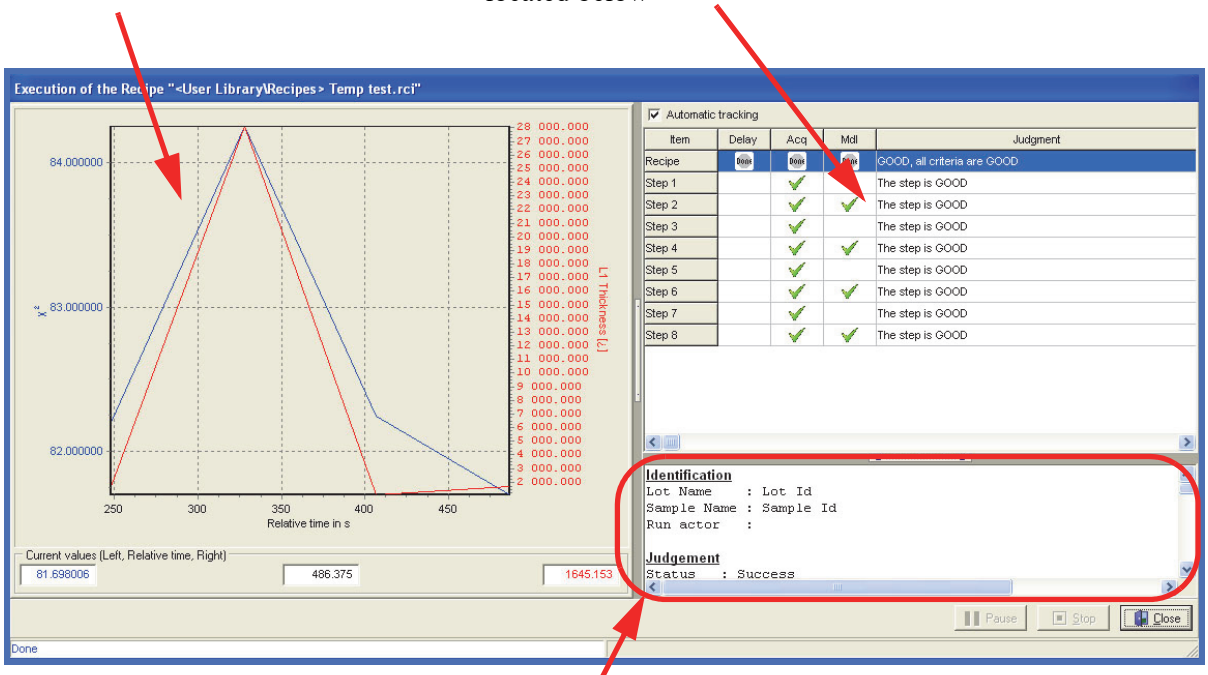
**9** If necessary, click on the Output Description button to adapt the output parameters to your needs.

**10** Click on the ***Run*** button launch the Immediate Analysis Recipe. A first window asks for sample information. Enter the informations, then click on the ***OK*** button to Run the Recipe.

**11** At the end of the Recipe, the following window will be displayed:

Graphic presentation results

Select each step to display the detailed results on the frame located below



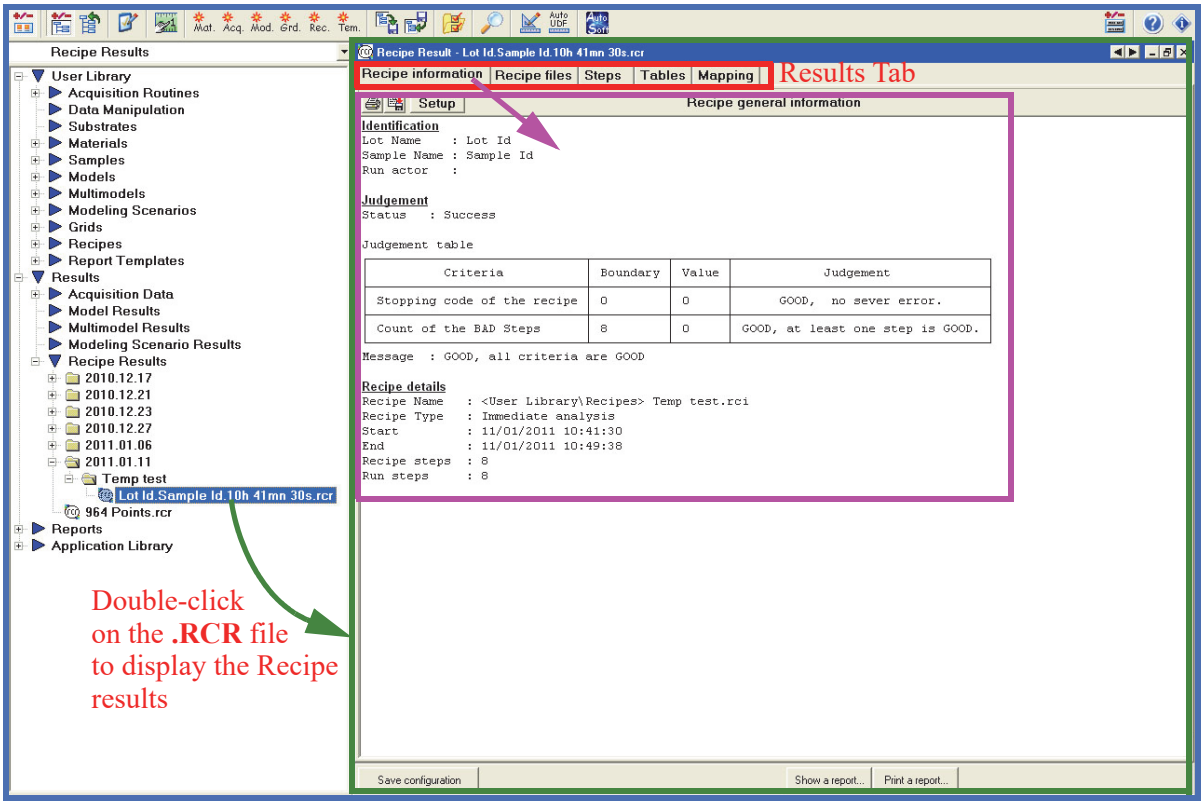

Results of the above complete Recipe

Click on the ***Close*** button to close the window.

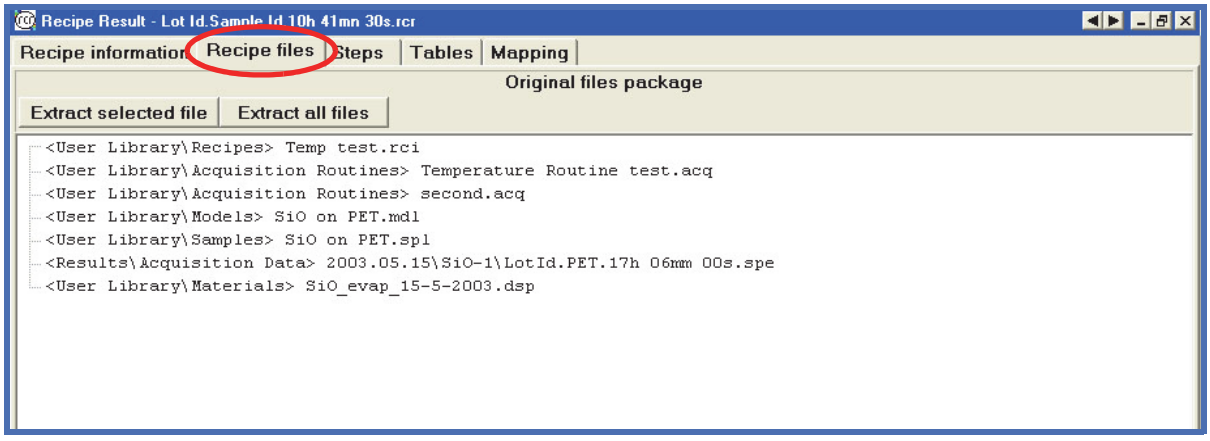
05/20/2016

**12** The **Results Recipe** file has been created and the parameters can be consulted. Double-click on the .RCR file to display these results. Several parameters windows are available. The Result tab, located on the top allows the user to navigate along the results.

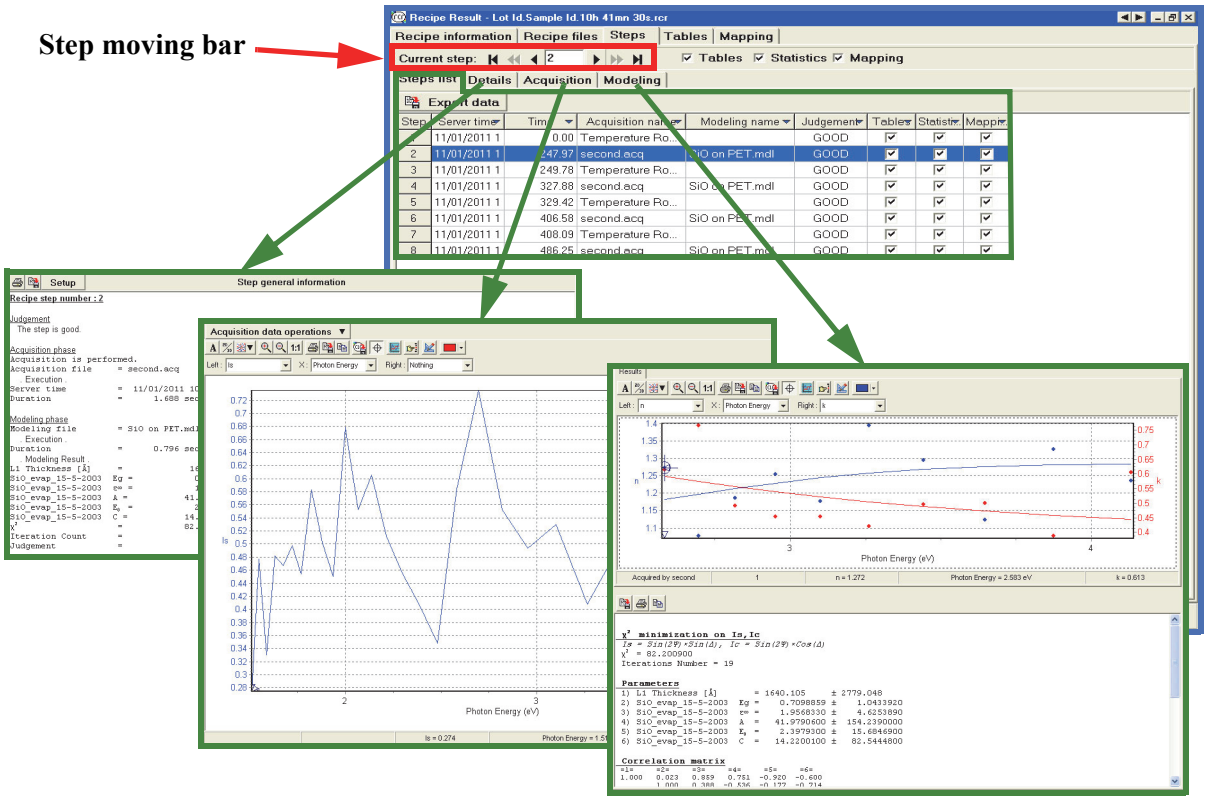
**Recipe information**: the **Recipe information** window groups the general information about the Recipe analysis conditions.



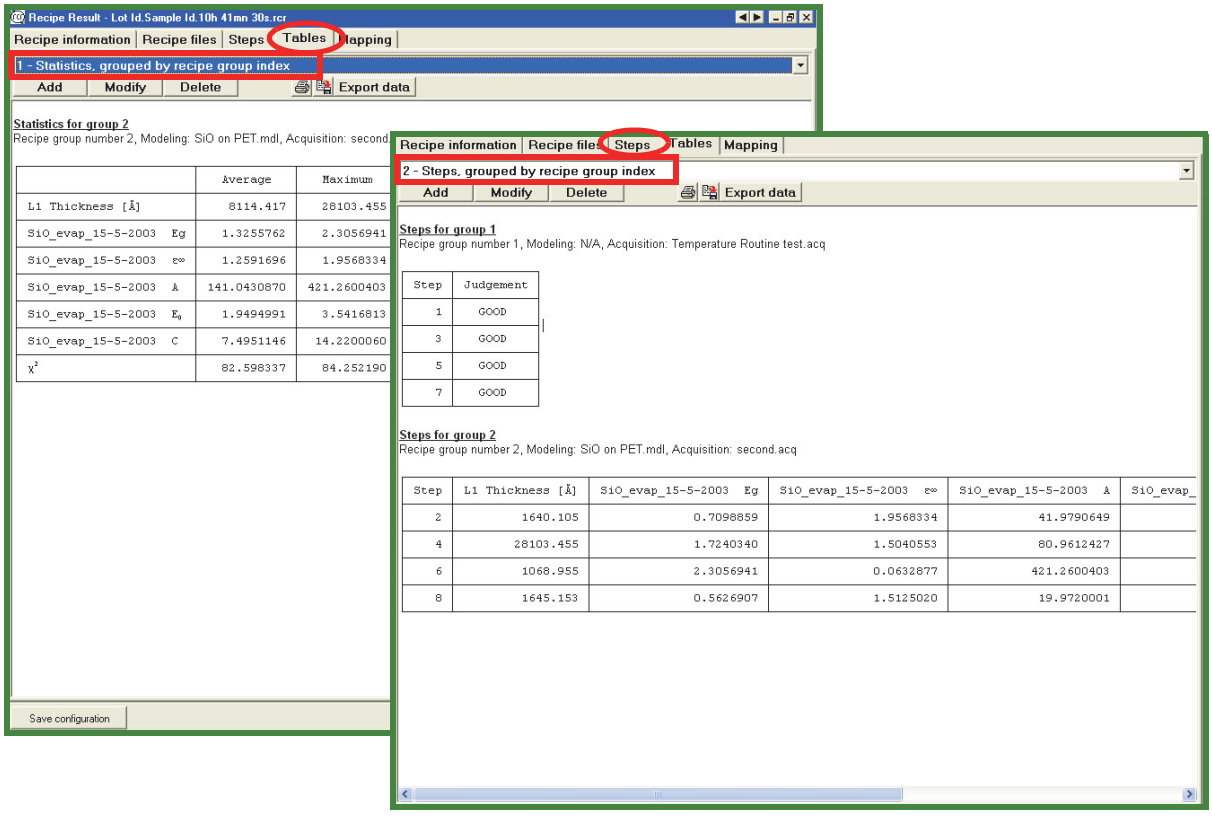

**Recipe files**: the Recipe files window lists all the files used by the Recipe.



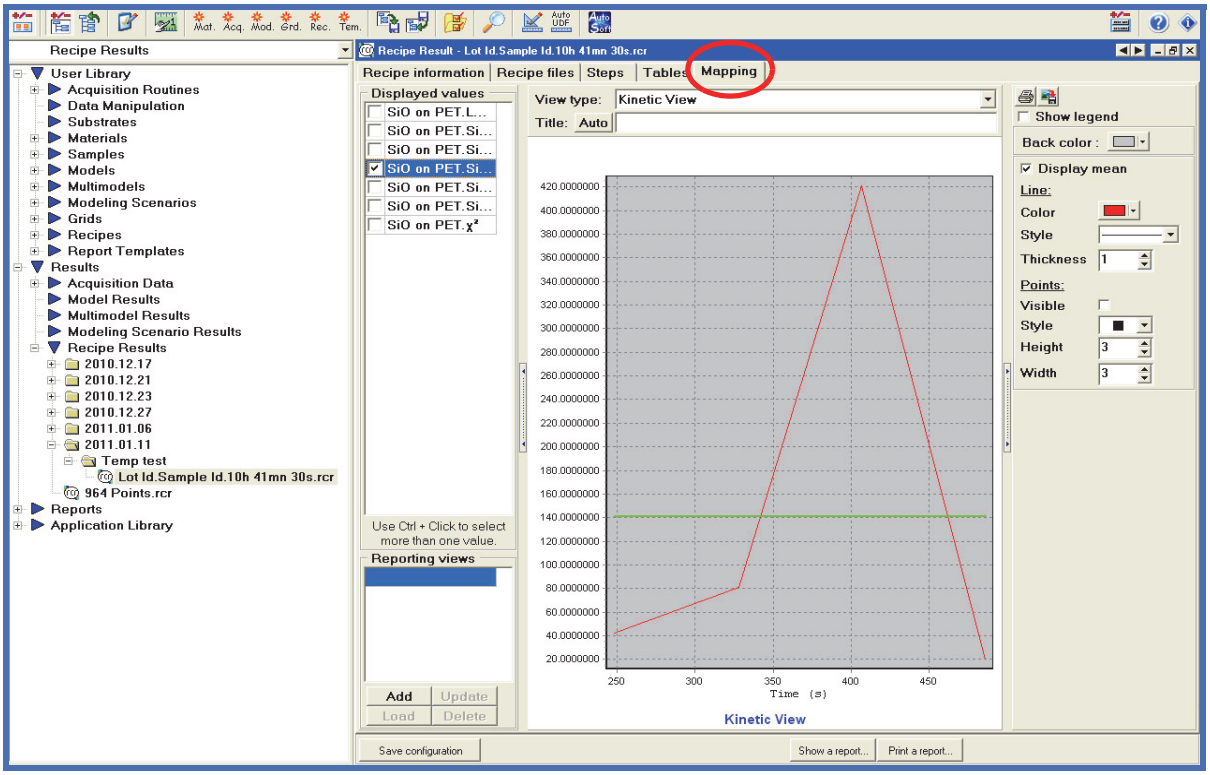
**Steps**: the ***Steps*** tab shows the complete information by steps from the steps sequencing to the sample ***Acquisition*** and ***Modeling***.




**Tables**: the **Tables tab** displays the results by step and by parameter and also statistics according to the previous entered criteria.



**Mapping**: the **Mapping** tab shows a 3D presentation of the Recipe. Of course, a real mapping with many acquisition points will render the overall view more usable.




05/20/2016

# 24 Troubleshooting

In order to best respond to any bugs you may encounter with DeltaPsi2 Software, please provide the following information:

Company Name:
Name:
E-mail address:
DeltaPsi2 version number:
A brief description of your system:
A description of the problem. Please include any actions that produced the error, and if the error is reproducible.
Please include any screen shots or error warning text if possible.

Send your bug reports to Services.jyfr@horiba.com. If possible, please attach screen-shots zipped in .jpg or .gif format.

# Index

**Numeral**
2D/3D switch 33
3D layout 265

**A**
Accumulation 55, 60
Accumulation Number (MM-16) 82
Acquisition 42
Acquisition (kinetic) 290
Acquisition (UVISEL Ellipsometer) 43
Acquisition Conditions (kinetic) 294
acquisition data format file 28
Acquisition MM-16 80, 84
Acquisition parameters 46
acquisition routine format file 28
Acquisition Scenario 132
Acquisition setup (standard) for MM-16 80
Acquisition with UVISEL2 70
Advanced Acquisition (MM-16) 83
AF 260
angle 55
Angle of incidence 45
Angle of incidence (check) 203
Angle of incidence fitting 183
AOI 45
AOI fitting 183
Application Library 14
Application library 29
AutoFocus 260
Autofocus Adjustment (UVISEL VIP System) 144
Automatic Update 185
Axis attributes 36
Azimuth 186

**B**
Background 48
background (MM-16) 82
Binary format 293
Bound Multimodel format file 28
Bruggeman effective medium 200

**C**
calculated file 34
Calibration MM-16 78
Cartesian Array Grid 236
Check 202
check criteria 202
Check Files 24
Checking files 310
Circular Grid 230
clc extension 308
Clear Results before fit 204
clipboard (copy to) 34
Coherency Length 191
Color 36
Color Grid Setup 227, 245
Color selection 39
Complex Grid 244
Conversion to Kinetic 297
Conversion to spectroscopic 297
Cooling/Heating Stage 315
Copy to clipboard 34
Correlated 197, 201
Correlated (layers fitting) 184, 197
Correlated layer 201
Cross Grid 235
Cursor 35
Curve Attributes 37

**D**
D AOI Settings 180, 194
Data files 12
Data Manipulation 39, 117
Data Manipulation Scenario 132
deletion 21
Deposition bar 298
Deposition Rate 303
Die 272
die (adding on grid) 248
Dispersion Parameters 203
Display Mode 40
Displayed data selection 38

**E**
Edit Feature 120
Eg 209
Ellipsometric (Measurement Mode) 174, 188
Ellipsometric Data 50
Ellipsometry configuration 174, 189
EMA 200
epsilon i 50
epsilon r 50
Error values 19
eServer 25
etching 300
Evaluated Values (MM-16) 82
Export files 24
Export/Import Package 14

**F**
File Explorer 16
file formats 15
Files formats 311
Fit Volume fraction 199
Fitting parameters 195
Fitting Process (kinetic) 304
Fitting process (launching) 203
Fitting properties setup (general) 182
Fitting setup 182
Flow Report 283
folder (create new) 13
Folder creation 13
Font Size 37
formats (file) 15
Formatted Report 281
Full Scale/Customized 37
Full Screen 34

**G**
Ge 209
General Attributes 35
General Display Settings 35
General Setup 21
Generating report 285
Graph Setup 56, 63
Graphics export 34

05/20/2016/

Graphics Interface 23
Graphics Manipulation Screen 33
Grating properties 185
Grid Defintion (Complex) 248
Grid format file 28
GridEditor 225
Group (recipe) 271

**H**
h and Tol 196
Hardware Options 44
Help online 26
High Accuracy merge 53
High Voltage 47, 54
High Voltage (kinetic) 296
High Voltage (MWL) 60
High Voltage (S0 range setup) 45
HTML Report 284

**I**
Icon Palette 32
Immediate Analysis Recipe 254
Immediate Recipe with Temperature Control 320
Import files 24
Import/Export feature 29
Incidence angle 55, 294
Index number 212
Initial Acquisition (MM-16) 82
Integration Time 47
Integration time 55
Integration time (MM-16) 82
Is and Ic 46

**J**
Joystick (stage) 49

**K**
k 50
keys (processing modules) 20
kib extension 308
kin extension 308
Kinetic measurements 289
Kinetic model format file 28
Kinetic Parameters 293
Kinetic Recipe 303

**L**
Label 37
Label decimal precision 37
layer (build) 169
layer (manipulation) 170
Layer repetition 173
LBR 201
Left slope projection 186
Library (application) 14
Library (data files) 12
License keys 20
Limited Backside Reflection 201
Line Grid 233
Linkam 315
Lot (recipe) 273

**M**
Manual Grid 238
Mapping (representation) 279, 326
Mapping Recipe (complex grid) 267
Mapping Recipe (simple grid) 255
Marquart 195
material format file 28
Materials Choice (fitting) 198
Maxwell-Garnett 200
Measurement 22
Measurement Mode 174, 188
Measurement time (MM-16) 82
Merge (Measurement Mode) 174, 188
MG 200
Micro-Spot 47
MicroSpot 53, 294
MM-16 Acquisition 80
MM-16 Advanced Setup 81
MM-16 calibration 78
MM-16 Views 75
Model Building 166
Model Creation (kinetic) 297
Model format file 28
Modeling 157
Modeling Conditions 174, 188
Modeling Output precision setup 18
Modeling Results (simple grid) 263
Modeling results file format 28
Modeling Scenario 219
Modeling Scenario format 28
Modeling Scenarios format file 28
Models update 32
Monochromator/Multiwavelength switch 44
Motor 51
Motor increment value 51
Motors 51
Motors (control panel) 50
MS Word Report 284
Multiguess 197, 200
Multimodel files format 28
Multispectra files 174, 189
Multistart 197, 201
Multiwavelength Acquisition 58
Multiwavelength/monochromator switch 44
MWL acquisition 45
MWL Acquisition Setup 58

**N**
n 50
Navigation Display Mode 41
New Curves Attributes 37
Noise correction 120

**O**
Observable (add an) 206
Observable Setup 17
Operations (Data Manipulation) 120
Options (software options) 13
Output check criteria 202
Output Description 264, 276, 287

**P**
Package (built) 30
Package (import) 30
Palette area 32

05/20/2016/

Pattern Recognition 260
Period (grating) 186
Point (recipe) 272
Point by Point Calculated 161
Polarimetric (Measurement Mode) 174, 188
Post-calculation (Model) 206
Post-calculations (Layers) 208
Post-modeling 220
PR 260
Precision setup (modeling output 18
Pre-modeling 220
pre-modeling 224
Print 34
Profile (create) 13
Profile (user) 12
Progress Screens Setup 17
Psi and Delta 50

**Q**
Quality (MM-16) 82

**R**
R2w 46
real time acquisition 51
Real Time Graphics Screen 46
Recipe 254
Recipe (kinetic) 303
Recipe Environment Restore 114
Recipe files 277, 324
Recipe information 277, 324
Recipe Options 19
Recipe Results (complex) 275
Recipe Results files format 28
Recipes format file 28
Reflectance (AutoSE) 95
Reflectance (UVISEL) 88
Reflectance (UVISEL2) 99
repetition (layer) 173
Report activation 283, 286
Reporting 280
reports files format 29
Reprocessing 113
Results deletion 21
Right slope projection 186
Running DeltaPsi2 11
Rw 46

**S**
S0 46
S0 (range setup) 45
S0 Min/Max 54, 60, 296
S1 46
S2 46
Sample format file 28
Sample Vision (UVISEL2) 68
Screen management 27
script 220
Script (Modeling Scenario) 221
Search feature 24
Segments number 178, 192
Serial Multimodel 210
Serial Multimodel format file 28
Shape 179, 193
Show/Hide button 25
shutter 44
Simple Grid 226
Simplex 195
Simulated Acquisition 109
Simulation 173
Simulation (kinetic) 303
slope (grating) 186
Smart merge 53
Software options 14
Spectral Data 174, 189
Spectral Range 174, 188
Spectral Range (MWL) 62, 73
Spectral Resolution 191, 192
Spectroscopic Acquisition 51
Spectrum Range 55
SSC 200
Standard merge 53
Start 51
Steps 278, 325
Steps (recipe) 259, 271
Stop (motors) 51
substrate format file 28
Substrate Setup 228, 246

**T**
Tables 278, 325
Target value 51
TCC Server 315
Temperature Routine (create) 319
Template (create report) 280
templates files format 28
Text Display Mode 40
Text format 293
Text Mode Switch 33
Thickness (fitting) 196
Thickness calculation 207
Thickness Fitting 196
Thickness table 207
Tilt Vision 68
Title Font Size 37
Tool Bar setup 16
Transmittance (AutoSE) 95
Transmittance (UVISEL) 88
Transmittance (UVISEL2) 99
TreeView 11, 24
TreeView description 28
TreeView Options 29
Troubleshooting 327

**U**
UDF 160
Up 51
Update Models 32
USB Server 25
User defined formula 160
User Library 29
User Target (MM-16) 82
User unit (Spectro. Acq.) 54
UV Filter 44
UV protection 54, 295
UVISEL VIP
acquisition 87

05/20/2016/

Autofocus adjustment 144
Getting Started session 141
Views 86
UVISEL2 65

**V**
V.A.S.E. (Measurement Mode) 174, 189
Vacuum activation (stage) 49
value (motors) 51
View screen 22
Views (UVISEL2) 65
Views MM-16 75
Views screen 43
Volume fraction 199

**W**
Warranty 4

**X**
XYZ Stage 49

**Z**
Zoom in 33
Zoom out 33, 34

05/20/2016/