

UV WINLAB SOFTWARE



User's Guide

Release Date	Version
November 2011	UV WinLab Software 6.0.4

Any comments about the documentation for this product should be addressed to:

User Assistance
PerkinElmer
Chalfont Road
Seer Green
Beaconsfield
Bucks.
HP9 2FX

Or emailed to: info@perkinelmer.com

Notices

The information contained in this document is subject to change without notice.

Except as specifically set forth in its terms and conditions of sale, PerkinElmer makes no warranty of any kind with regard to this document, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose.

PerkinElmer shall not be liable for errors contained herein for incidental consequential damages in connection with furnishing, performance or use of this material.

Copyright Information

This document contains proprietary information that is protected by copyright. All rights are reserved. No part of this publication may be reproduced in any form whatsoever or translated into any language without the prior, written permission of PerkinElmer, Inc.

Copyright © 2011 PerkinElmer, Inc.

Produced in the UK.

Trademarks

Registered names, trademarks, etc. used in this document, even when not specifically marked as such, are protected by law.

PerkinElmer is a registered trademark of PerkinElmer, Inc.

Table of Contents

Introduction	11
The UV WinLab HTML Help System	12
Security	13
Security Settings	14
Users.....	15
Groups	18
Locking out users.....	24
Passwords	26
Summary.....	29
Checksum Failures	30
Signatures	32
Audit Trail.....	36
Login History	38
Explorer Window.....	41
The Explorer	42
Folders	49
Methods	55
Viewing Methods in the Explorer.....	55
Creating Methods	56
Using the New Method Wizard.....	57
Using an existing method to create a new method.....	58
Reviewing and approving Methods.....	61
Viewing, editing and running methods	63
Example Methods for Medium Performance Instruments.....	65
ASBC Color and Turbidity in Beer	67
Bilirubin in CSF (UK Method)	68
Bradford Protein Assay	70
Chlorophyll a, b and c (Trichromatic) Determination	72
Chlorophyll Determination (US10200H).....	74
Cytochrome P450 Determination	76
G6PD Kinetics	78
EU Olive Oil Determination.....	79
Lambda Max	81
Lowry Protein Concentration	83
Naphthalene (ASTM D1840-03).....	85
Nitrate Determination (EPA 352.1/US Test Method 9200)	87
Nucleic Acid Concentration and Purity.....	89
Percentage Strength at 500 nm.....	91
Phosphate in Water (EPA 365.2 & 365.3/DIN 38 405)	92
Porphyrin Scan.....	94
Sulfite in Food by Enzymatic Determination	95
Warburg–Christian Assay for Protein and Nucleic Acid Concentration.....	97
Wine Color Analysis.....	99
Scan and Second Derivative	101
Example Methods for High Performance Instruments	103
Bandpass Filter Characterization.....	104
Camouflage Using Integrating Sphere	105
Color Difference (Delta E)	106
Measurement of Glass and Architectural Materials to EN410	108
Cut-Off Filter Characterization	109
Film Thickness using Interference Fringes.....	110
Opacity Calculation for Paints and Coatings.....	111
Haze ASTM D-1003	113
HP Reflectance Color User Selectable	115
Simple Transmittance Color Method 5 nm.....	117

4 . UV WinLab Software Guide

Terrestrial Solar Irradiance.....	119
Whiteness & Yellowness Index	120
Tasks.....	121
Viewing tasks in the Explorer	121
Creating tasks from methods	123
Running and saving tasks	124
Viewing tasks and re-processing data.....	127
Reviewing and approving tasks.....	129
Instruments	132
Viewing Instruments in the Explorer.....	132
Adding and deleting instruments.....	133
Adding a Medium performance instrument (Lambda 25, 35, 45, 20, 40, 40P, 20Bio and 40Bio)	134
Adding a High performance Instrument (Lambda 650, 650R, 650S, 750, 750S, 850, 850R, 800, 900, 950, 1050 NB and 1050 WB)	135
Instrument Performance Verification	139
Instrument Details	139
Instrument Properties	139
Instrument Event Log.....	143
Calibrate Instrument	143
Manual Control	147
Autozero.....	148
Calibrate Instrument	148
Instrument Properties	153
Optimize Gains.....	155
IPV Setup.....	157
Creating an IPV Setup	157
Saving an IPV Setup.....	161
Importing and exporting IPV Setups.....	161
IPV reports and summaries.....	162
Running IPV tests	164
How frequently can I perform an IPV test?	164
Reviewing, approving and locking an IPV Setup	166
IPV Test Subsets.....	168
Absorbance Zero Stability Test	169
Baseline Stability Test	174
Noise	178
Noise at Wavelength 1 Test.....	179
Noise at Wavelength 2 Test.....	183
Noise at Wavelength 3 Test.....	187
Photometric Accuracy	191
Photometric Accuracy K2Cr2O7 Solution Test	192
Photometric Accuracy K2Cr2O7 Solution Test at 430 nm	199
Photometric Accuracy with Potassium Nitrate Solution Test	206
Photometric Accuracy Glass.....	211
Photometric Linearity Grey Glass Test.....	217
Photometric Repeatability Glass Test	222
Resolution	228
Resolution with Benzene Vapor Test	229
Resolution with Toluene Solution Test.....	234
Resolution with Toluene in Methanol Test	240
Stray Light	245
Stray Light with Potassium Chloride Solution at 200 nm Test.....	247
Stray Light with Potassium Chloride Solution Test.....	253
Stray light with Sodium Iodide (NaI) Solution at 220 nm Test.....	259
Stray light with Sodium Nitrite (NaNO ₂) at 340 nm and 370 nm Test.....	264
Wavelength Accuracy.....	270

Wavelength Accuracy with Deuterium Lamp Test	271
Wavelength Accuracy Check with Solution Test.....	276
Wavelength Accuracy with Glass Filters Test	282
Wavelength Accuracy with Third-Party Lamp Test	287
Wavelength Repeatability D2 Test.....	292
Running an IPV.....	296
Queries.....	300
Sample Queries.....	303
IPV Queries	306
Calibration Queries	308
The Query Window	311
Results Table	315
Display Pane	316
Adding Comments	318
Shewhart Chart.....	319
Quartile Chart	322
Histogram.....	325
Copying a Spectrum, Chart or Table	327
Displaying Multiple Spectra	328
Printing a Query	329
Queries – Options dialog.....	330
Columns	331
Results Tree	332
Reviewing and Approving Results.....	337
Add Spectra	339
Report Templates.....	340
Reports	343
Recycle Bin	344
Database Tools	345
Legacy File Converter	350
The Workspace.....	351
The Workspace	352
Method.....	360
Task.....	362
The Instrument Status Bar.....	363
Data Collection.....	364
Medium Performance Instruments Data Collection.....	365
Scan Task – Data Collection	366
Wavelength Task – Data Collection	369
Timedrive Task – Data Collection	373
Manual Control – Data Collection.....	376
High Performance Instruments – Data Collection.....	379
D2 Lamp	379
Tungsten Lamp	380
Filter	380
Lamp Change	382
Monochromator	382
Cycles	382
Common Beam Mask (CBM)	383
Slits	383
Detector Settings.....	385
Beam Selection.....	387
Common Beam Depolarizer (CBD)	387
Rear and Front Beam Attenuators.....	388
Depolarizer/Polarizer.....	389
Sample Compartment	390
Detector Change	390
Accessories	400

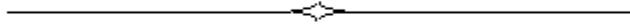
6 . UV WinLab Software Guide

Scan Task – Data Collection	405
Timedrive Task – Data Collection	407
Wavelength Task – Data Collection	409
Polarization Task – Data Collection	412
Manual Control – Data Collection	414
Program.....	417
Formatting the Program Table	418
Instrument Settings	419
Accessory.....	425
Sipper Accessory	426
Peltier Accessory	428
Autosampler Accessory	430
Autosampler Accessory – Goto position	435
Autosampler Accessory – Depth.....	438
Cell Changer Accessory	439
Cell Changer for Medium Performance Lambda Spectrometers (Lambda 20, 40, 40P, 20Bio, 40Bio, 25, 35, 45)	439
Cell Changer for High Performance Lambda Spectrometers (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, 1050 WB and 1050 NB)	441
Manual Control for Cell Changer.....	445
Universal Reflectance Accessory (URA)	446
Data Collection	446
Angle	446
Spot size	447
Actions	448
Corrections and baselines	448
Small Samples	449
URA Optics	449
Corrections.....	453
Baseline Corrections	453
Reflection Corrections	457
Attenuator Corrections	458
Expire Corrections	459
Viewing Corrections.....	459
Accessories.....	462
Sample Info	463
Adding and deleting samples	463
Formatting the Sample Table	469
Measurements and replicates.....	473
Control samples	476
Data Collection Settings.....	477
Accessory settings.....	481
Custom columns	487
Formatting columns	488
Using a pre-defined Sample Table.....	494
Running a Sample Table.....	494
Processing.....	498
Equation	501
Conditional Formatting	511
Arithmetic	513
Adding Spectra	515
Subtracting Spectra	516
Multiplying Spectra	517
Dividing Spectra	518
Convert X and Convert Y	519
Convert X.....	520
Convert Y	521

Derivative	523
Difference.....	526
Interpolate.....	527
Normalize	528
Peak Table.....	530
Smooth	533
Quant.....	535
Beer's Law Quant.....	536
Calibration.....	537
Limits.....	539
Scanning Quant Parameters.....	541
Wavelength Quant Parameters.....	543
Quant Calibration	544
Setting up the Standards Table	544
Measurements and replicates	550
Rate	554
End Point Analysis	556
Slope Analysis	560
Results	563
Custom Columns	567
Formatting Columns.....	568
Output.....	573
Templates.....	573
Report frequency.....	574
Report output.....	574
Data Export.....	575
Call Application.....	583
Graphs	586
Audit Trail.....	596
Event Logs.....	597
Method Event Log.....	597
Task Event Log.....	597
Sample Event Log.....	598
Welcome to Communiqué.....	599
Welcome to Communiqué	600
Working with Templates	602
Working with Margins.....	605
Working with Headers and Footers.....	606
Working with Page Types	608
Working with the Tree View	611
Using the Layout Tools	612
Text Block	613
Table.....	617
Graphic.....	626
Equation	629
Page Break	634
Edit Border	635
Edit Caption	637
Section	639
Create Group	642
Insert H Line.....	644
Insert V Line.....	646
Using Data Objects.....	648
Using Custom Objects	657
Indexing	659
Laying out items	661
Using the Rulers and Grid	663
Using Communiqué Utilities	664

Reports Filter.....	667
Templates Filter.....	668
Archiving Data.....	669
Previewing a Template.....	670
Reviewing and Approving	671
Printing a Template	672
Displaying the Audit Trail and Event Log	673
Understanding the Menu Commands.....	675
Other Interactions	679
UV WinLab Data Objects	681
UV WinLab Data Objects	682
Global	683
Task List	700
Task List Task Samples	705
Raw Spectrum/Processed Spectrum.....	718
Wavelength Data.....	728
Peak Table	730
Table Columns	733
Sample Event Log.....	736
Corrections.....	743
Raw Points/Processed Points.....	752
Task List – Table Objects	759
Task List – Method	763
Task List – Created By/Modified By	771
Task List – Instrument	775
Task List – Quant Calibration.....	779
Calibration Graphs.....	779
Calibration Details	782
Standards Table.....	787
Standards.....	788
Task List – Task Event Log	798
Task List – Measurements or Replicates Table Samples.....	804
Task List – Custom Table Samples	811
Method List	813
Data Collection Settings	819
Timedrive.....	834
Corrections Settings	836
Quant Settings	840
Rate Settings.....	849
Pre-Processing and Post-Processing	859
Sample Table Definition	864
Standards Table Definition	870
Results Table Definition	876
Reporting Options.....	880
Method Event Log.....	894
Sample List	900
Sample Table and Results Table	904
Analyst.....	914
Spectra	917
Wavelength Data.....	926
Peak Table	928
Table Columns	931
Sample Event Log	934
Corrections.....	942
Data Points	952
User List	959
Instrument List.....	962
Query Results List.....	966

System Data	972
Miscellaneous.....	974
Templates.....	981
Reports	989
Communiqué FAQ	993
Further Information.....	999
Further Information.....	1000
Recommended Practices.....	1001
Handling Calibration Standards	1002
Choice of Solvent	1003
IPV References	1004
UV Spectroscopy	1006
Electronic Transitions	1007
Sources of Degradation of Spectrophotometric Performance	1009
Environment.....	1009
Line Power Supply	1009
Aging Lamp.....	1009
Alignment	1009
Warm-up Time	1009
Sample Handling	1009
Aging of Instrument.....	1010
Proper Care of the Instrument.....	1010
Frequently Asked Questions (FAQs).....	1011
Administration	1011
Databases and File Structure.....	1011
Methods and Tasks.....	1013
IPV (Instrument Performance Verification)	1016
Printing	1017



Introduction

The UV WinLab HTML Help System

This document is a version of the Help file formatted for printing.

Throughout the document we refer to two types of instrument supported by the software:

- Medium performance instruments – Lambda 20, 40, 40P, 20Bio, 40Bio, 25, 35, 45.
- High performance instruments– Lambda 650, 650S, 750, 750S, 800, 850, 900, 950, 1050 NB and 1050 WB

Security

Security Settings

There are two versions of UV WinLab – Standard Security and Enhanced Security (ES). Some of the settings described in these topics apply to both versions of the software, whereas others vary depending on the type of security that you have. Where options vary, both are explained.

The Security settings are only available to Administrators. From the Setup Users and Groups dialog (accessed by selecting **Setup Users and Groups** from the Administration menu in the Explorer), you are able to set up and edit users, groups, and password settings.

Within the Enhanced Security version of UV WinLab, Administrators are also able to define the signature points within the software, and view the Audit Trail, Login History and Summary.

Click on a link below for further information.

[Users](#)

[Groups](#)

[Locking out users](#)

[Passwords](#)

[Signatures](#)

[Audit Trail](#)

[Login History](#)

[Summary](#)

[Checksum Failures](#)

Users

What are the default users?

The following users are pre-defined in UV WinLab:

Standard version – Administrator, Analyst, PEDeveloper, PEService.

Enhanced Security version – Administrator, Analyst, Approver, Database Manager, Developer, PEDeveloper, Reviewer, PEService, Supervisor.

The Administrator is purely for setting up users, groups, passwords and signature points (signature points are only available in the Enhanced Security version of UV WinLab). They do not have access to any functionality within the rest of the software. To use the software, a person must be a member of one of the other default groups or a member of a new group that the Administrator creates.

It is possible to change the group membership of the default users.

NOTE: By default, in the Enhanced Security version of UV WinLab, the default users are only members of the default group with the same name. For example, the Analyst user is a member of the Analysts group, and the Developer is a member of the Developers group.

NOTE: In the Standard version of UV WinLab, the Administrator is a member of the Administrators group, the Analyst and PEDeveloper are members of the Users group, and PEService is a member of the Administrators and Service groups.

How do I add a new user?

1. If the Setup Users and Groups dialog is not displayed, select **Setup Users and Groups** from the Administration menu.

The Setup Users and Groups dialog is displayed.

2. Select the Users tab and click **New**.

A New User dialog is displayed.

3. Enter the User name, Full name, Password, and repeat the Password in the Confirm password entry field.

The password length is defined on the Password Control tab.

NOTE: The Password is case-sensitive. It can consist of letters, numbers and single spaces only.

4. Select **Enabled** if you wish the user to be able to login or **Disabled** if you do not wish them to be able to login at the current time.

5. If **Enabled** is selected, select **User must change password at next login** to force the user to change their password when they log in.

NOTE: In the Enhanced Security version of UV WinLab, **User must change password at next login** is always selected, forcing the user to change their password when they log in for the first time.

6. Click **OK**.

The **User name** drop-down list is updated with the new user.

How do I delete a user?

1. Select the user from the **Name** drop-down list on the Users tab and then click **Delete**.
A message will be displayed asking you to confirm that you wish to delete the user.
2. Click **Yes**.
The user is deleted.

NOTE: It is not possible to reuse a **User name** that has been deleted.

NOTE: It is not possible to delete the Administrator from the Administrators group.

How do I assign a user to a Group?

Users can be assigned to one or more groups.

1. Select the user from the **Name** drop-down list on the Users tab.
2. Select the Group from the list of **Available groups for user** and then click **Add**.
The Group is added to the **User is a member of** list.

NOTE: When a group is added to the **User is a member of** list, it no longer appears in the **Available groups for user** list.

How do I remove a user from a Group?

- To remove a user from a group, select the Name on the User tab, then, select the group from the **User is a member of** list, and click Remove.
The user is removed from the group.
The group is removed from the **User is a member of** list, and is added to the **Available groups for user** list.

How do I reinstate Locked Out Users?

If the Lockout is set to **Permanent** and the user has failed to login correctly within the allowed number of attempts, the administrator must assign a new password before they are able to login again.

When the administrator next logs in after a user has been locked out, a list of Locked Out Users is displayed.

1. Highlight the name of the user that you wish to reinstate and then click **Edit**.
The Edit User dialog is displayed.
2. Enter a new **Password** and repeat it in the **Confirm password** field.
3. Click **OK**.
The user is removed from the list of Locked Out Users.
4. Click **OK** to close the Locked Out Users dialog.
The previously locked out user will now be able to login using the new password, which they will be forced to change if you are using the Enhanced Security version of UV WinLab. If you are using the Standard version, the user will only be forced to change the password if **User must change password at next login** was selected.

NOTE: If you click **OK** rather than **Edit** when the list of Locked Out Users is displayed, the list is closed and the Explorer starts. Any locked out users will remain locked out. The list will be redisplayed each time you login until any locked out users have each been assigned a new password.

NOTE: Users locked out for a specified duration can be unlocked by the administrator in the same manner.

How do I disable an existing individual user?

1. From the Administration menu, select **Setup Users and Groups**.
The Setup Users and Groups dialog is displayed.
2. Select the **Users** tab.
3. Select the user's **Name** from the drop-down list and click **Edit**.
An Edit User dialog is displayed.
4. To disable the user, select **Disabled**.
When the disabled user attempts to login an error message will be displayed informing them that their login failed.
5. Click **OK**.
The **User** is disabled. The Group membership section of the Setup Users and Groups dialog is grayed when a disabled name is selected in the Name list.

NOTE: To enable the user, select **Enabled** on this dialog. You must also enter and confirm a new password when enabling a user.

NOTE: All user updates are recorded in the audit log in the Enhanced Security version of UV WinLab.

NOTE: In the Enhanced Security version of UV WinLab, the Administration [Audit Trail](#) records all changes to security settings in compliance with 21 CFR Part 11. All changes to users, groups, password settings and signature points are recorded.

Groups

What are the default groups and what can members of these groups do?

The following table lists the default groups together with what members of the group are able to do:

NOTE: The information in the table below refers to the Enhanced Security (ES) version of the software.

User	User is a member of group	Member of the group is able to
Administrator	Administrators	<p>NOTE: This group cannot be edited.</p> <p>The permissions for the Administrator are not listed. The Administrator is only able to perform Administration tasks – setup users, groups and passwords. They can also un-delete reports and report templates.</p>
Database Manager	Database Managers	Create and edit methods and IPV set-ups, Manage and delete methods and IPV set-ups, Manage tasks, Manage the database.
Analysts	Analysts	Continue tasks, Run calibrations, Run methods, Run queries.
Supervisors	Supervisors	Configure instruments, Edit and save calibrations, Edit queries, Print reports, Run instrument performance verifications, Run methods, Run queries.
Developers	Developers	Create and edit methods and IPV setups, Edit and save calibrations, Edit queries, Edit report templates, Manage and delete methods and IPV set-ups, Print reports, Re-process results, Run calibrations, Run instrument performance verifications, Run methods, Run queries, Run calibrations, Continue tasks.
Reviewers	Reviewers	Edit and save calibrations, Edit queries, Edit report templates, Manage and delete methods and IPV set-ups, Print reports, Review methods and IPV setups, Review report templates, Review reports, Review results, Run calibrations, Run instrument performance verifications, Run methods, Run queries.
Approvers	Approvers	Approve methods and IPV setups, Approve report templates, Approve reports, Approve results, Edit and save calibrations, Edit queries, Edit report templates, Manage and delete methods and IPV set-ups, Print reports, Run calibrations, Run instrument performance verifications, Run methods, Run queries.

NOTE: The Administrators group is not listed on the Groups tab because the group does not have any permissions in the software apart from setting up Users' Groups and Signature points. It is not possible to change any of the permissions associated with the Administrator's group, or delete the Administrators group.

NOTE: If you are using the Standard Security version of the software, the system provides three pre-configured groups, Administrator, User and Service. Users who are members of the User group have all access rights or permissions in the system other than Administrator and Service rights.

How do I create a new group?

NOTE: It is not possible to create a new group within the Standard version of UV WinLab.

1. From the Administration menu, select **Setup Users and Groups**.
The Setup Users and Groups dialog is displayed.
2. Select the Groups tab and click **New**.
A New Group dialog is displayed.
3. Enter a **Group name** and click **OK**.
The drop-down list is updated to include the new group.
4. Select the permissions for the Group.
By default, none of the options in the **Permissions** list are selected.
5. Click **OK**.
The new Group is created.

What are the available permissions?

The table below lists the available permissions within UV WinLab and what the permission allows the user to do.

Approve methods and IPV set-ups	Approve locked methods and locked IPV set-ups.
Approve own results	<p>Allows a user to approve their results within the Workspace or Results Browser.</p> <p>NOTE: Run queries permission is needed in addition if you wish to approve results from within the Results Browser.</p>
Approve report templates	Approve report templates. (This sets the status of the report template to Approved)
Approve reports	Approve reports. (This sets the status of the report to Approved)

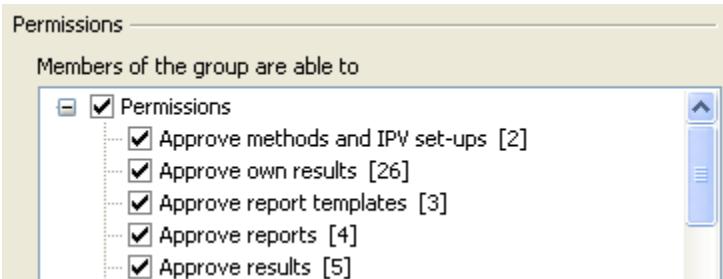
Approve results	Approve a task. (This sets the status of the task to Approved). NOTE: Run queries permission is needed in addition if you wish to approve results from within the Results Browser.
Configure instruments	Add new instruments, delete instruments, enable instrument calibration for High performance instruments, apply an IPV setup to an instrument, set a default instrument, edit instrument.
Create and edit methods and IPV set-ups	Create, edit, copy, paste, lock and unlock methods; import and export methods, create IPV set-up and apply to an instrument, re-perform failed IPV, postpone an IPV, perform an IPV on demand.
Delete reports	Delete reports.
Delete results	Delete and restore tasks. NOTE: This permission is only available in the Standard version of UV WinLab.
Edit and save calibrations	Modify and save existing calibrations.
Edit queries	Create, edit and run results queries; cut, paste, delete and restore queries.
Edit report templates	Opens Communique report creator to create, edit and save report templates; Delete user created report templates (default templates cannot be deleted).
Manage and delete methods and IPV set-ups	Delete, restore, cut and paste methods; move methods between folders.
Manage tasks	Rename and delete folders, rename folder items, cut and paste tasks between folders.
Manage the database	Run database utilities, use legacy file converter, determine the visibility of folders, empty the recycle bin.
Print reports	Print preview and print reports.
Reprocess results	Enables a completed task to be reprocessed (a copy of the task is created and can be renamed). NOTE: Only the Reporting, Processing, Quant and Rate pages in the Workspace can be edited.
Review methods and IPV set-ups	Review a locked method or locked IPV set-up.
Review report templates	Review report templates.
Review reports	Review reports.

Review results	Review a task. (This sets the status of the task to Reviewed) NOTE: Run Queries permission is needed in addition if you wish to review results from within the Results Browser.
Run calibrations	Create and save calibrations.
Run instrument performance verifications	Perform instrument performance verifications.
Run methods	Run methods to create tasks, enables Autozero, add comments to sample table.
Run queries	Run pre-defined results queries. NOTE: The Run queries permission does not allow you to create a new query, only run a previously setup query against the current database.
Service	Access the Instrument filter properties in the Explorer and access the filter table on the Instrument page in the Workspace (High performance instruments).

How do I define what members of a group are able to do?

The permissions available to each group are selected on the Groups tab. The permissions are listed as a tree structure. When a new group is created, none of the permissions are selected by default.

1. To select all the permissions click **Permissions** at the top of the tree:



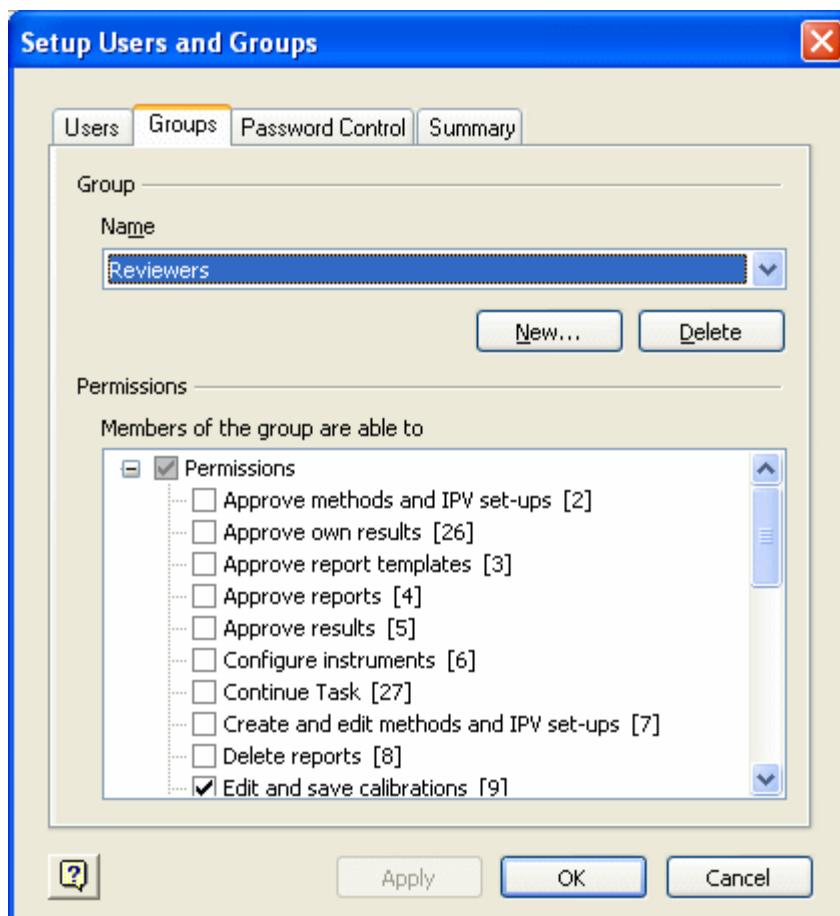
All the permissions are automatically checked.

OR

To assign one or more (but not all) permissions, click in the box next to the permission you wish to assign to the group.

A tick indicates that the permission is available for the group.

NOTE: When only some of the permissions are selected, the check box at the top of the tree is grayed to indicate that not all of the options are selected:



2. When all the required permissions are selected, click **Apply**.

The available options for the group are updated.

NOTE: The numbers at the end of each permission relate to the database. When you view the audit trail, it is these numbers that are listed rather than the description of the permission. You will need to refer back to the list of permissions on the Groups tab to find the number that relates to a particular permission, or look at the summary where the permission together with the number is given.

How do I delete a group?

1. To delete a group, select the Group from the **Name** drop-down list and then click **Delete**.

A message is displayed asking you to confirm that you wish to delete the selected group.

2. Click **Yes**.

The group is deleted.

How do I assign a user to a Group?

Users can be assigned to one or more groups.

1. Select the user from the **Name** drop-down list on the User tab.
2. Select the Group from the list of **Available groups for user** and then click **Add**.
The Group is added to the **User is a member of** list.

If you wish to create more than one new user and assign each of them to a group/groups you must click **Apply** after assigning the groups to the first user before creating the next new user otherwise the group assignments for the currently selected user will be lost.

NOTE: When a group is added to the **User is a member of** list, it no longer appears in the **Available groups for user** list.

How do I remove a user from a Group?

- To remove a user from a group, select the Name on the User tab, then, select the group from the **User is a member of** list, and click Remove.
The user is removed from the group.
The group is removed from the **User is a member of** list, and is added to the **Available groups for user** list.

NOTE: The Administration [Audit Trail](#) records all changes to security settings in compliance with 21 CFR Part 11. All changes to users, groups, password settings and signature points are recorded.

Locking out users

How do I limit the number of failed login attempts?

1. From the Administration menu, select **Setup Users and Groups**.

The Users and Groups dialog is displayed.

2. Select the Password Control tab.

3. Click Account Lockout.

The Account Lockout dialog is displayed.

4. Enter the Number of failed logins allowed before lockout.

For example, if **Number of failed logins allowed before lockout** is set to 5 failed login attempts, the user is allowed 5 failed attempts at login. On the 5th failed attempt they are locked out until the Administrator allows them access again (Permanent) or for a specified period of time (Duration). The default is lockout after 5 failed login attempts for both Enhanced Security and Standard Security. The minimum number of failed login attempts before a user is locked out is 1. The maximum number of allowed failed login attempts before a user is locked out is 10.

5. Select the Lockout Duration as **Permanent, until Administrator unlocks**, or **Duration**. If you select **Duration**, enter the time (in minutes) for the lockout.

NOTE: If **Number of failed logins allowed before lockout** is set to 1 the user will be locked out when they have one incorrect login attempt. That is, they are not allowed an incorrect login attempt, otherwise they will be immediately locked out.

What is Permanent Lockout and Duration?

Permanent until administrator unlocks means that the user will be unable to login again until the administrator has unlocked their account and assigned a new password. **Duration** prevents the user being able to login again until the time specified has elapsed. **Duration** is grayed if Permanent is selected.

If **Duration** is selected, the default is 60 minutes. The minimum **Duration** is 1 minute and the maximum **Duration** is 32767 minutes (22.75 days).

NOTE: In the Enhanced Security version of UV WinLab, details of failed login attempts are recorded in the Login History.

How do I reinstate Locked Out Users?

If the Lockout is set to **Permanent until administrator unlocks** and the user has failed to login correctly within the allowed number of attempts, the administrator must assign a new password before they are able to login again.

When the administrator next logs in after a user has been locked out, a list of Locked Out Users is displayed.

1. Highlight the name of the user that you wish to reinstate and then click **Edit**.

The Edit User dialog is displayed.

2. Enter a new **Password** and repeat it in the **Confirm password** field.

3. Click **OK**.

The user is removed from the list of Locked Out Users.

4. Click **OK** to close the Locked Out Users dialog.

The previously locked out user will now be able to login using the new password, which they will be forced to change if they are using the Enhanced Security version of UV WinLab. In the Standard version of UV WinLab, users are only forced to change their password if **User must change password at next login** has been selected.

NOTE: If you click **OK** rather than **Edit** when the list of Locked Out Users is displayed, the list is closed and the Explorer starts. Any locked out users will remain locked out. The list will be redisplayed each time you login until any locked out users have each been assigned a new password.

NOTE: Users locked out for a specified duration can be unlocked by the administrator in the same manner.

Passwords

NOTE: Passwords are case sensitive.

How do I change a Password?

1. To change a user's password, select the user from the **Name** drop-down list on the Users tab and then click **Edit**.
An Edit User dialog is displayed.
2. Enter the new **Password** and repeat it in the **Confirm password** entry field.
3. Click **OK**.

The new password is implemented. The next time the user logs in they will be forced to change their password if you are using the Enhanced Security version of UV WinLab. In the Standard version of UV WinLab, the user will only be forced to change their password if **User must change password at next login** has been selected.

NOTE: If you have the Standard version of UV WinLab, you will be unable to change the default passwords accessed via the Login dialog for the first day after installation. If you attempt to change a password, the following message is displayed: 'You have changed your password less than 1 days ago. You cannot change your password again'. This is due to the default setting for **Minimum password age**, where **Allow changes after (days)** is set to 1. If you wish to change the password within the first day, the Administrator must select the **Allow Changes Immediately** option on the Password Control tab of the Setup Users and Groups dialog.

How do I define when users must change their password?

NOTE: The settings on the Password Control tab apply to all users. It is not possible to define individual Password controls for each user.

On the Password Control tab, **Maximum password age** defines the maximum number of days that users can retain the same password before they must change it. By default the password expires after **42 days**. The minimum is 1 day and the maximum is 999 days.

The **Maximum password age** cannot be set less than or equal to the **Minimum password age**.

NOTE: If you want to set the Maximum password age to 1 day the Minimum password age must first be set to Allow changes immediately.

If it is not necessary for users to change their password, select **Password never expires**.

Within the ES version of the software users are forced to change their password the very first time they login.

How do I define the minimum length of time that users must retain the same password before they are able to change it?

On the Password Control tab, Minimum password age defines the number of days that users must retain the same password before being allowed to change it. The default is to **Allow changes after 1 days**. **Allow changes after x days** prevents users from changing their password several times in a short space of time in order to return to a previous password.

To allow users to be able to change their password immediately, select **Allow changes immediately**.

The **Minimum password age** cannot be set greater than or equal to the **Maximum password age**. The minimum is 1 day and the maximum is 999 days.

NOTE: If you want to set the Maximum password age to 1 day the Minimum password age must first be set to Allow changes immediately.

Within the ES version of the software, users will be forced to change their password the very first time they login.

Within the Standard Security version of the software, select **Change Password at First Login** to force users to change their password the very first time they login.

How do I define the length of a password?

Minimum password length on the Password Control tab defines the minimum number of characters that must be used in the password. By default, **At least 6 characters** is selected. The minimum is 1 and the maximum is 16 characters.

Allow blank password is an alternative option. This means that users are not required to enter a password on login, providing the user has previously opted to not enter a password when they last changed their password, or if they have never logged in before.

NOTE: A second blank password cannot be used immediately when a blank password expires if **Number of passwords to remember** has been selected in the Password uniqueness section of the Password Control tab.

NOTE: **Allow blank password** is not available within the Enhanced Security (ES) version of the software.

Can a password be reused?

Users are able to reuse a previous password. By default, Number of passwords to remember is set to **24**.

Password uniqueness on the Password Control tab defines the number of new passwords that must be used before a previous password can be reused. For example, if the first password is 'security', and **Number of passwords to remember** is set to 3 entries, users must use 3 other passwords in addition to their current password before they are able to reuse 'security' as their password. The minimum is 1 and the maximum is 24.

If **Do not keep password history** is selected a user is able to reuse a password whenever they wish.

NOTE: **Do not keep password history** is not available in the Enhanced Security version of UV WinLab.

Do users have to enter a password?

If you are using the Enhanced Security (ES) software, a password is mandatory. However, this is not the case within the Standard Security version of the software. Users can either be asked for just their Username or their Username and password.

1. From the Administration menu in the Explorer, select **Setup Users and Groups**.
The Setup Users and Groups dialog is displayed.
2. Select the Password Control tab.
3. In the Login and Type section, select either **PerkinElmer Login** or **No Passwords Login**.
PerkinElmer Login requires the user to enter the Username and Login. No Passwords Login requires just the Username.

NOTE: If **No Passwords Login** is selected, the user will be able to select their user name from a drop-down list when the login dialog is displayed. If a password is required, the user will have to enter their name manually as a drop-down list will not be available (for security reasons).

Is there a record of previous passwords?

Within the ES version of the software a record of previous passwords is automatically made and there is no way of turning this off.

Changes to passwords are recorded in the Audit Trail. However the actual passwords are not visible; the word **Hidden** is displayed instead.

Within the Standard Security version, select **Do not keep password history** if you do not wish to retain a record of the number of previous passwords.

NOTE: The Administration [Audit Trail](#) records all changes to security settings in compliance with 21 CFR Part 11. All changes to users, groups, password settings and signature points are recorded.

Summary

NOTE: The Summary is only available in the Enhanced Security version of UV WinLab.

NOTE: The Summary can only be viewed by the Administrator.

What is included in the Summary?

The Summary records all information about the security settings:

- Password control – it records login type, maximum password age, minimum password age, minimum password length, password uniqueness, lockout count and lockout duration.
- Permissions – it records the number of permissions and lists all the permissions with their associated number.
- Users – it records the number of users. For each user it records the username, full name, status, last login, the group the user belongs to, and the permissions of that group.
- Groups – it records the number of groups. For each group it records the group name, the users in the group, and the group permissions.

How do I view the Summary?

1. From the Administration menu in the Explorer select **Setup Users and Groups**.
The Setup Users and Groups dialog is displayed.
2. Select the Summary tab.
The summary is displayed.

How do I print the Summary?

- To print the Summary click Print.
All the information is printed.

How do I export the Summary?

1. To export the Summary click **Export**.
A Save As dialog is displayed.
2. Select the required destination and enter a filename.
The summary is exported as a *.csv file and can be opened, for example, in Microsoft Excel.

Checksum Failures

NOTE: Checksums (and hence Checksum failures) are only applicable in the Enhanced Security version of UV WinLab.

NOTE: The only remedy to an error message stating there is a checksum failure, and preventing you from accessing UV WinLab, is to restore from a backup database.

Why did a checksum failure occur?

UV WinLab ES uses a variety of security techniques to ensure that files cannot be tampered with either accidentally or deliberately. Checksums ensures that data has not been tampered with. Under normal operation checksums are used in the application to validate the data security, however, a checksum failure can occur after a number of situations:

- Hard disk failure
- Power failure
- Software crash, either the Application or Windows or another application
- Deliberate attempt to falsify data.

If they occur then the reasons for them should be investigated and the reasons understood, before simply recovering from the problem.

What should I do if the UVWinLab database gets a checksum failure?

If the UVWinLab database gets a checksum failure the software will continue operating but the data in error will not be accessible. You will receive an error message if you try to open an invalid method or task.

It is essential that backups are regularly made of the Repository in order to recover from a checksum failure. The Windows Administrator can restore from this backup if the database becomes corrupt as follows:

1. Log on as Windows Administrator.
2. Rename or move UVWinLab.mdb from C:\Documents and Settings\All Users\Application Data\PerkinElmer\UVWinLab
3. Copy your backup file as UVWinLab.mdb to replace the old one.

It should then be possible to view all the data again. Data collected after the last backup will be lost when the backup is restored.

What should I do if the Security database gets a checksum failure?

It is essential that backups are regularly made of the security database in order to recover from a possible database failure. This should be done at the same time as the UVWinLab database is backed up.

In addition, the security system automatically backs up the users.mdb database at the end of a session, and on exit from the Administration dialog, in a subdirectory called \Backup as C:\Documents and Settings\All Users\Application Data\PerkinElmer\SecuritySystem\backup\users.bak

The Windows Administrator can restore from this database if the active database becomes corrupt and gives a checksum failure as follows:

1. Log on as Windows Administrator.
2. Rename or move users.mdb from C:\Documents and Settings\All Users\Application Data\PerkinElmer\SecuritySystem
3. Copy \backup\users.bak as users.mdb to replace the old one.

It should then be possible to log on again. Some data may be lost if there were any changes to the database that were not backed up.

What databases and files should be backed up?

It is essential that backups are made regularly of key files and databases in order to secure the data in case of computer failure or accidental loss or damage, or even intentional damage.

The following files/directories must be backed up:

- C:\Documents and Settings\All Users\Application Data\PerkinElmer\SecuritySystem\users.mdb
This is the security database of users, groups, passwords and permissions.
- C:\Documents and Settings\All Users\Application Data\PerkinElmer\SecuritySystem\backup\users.bak
This is a backup of the security database automatically created by UV WinLab.
- C:\Documents and Settings\All Users\Application Data\PerkinElmer\UVWinLab\Communique.mdb
This is the database of reports and report templates.
- C:\Documents and Settings\All Users\Application Data\PerkinElmer\UVWinLab\UVWinLab.mdb
This is the database of methods and tasks.

NOTE: Backups of **users.mdb** and **UVWinLab.mdb** should be made at the same time to ensure that they are synchronized. If they are not synchronized and a problem occurs that means a backed-up database needs to be restored, there may be issues when attempting to use the UV WinLab Explorer.

Signatures

NOTE: Signatures are only available in the Enhanced Security version of UV WinLab.

What is an electronic signature?

An electronic signature as defined by 21 CFR Part 11 means a computer data compilation of any symbol or series of symbols executed, adopted, or authorized by an individual to be the legally binding equivalent of the individual's handwritten signature.

What is a Signature Point?

A Signature Point is a point in the software that requires a signature. The Signature Points are pre-defined. For example, one Signature Point is Save Method, so when a method is saved it requires a Signature and a dialog automatically appears. The user has to enter their User name, Password (if this option has been selected) and Reason for saving the method. They may also be able to add additional comments if this option has previously been selected by the Administrator.

What Signature Points are available in UV WinLab?

The following Signature Points are available:

- Approve Instrument Performance Verification Results
- Approve Instrument Performance Verification Setup
- Approve Method
- Approve Report
- Approve Report Template
- Approve Results
- Archive Communique Logs
- Archive Communique Reports
- Archive Communique Templates
- Calibrate Instrument
- Delete Method
- Delete Report
- Delete Report Template
- Delete Results
- Exclude measurement from results or calibration
- Hold Report
- Lock Instrument Performance Verification setup
- Lock Method
- Print / Reprint Report
- Reset Lamp Usage
- Restore Method from Recycle Bin
- Restore Report from Recycle Bin
- Restore Results from Recycle Bin
- Review Instrument Performance Verification Results
- Review Instrument Performance Verification Setup
- Review Method
- Review Report

- Review Report Template
- Review Results
- Save Calibration
- Save instrument Performance Verification Results
- Save Instrument Performance Verification Setup
- Save Method
- Save Modified Calibration
- Save partial results
- Save Report Template
- Save Results
- Unlock Method.

What settings are available for a Signature Point?

The list of Signature Points within the software are already defined. The Administrator is able to define the settings (that is, whether a signature and comments are required) for each Signature Point individually or apply the same settings to all Signature Points. In addition, the Administrator defines the list of reasons that may have caused each Signature Point to occur. The user then selects a reason from this pre-defined list in the Signature Point dialog.

A Signature Point will only require a signature if **Signature required** is selected. Otherwise, the software will ignore the Signature Point and the user will not be prompted for a signature.

Defining the settings for each Signature Point

1. From the Administration menu in the Explorer, select **Signature Points**.
The Signature Points dialog is displayed.
2. Select the Signature Point **Name** from the drop-down list of available names.
3. If a Signature is required for a Signature Point, select **Signature required**.
4. If you wish the user to be able to add comments if required, select **Prompt for comments**.

When the Signature Point dialog is displayed in the software, the user will be prompted to select a reason. For example, if they are rejecting data because they used the wrong method, they would select **wrong method implemented** from the drop-down list of available reasons. The list of reasons is also defined on this tab.

1. To add a new reason, click **New** and enter the new Reason.

OR

To delete a reason, select the Reason from the Reasons list and click **Delete**.

OR

To edit a reason, select the Reason from the Reasons list, click **Edit** and modify the text.

2. Repeat steps 5 and 6 as many times as necessary to add reasons for the Signature Point.
3. Repeat steps 3 to 6 for each Signature Point **Name**.

Defining the same settings for all Signature Points

1. From the Administration menu in the Explorer, select **Signature Points**.
The Signature Points dialog is displayed.
2. To define the same settings for all Signature Points, click **Update All**.
The Update All Signature Points dialog is displayed.
3. In the Require Signature section, select either All Points require a signature, No Points require a signature, or Do not change the current settings.
If **Do not change the current settings** is selected, no change will be made to the **Prompt for Comments** settings.
4. In the Prompt for comments, select either All Points require a prompt, No Points require a prompt, or Do not change the current settings.
If **Do not change the current settings** is selected, no change will be made to the **Prompt for Comments** settings.
5. Click **OK**.
The Update All Signatures dialog closes and the Signature Points dialog is re-displayed.

NOTE: If Signature required and Prompt for comments are not selected, when a signature point occurs in the software a reason drop-down list will still appear and the user will be required to select a reason. To prevent the dialog appearing, all reasons for the particular Signature point must be deleted.

How do I add a reason for a Signature Point?

1. Select the **Name** of the Signature Point to which you wish to add a new reason.
2. Click **New**.
The New Reason dialog is displayed.
3. Enter the new reason and click **OK**.
The reason is added to the list of reasons for the Signature Point.

How do I edit a reason for a signature?

1. From the Administration menu in the Explorer, select **Signature Points**.
The Signature Points dialog is displayed.
2. Select the **Reason** to edit from the list in the **Text** field.
3. Click **Edit**.
The Edit Reason dialog is displayed.
4. Edit the **Reason** and click **OK**.
The Edit Reason dialog closes and the updated reason appears in the **Text** field. The changes are recorded in the Audit trail.

How do I delete a reason for a signature?

1. From the Administration menu in the Explorer, select **Signature Points**.
The Signature Points dialog is displayed.
2. Select the **Reason** to delete from the list in the **Text** field.
3. Click **Delete**.
A message is displayed asking to confirm the deletion.
4. Click **OK** to confirm.
The Reason is deleted from the **Text** field. The changes are recorded in the Audit trail.

How do I reorder the reasons listed for a Signature Point?

The order of the reasons in the **Text** field is the order in which the reasons will appear in the drop-down list that the user will see.

To reorder the list:

1. From the Administration menu in the Explorer, select **Signature Points**.
The Signature Points dialog is displayed.
2. Select the **Reason** that you wish to move the position of.
3. Use the arrow keys on the right hand side of the list to move the **Reason** up or down as required.

Audit Trail

NOTE: The Audit Trail is only available in the Enhanced Security version of UV WinLab.

NOTE: The Administration Audit Trail can only be viewed by the Administrator.

The Administration Audit Trail records all changes to security settings in compliance with 21 CFR Part 11. All changes to users, groups, password settings and signature points are recorded. It also records when the Login History or Audit Trail have been exported and cleared.

How do I view the Audit Trail?

1. From the Administration menu in the Explorer select **View Audit Trail**.
The View Audit trail dialog is displayed.
2. Select the Audit Trail tab.
The audit trail is displayed. For each change recorded, the following information is given in the Audit Trail:
 - **Function**– the item that was changed, for example, Add New User
 - **Previous Value**– the state of the item before it was changed
 - **Current Value** – the new state
 - **Full Name** – the full name of the user who made the change
 - **User Name**– the login user name of the user who made the change
 - **Date Modified**– the date and time of the change.

NOTE: When permissions are added or deleted for a particular [group](#), the Audit trail entry is a number rather than the full permission name. To relate the number to a permission, select Setup Users and Groups from the Administration menu and then select the Groups tab. Each of the permissions listed has the related number against it. The permission and associated number are also displayed in the [summary](#).

How do I print the Audit Trail?

- To print the Audit Trail click Print.
All the information currently held in the Audit Trail is printed.

How do I export the Audit Trail?

1. To export the Audit Trail click **Export**.
A Save As dialog is displayed.
2. Select the required destination and enter a filename.
The Audit Trail is exported as a *.csv file and can be opened, for example, in Microsoft Excel.

NOTE: Exporting the Audit Trail does not clear the Audit Trail details from the dialog.

How do I clear the Audit Trail?

- To clear the Audit Trail, click Clear Audit.
You will be asked to confirm that you want to clear the Audit Trail log. All details are removed.

NOTE: It is only possible to clear Audit Trail entries that have previously been exported. If the Audit Trail contains additional entries from after it was last exported, only those entries that have been exported will be deleted. If none of the entries have been exported, a warning message will be displayed when you attempt to clear the Audit Trail.

Login History

NOTE: The Login History is only available in the Enhanced Security version of UV WinLab.

The Login History can only be viewed by users who are members of a group that has permission to perform administration tasks.

How do I view the Login History?

1. From within the Explorer select **View Audit trail** from the Administration menu.
The View Audit trail dialog is displayed.

2. Select the Login History tab.
The login history is displayed. This details every login attempt, since the history was last cleared, by:
 - **Full Name**
 - **User Name**
 - **Status** – **OK** indicates that the user logged in with the correct password, **Failed** indicates that a login was attempted with an incorrect password.
 - **Logged In** – date and time.
 - **Logged Out** – date and time.

NOTE: If a non-existent **User Name** is entered during login a failed login attempt is recorded. **Not Found** is entered in the **Full Name** field of the Login History, and the incorrectly entered **User Name** is also recorded.

NOTE: There is no limit to the size of the Login History.

How do I print the Login History?

- To print the Login History click Print.
All the information currently held in the Login History is printed.

How do I export the Login History?

1. To export the Login History click **Export**.
A Save As dialog is displayed.

2. Select the required destination and enter a filename.
The Login History is exported as a *.csv file and can be opened, for example, in Microsoft Excel.

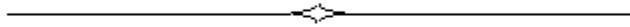
NOTE: Exporting the Login History does not clear the Login History details from the dialog.

How do I clear the Login History?

- To clear the Login History, click Clear History.

You will be asked to confirm that you want to clear the Login History log. All Login details are removed. The first Login details to appear after the Login has been cleared will be the date and time that you log out.

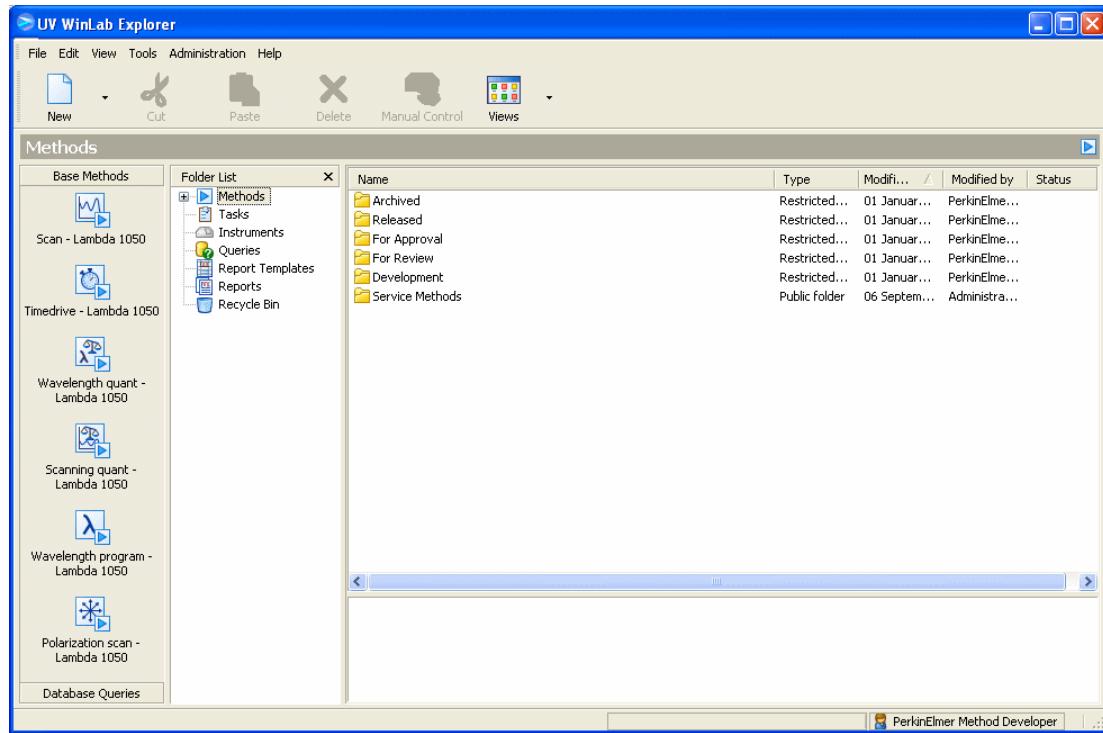
NOTE: It is only possible to clear Login History entries that have previously been exported. If the Login History contains additional entries since it was last exported, only those entries that have been exported will be deleted. If none of the entries have been exported, a warning message will be displayed when you attempt to clear the Login History.



Explorer Window

The Explorer

The Explorer window is shown below. It enables access to all major parts of the software. From the Explorer you can configure instruments, organize and launch tasks, setup report templates and view results.



What do the commands on the File menu do?

New		
	Method	Starts the New Method Wizard. This helps you configure a new method.
	Instrument	Displays the New Instrument Wizard that helps configure a new instrument.
	Query	Displays a further submenu with the options Samples, Verifications, and Calibrations. This enables you to create queries to view sample information, IPV results and calibrations created as part of Quant methods.
	Report Template	Starts Communiqué (the reporting package) with a blank report template.
	Folder	<p>Adds a new folder to the Folder List.</p> <p>NOTE: New Folder option is only available when Methods, Tasks or Queries are selected in the Folder List.</p>
	Shortcut Group	<p>Adds a new group to the shortcut bar.</p> <p>NOTE: Only the user that has added the shortcut will be able to see it when they are logged in.</p>
View		Opens the selected Method or Task for viewing only.
Edit/ Re-process		Opens the selected Method for editing. If a Task is selected, the command becomes Re-process. The processing commands in the task can be changed and the raw data re-processed and saved.
Run/Continue		Runs the selected Method to create a Task. If a Task has been selected, the Run command becomes Continue.
Empty Recycle Bin		<p>Empties the Recycle Bin. This is only available when Recycle Bin is selected on the Folder List.</p> <p>NOTE: In the Enhanced Security version of UV WinLab, only UV WinLab Administrators are able to empty the Recycle Bin.</p>
Restore		Restores the selected files in the Recycle Bin back to their original locations. This is only available when Recycle Bin is selected on the Folder List.
Import		Allows you to import Methods and Report Templates.
Export		Allows you to export Methods and Report Templates.
Delete		Deletes the currently selected item and places it in the Recycle Bin.
Rename		<p>Highlights the name of the item that is currently selected, allowing it to be edited.</p> <p>NOTE: Only folders can be renamed.</p>
Exit		Exits the Explorer.

What do the commands on the Edit menu do?

Cut	Cuts the selected item(s). Only available when one or more items are selected.
Paste	Pastes the items from the clipboard. Only available when there is something on the clipboard.

Cut and Paste are used to move items between folders.

What do the commands on the View menu do?

Go To	Select one from: Methods, Tasks, Instruments, Queries, Report Templates, Reports, Recycle Bin.
Folder List	Switches the Folder List on and off. A check mark is displayed when the Folder List is switched on.
Shortcut Bar	Switches the Shortcut Bar on and off. A check mark is displayed when the Shortcut Bar is switched on.
Toolbars	Displays a submenu listing the available toolbars from where you can switch the Toolbars on and off. A check mark is displayed when a Toolbar is switched on.
Status Bar	Switches the Status Bar on and off. A check mark is displayed when the Status Bar is switched on.
Large icons	Displays the items in the top of the Main pane as 32 x 32 icons. A bullet is displayed if this is the option selected.
Small icons	Displays the items in the top of the Main pane as 16 x 16 icons. A bullet is displayed if this is the option selected.
List	Displays the items in the top of the Main pane as a list. Only the Names are displayed. A bullet is displayed if this is the option selected.
Details	Displays the items in the top of the Main pane as a list. The Name, Type, Modified on, Modified by and Status are listed. A bullet is displayed if this is the option selected.

What do the commands on the Tools menu do?

Instrument Performance Verification		NOTE: This menu item and resulting submenu is only available for Medium performance instruments (Lambda 25, 35, 45, 20, 40, 40P, 20Bio, 40Bio).
	Create IPV Setup	Displays the IPV setup. NOTE: Create IPV Setup is grayed unless a Medium performance instrument is selected.
	Apply to Instrument	Displays a dialog to associate an IPV setup with an instrument. NOTE: Apply to Instrument is grayed unless a Medium performance instrument is selected.
	Perform now	Displays a drop-down list of all available IPV setups. Runs the selected setup on the default instrument. NOTE: Perform now is grayed unless the default instrument is selected.
Set as Default Instrument		Makes the currently selected instrument the default instrument.
Manual Control		Opens the Manual Control task and applies it to the selected or default instrument.
Instrument Event Log		The Instrument Event Log records all calibration details, and all instrument property changes (for example, changing a lamp or detector). NOTE: Instrument Event Log is only available for High performance instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, 1050 WB and 1050 NB).
Calibrate Instrument		Opens the Calibration dialog that allows you to calibrate the attached default instrument. NOTE: Calibrate Instrument is only available when an instrument has been selected, and is only available for High performance instruments.
Instrument Properties		Displays the instrument property page. This allows you to view the current configuration and set up lamps and detectors for the instrument. NOTE: Instrument Properties is only available when an instrument has been selected, and is only available for High performance instruments.

What do the commands on the Administration menu do?

Setup Users and Groups	Displays the Users, Groups and Password Control and Summary tabs of the Security Module.
Options	Displays the Options dialog.
Signature Points	Displays the Signatures tab of the Security Module.
View Audit Trail	Displays the Login History and Audit Trail tabs of the View Audit Trail dialog.
Legacy File Converter	Displays the Legacy File Converter.

NOTE: Only **Setup Users and Groups** and **Options** are available within the Standard version of UV WinLab. The tabs Groups and Summary are not available in the Standard version of UV WinLab.

NOTE: The **Options** dialog, which enables you to set up the software to always save tasks on closing, is only available in the Standard version of UV WinLab. It is not available in the ES version of UV WinLab.

What do the commands on the Help menu do?

Contents and Index	Displays the on-screen HTML Help system.
Tutorials	Displays the on-screen HTML tutorials.
PerkinElmer on the Web	Goes to the PerkinElmer home page.
About	Displays the About box.

Which menu commands are available as Toolbar buttons?

	New (displays menu – Method, Instrument, Query, Report Template, Folder, Shortcut Group)
	Edit is only available when a Method is selected.
	View Method is only available when a Method is selected.
	View Task is only available when a Task is selected.
	Re-process is only available when a Task is selected
	Continue is only available when a Task is selected.

	Run is only available when a Method is selected.
	Delete
	Cut
	Paste
	Manual Control
	Folder View

The Toolbar can be toggled on and off.

- Click the right mouse button anywhere on the menu bar and select or deselect the toolbar.

What menu items are available when I right-click in the white space in the Explorer window?

White space is wherever there is no text or graphics.

The following menu commands are available when you click the right mouse button in the white space. Not all commands are always shown; it depends on the item highlighted in the Folder List. For example, **New-Method** is only available when Methods is highlighted in the Folder List.

View		
	Large icons	Displays the items in the top of the Main pane as 32 x 32 icons. A bullet is displayed if this is the option selected.
	Small icons	Displays the items in the top of the Main pane as 16 x 16 icons. A bullet is displayed if this is the option selected.
	List	Displays the items in the top of the Main pane as a list. Only the Names are displayed. A bullet is displayed if this is the option selected.
	Details	Displays the items in the top of the Main pane as a list. The Name, Type, Modified on, Modified by and Status are listed. A bullet is displayed if this is the option selected.
Arrange by		
	Name	Arranges the Main pane by Name.
	Type	Arranges the Main pane by Type of Method / Folder.

	Modified on	Arranges the Main pane by the date the Method / Folder was modified on. This is the default.
	Modified by	Arranges the Main pane by the user who modified the Method / Folder.
	Status	Arranges the Main pane by Method status.
Paste		
New	Method	Starts the New Method Wizard. This helps you configure a new method.
	Folder	Creates a new folder and highlights the name so that it can be amended.
Properties		Displays the Methods properties (name, description, permissions). See folders for further information about folder permissions.

NOTE: If you are unable to see any of the menus or menu items listed here it is because you do not have the required permission. Permissions are defined by the UV WinLab Administrator. Please contact your UV WinLab Administrator for further details about your permissions.

Folders

Folders can be added to create a hierarchy of information. The information here is mostly concerned with the folders shown for methods and tasks.

What folders will I see?

In the Standard version of UV WinLab, all folders are public and can be seen by everyone.

In the Enhanced Security version of UV WinLab:

The groups of which you are a member determine the folders you will see within the Explorer. The groups and the members of the groups are assigned by the Database Manager. You are able to view folders that are assigned to the groups of which you are a member, and any folders that are assigned 'Public'.

For example, the Database Manager creates 2 groups – Group A and Group B. For each group, the Database Manager creates two folders which have restricted access (that is, they can only be seen by the group to which they have been assigned) – folder 1 and folder 2 for group A, and folders 3 and 4 for group B.

If you are a member of group A, you will only see folder 1 and folder 2. However, if you are a member of group A and group B, you will see all four folders.

NOTE: If you are a Database Manager you will see all the available folders listed within the Explorer, including any private folders created by other users. Any private folders will have the name of the user to which they are assigned appended to the name of the folder.

Can other people see my folders?

The Database Manager is able to see all folders. If the folders assigned to a particular group are **Restricted**, only the members of that group are able to see the folders. If folders have been assigned as **Public**, everyone is able to see them. Only you and the Database Manager are able to see any private folders that you create.

How can I tell if a folder is Public, Private or Restricted if I do not have the necessary permission?

- Select the required folder, click the right mouse button on the folder and then select **Properties**.

The folder's Properties dialog is displayed. The **Type** of folder, together with its **Location**, what the folder **Contains** (number of folders and folder items), the date it was **Created**, and the **Owner** are displayed.

How can I tell if a folder is Public, Private or Restricted if I have the necessary permission?

In the Standard version of UV WinLab:

- Select the required folder, click the right mouse button on the folder and then select **Properties**.

The folder's Properties dialog is displayed.

Within the Enhanced Security version of UV WinLab:

1. Select the required folder, click the right mouse button on the folder and then select **Properties**.

The folder's Properties dialog is displayed.

2. Select the Permissions tab.

The selected radio button shows the status of the folder; that is, whether it is public, private (and, if so, who owns it), or restricted (and, if so, which groups have access to it).

NOTE: Only a person with **Manage the Database** permission is able to see the Permissions tab.

How do I add or delete folders?

Within the Standard version of UV WinLab:

- To add a folder, select **New** from the File menu and then select **Folder** from the submenu.

OR

Click  on the toolbar and select **Folder** from the drop-down list.

OR

Click the right mouse button (right-click) on the white space in the Main pane of the Explorer window and select **New** and then select **Folder** from the submenu.

A new folder is added to the Explorer, and you can name the folder.

NOTE: You can also add subfolders. Select the folder you wish to add a subfolder to, then add a new folder as described above.

- To delete a folder, select the folder and then select **Delete** from the File menu.

OR

Click  on the toolbar.

OR

Right-click on the white space in the Main pane of the Explorer window and select **Delete**.

You will be asked to confirm that you wish to delete the folder.

NOTE: You can only delete one folder at a time. You can only delete an empty folder. If a folder contains subfolders that are empty, it cannot be deleted.

Within the Enhanced Security version of UV WinLab:

Anyone can add private folders to their Explorer. Only you and the Database Manager will be able to see your private folders.

- To add a folder, select **New** from the File menu and then select **Folder** from the submenu.
A new private folder is added to your Explorer, and you can name the folder.
- To delete a folder, select the folder and then select **Delete** from the File menu.
You will be asked to confirm that you wish to delete the folder.

If you are a Database Manager you can add folders and configure them (that is, determine who will be able to see them in their Explorer). A Database Manager is also able to re-assign a private folder.

NOTE: Folders cannot be created within the following Folder List items – Instruments, Report templates, Reports, and Recycle Bin.

How do I add a description to a folder?

NOTE: Within the Enhanced Security version of UV WinLab, only the Database Manager is able to add a description to public folders and restricted folders. Users are able to add descriptions to any private folders that they create.

1. Select the required folder, right-click on the folder and then select **Properties**.
The Folder Properties dialog is displayed.
2. Enter the required information in the Description field of the General tab and click **OK**.
The description is saved in the Properties dialog.

NOTE: It is possible to edit a description by selecting the folder, right-clicking on the folder and then selecting **Properties**.

When a folder is selected within the Explorer, any description that has been entered appears in the Display pane along with other information about the folder.

How do I rename a folder?

1. Select the required folder and then select **Rename** from the File menu.
The name of the folder is highlighted.
2. Enter the new name of the folder and then press **Enter** on the keyboard.

How can I move methods and tasks between folders?

1. To move a method or task between folders, select the method (or task) and then select **Cut** from the Edit menu.
2. Select the folder to contain the method and then select **Paste** from the Edit menu.
The method is moved to the selected folder.

NOTE: Within the Enhanced Security version of UV WinLab, you are only able to move methods (using cut and paste) if you are a member of any group that has permission to Manage the database. Only the Administrator is able to assign this option to particular groups.

NOTE: Methods and tasks can be cut and pasted between folders but they cannot be copied.

How can I import a method into a folder?

1. Highlight the required folder in Method Explorer.
2. Select **Import** from File menu.
The Open dialog is displayed.
3. Select the required method and click **Open**.
The method is imported into the selected folder.

NOTE: This is covered in more detail in Importing and Exporting Methods.

How do I configure folders?

NOTE: Within the Enhanced Security version of UV WinLab, only the Database Manager is able to configure folders – that is, determine the visibility of the folder to users – using the Permission tab within the Folder properties.

There are three levels of access: **Public**, **Private** and **Restricted**.

Public All users are able to see Public folders.

To assign a folder as **Public**:

1. Select the folder, right-click on the folder and then select **Properties**.
The Folder Properties dialog is displayed.
2. Select the Permissions tab.
3. Select **Public** and then click **OK**.
All users are able to see the Public folder .

Private A Private folder can only be seen by the person who created the folder, and a Database Manager. A Database Manager can re-assign a private folder to another user.

To re-assign a Private folder:

1. Select the folder, right-click on the folder and then select **Properties**.
The Folder Properties dialog is displayed.
2. Select the Permissions tab.
3. Select **Private**.
4. Select the new **Owner** from the drop-down list.
5. Click **OK**.

The private folder is assigned to the new owner. Only the Database Manager and the new owner are now able to see the folder.

Restricted A Restricted folder can only be seen by members of the group(s) that are given access to the folder by a Database Manager.

To assign groups to a folder:

1. Select the folder, right-click on the folder and then select **Properties**.
The Folder Properties dialog is displayed.
 2. Select the Permissions tab.
 3. Select **Restricted**.
A list of **Available Groups** is displayed.
 4. Select a group from the **Available groups** list and click **Add**.
The group is added to the **Groups with access to this folder** list.
 5. Repeat step 3 to add further groups to the **Groups with access to this folder** list, as required.
 6. Click **OK**.
 7. All groups in the **Groups with access to this folder** list will have access to the Restricted folder.
- To remove a group from the **Groups with access to this folder** list, select the group and click **Remove**.
The group is removed from the **Groups with access to this folder** list and added to the **Available Groups** list.

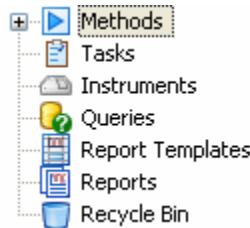
NOTE: If you are a Database Manager and wish to alter a folder description AND the visibility of a folder, enter any necessary changes on the General tab of the folder's Properties dialog and click **Apply** before selecting the Permissions tab and making the necessary changes. This ensures that the changes made on the General tab are saved when the Permissions tab is selected.

What menu items are available when I right-click on a folder?

The following menu items are available when you click the right mouse button on a folder:

Open	Opens the folder.
Cut	Cuts the folder to the clipboard.
Paste	Pastes the folder from the clipboard.
Delete	Deletes the folder to the recycle bin.
Properties	Displays the properties of the folder.

Methods



A method is a template that is used to create a task. A method can be reused any number of times. When data is collected and it is saved, a task is generated. The Methods level within the Explorer contains some default Methods folders and subfolders including Development, For Approval, Released, Archived, and Example Methods. When you Add A New Instrument, a number of base methods specific to your instrument are installed. You also have the option to install a number of Example Methods, arranged into folders by application.

Do I have to use a method to create a task?

Yes, you always have to start from a Method. A Task is a method that contains data that has been collected.

Viewing Methods in the Explorer

What is displayed in the Main pane when a method is selected in the Folder List?

When a Method type is selected, the top section of the Main pane shows the folders and items on the relevant section of the Folder List.

The Full details view (obtained by selecting **Details** from the View menu) displays the columns: Name, Description, Type, Created by, Created on, Modified by, Modified on, Revision, Method ID, Instrument type, and Status.

The available method types are: Scan, Timedrive, Scanning Quant, Wavelength Quant, Wavelength Program, and Polarization.

NOTE: Polarization is only available for High performance instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 850R, 900, 950, 1050 WB and 1050 NB).

The available status types are: locked, unlocked, approved, reviewed, draft, superceded, one off, and predefined.

Hovering the mouse over a Method name displays a tooltip that displays the Name, Type and Description of the Method.

What is displayed in the Display pane when a method is selected in the Main pane?

When a Method is selected in the Main pane, the Display pane contains the details of the method: Name, Description, Type, Created by, Created on, Modified by, Modified on, Revision, Method ID, Instrument type and Status.

This information can be copied to the clipboard using CTRL+C, but it cannot be edited.

What menu items are available when I right-click on a method in the Explorer Window?

Run	Runs the selected method.
View	Opens the currently selected method as read-only.
Edit	Edits the currently selected method.
Cut	Cuts the method to the clipboard.
Delete	Deletes the method to the recycle bin. If you are using the ES version of UV WinLab, the Delete Method security dialog will be displayed and must be completed before the method can be deleted.
Create Desktop Shortcut	Adds a shortcut icon for the currently selected method to the Desktop.

NOTE: You must have the correct permission to access these menu items. For further information about your permissions, please contact your UV WinLab Administrator.

Can I add additional folders in which to store my methods?

Yes.

- From the File menu select **New** and then select **Folder**.
A folder is added and can be renamed.

For further information about folders, please see folders.

Creating Methods

What types of methods are available?

NOTE: You cannot change the base methods installed with your instrument. You can only save it as a new method.

Some example methods are provided. There are two example method folders: Medium Performance Instruments and High Performance Instruments. The example methods are divided into subfolders by application. You can choose which methods groups are imported when you add your instrument. The methods are specific to the type of instrument (Medium or High) and they cannot be interchanged (it is not possible to run a method created for a Medium performance instrument on a High performance instrument). For further information about example methods, see:

Example Methods for Medium Performance Instruments

Example Methods for High Performance Instruments

These methods can also be used as a starting point to create your own methods. Or, you can use the Method Wizard to define a new method. See "How do I create and save a new method?" below.

How do I create and save a new method?

You can create a new method using the New Method Wizard, or you can change one of the example methods supplied and save it with a new name.

Using the New Method Wizard

The Method Wizard allows method creation without having to edit existing pre-defined methods. In addition, an instrument and accessories can be specified without needing to be attached to an instrument.

1. From the File menu select **New**, and then from the submenu select **Method**.

OR

Select Methods from the Folder List, and then right-click within the white space in the right-hand pane. Select **New** and then select **Method** from the submenu.

OR



Select Methods from the Folder List, and then click .

The New Method Wizard is displayed.

2. Select the type of Instrument from the drop-down list and then click **Next**.

The description field below the drop-down lists displays the instruments available for the selected type.

The Select Instrument page of the Wizard is then displayed.

3. Select the type of instrument from the drop-down list, and then click **Next**.

This associates a particular instrument with a method.

NOTE: Only instruments that have previously been installed are listed. See Adding an instrument for further information.

4. Select the type of method from the drop-down list.

The method types are – Scan, Timedrive, Wavelength Quant, Scanning Quant and Wavelength Program. In addition, Polarization Scan is available for high-performance instruments.

5. If you wish to copy the instrument settings from an existing method into this new method, select **Copy settings from an existing method**.

6. Click **Browse Methods**, locate the required method and then click **OK**.

7. Click **Next**.

8. Select the accessories that will be used and then click **Next**.

It is possible to select more than one accessory. Where accessories cannot be used together, it will not be possible to make the selection.

The Finish page of the wizard is displayed.

9. **Edit upon completion** is selected by default. The method opens in the workspace once it has been saved. If you wish to just create the method but not edit it at this time, deselect **Edit upon completion**.

10. Click Save/Finish.
The Save Method dialog is displayed.
11. Select the location for the new method from the list of folders.
12. The method has a default name. Click **OK** to save the method with the default name, or enter a new name and then click **OK**.
13. If you are using the Enhanced Security (ES) version of UV WinLab a security dialog will be displayed prompting you for your **User name**, **Password**, **Reason** and **Comment**. Enter the required details and then click **OK**.
The wizard closes and the method is created. If you selected Edit upon completion, the method will open within the Workspace. Otherwise, the Explorer is re-displayed.

Using an existing method to create a new method

NOTE: It is not possible to turn one type of method into another, that is a scan method cannot be turned into a wavelength program method and cannot be expanded to include Quant, so you must start by opening a method of the correct type.

1. In the Explorer, select the example method to be used as the template for the new method.
2. Double-click on the method to open it in the Task Workspace and select **View**.
3. From the File menu select **Save Settings**, and then from the submenu select **As NewMethod**.
The Save As Method dialog is displayed.
4. Select the location for the new method from the list of folders.

NOTE: The folders available in the Folder List will depend on the permissions you have.

5. Enter a **Name** and **Description** and then click **Save**.
A Save As Method dialog is displayed.
6. If you are using the Enhanced Security version of UV WinLab, enter your **User name**, **Password**, **Reason** and **Comment**.
The fields that appear on this dialog depend on the settings previously defined by your UV WinLab Administrator.
7. Click **OK**.
The dialog closes and the method is saved. The status of the method in the Explorer is draft.

NOTE: The new method (which is now open) is in View mode. If you wish to edit the settings you must close the method and then right-click on the method in the Explorer and select Edit.

How do I edit a method?

- Select the required Method, and then from the File menu select **Edit**.
The Method opens in the Workspace and can be edited.

OR

Right-click on the Method and select **Edit** from the pop-up menu.

NOTE: You must have the correct permission to edit a Method. For further information about your permissions, please contact your UV WinLab Administrator.

How do I import a method?

1. In the Explorer, select the folder that you want to import the method into.
2. From the File menu select **Import**.
The Open dialog is displayed.
3. Select the method to import and click **Open**.
Method files have .wlm extensions. The method is imported into the selected folder.

NOTE: The method to be imported must not have the same name as a method that already exists in the folder. If the method has the same name, you can rename the method you are importing or import the method into a different folder.

NOTE: If a method is developed using a Lambda 35 or Lambda 45 instrument and then exported to a PC attached to a Lambda 25 where it is imported, the slit width must be 1 nm. The Lambda 25 can only operate with a slit width of 1nm.

How do I export a method?

1. In the Explorer, select the method that you want to export.
2. From the File menu select **Export**.
The Save As dialog is displayed.
3. Select the folder to export the method to, and then click **Save**.
The method is exported to the selected folder.

NOTE: If a method is developed using a Lambda 35 or Lambda 45 instrument and then exported to a PC attached to a Lambda 25, where it is then imported, the slit width must be 1 nm. The Lambda 25 can only operate with a slit width of 1 nm. Also, if a method is exported to a PC attached to a Lambda 20, the slit width must be 1 nm or 2 nm. The Lambda 20 can only operate with a (fixed) slit width of 1 nm or 2 nm.

What are the different status types for a method?

Pre-defined: This is the status of the default methods that ship with UV WinLab 6.0. Pre-defined methods are read-only and cannot be overwritten. You have to open the method and then select **Save Settings As New Method** from the File menu to create an editable copy. In UV WinLab 6.0 the example method status is set to draft and the above procedure is not necessary.

Draft: This is the status assigned to a newly created method. The method can be changed and saved as many times as required. When the method has been finalised the method can be locked and the status changes to Locked. In UV WinLab 6.0 the example method status is draft.

Locked: A Draft or Unlocked method can be Locked. In the Locked state, the method can be Viewed and Run, but the settings cannot be Edited. Once Locked, a method can be (1) Unlocked – enabling further changes to be made. The status then becomes Unlocked. In the Enhanced Security version of UV WinLab, a new (editable) revision of the method is created. The previous revision remains unchanged and is not editable. (2) Reviewed (This is only applicable to the Enhanced Security version of UV WinLab).

Unlocked: This is the status after a Locked, Reviewed (Enhanced Security version only), or Approved (Enhanced Security version only) method is Unlocked. In the Enhanced Security version of UV WinLab, the revision number of the method is incremented. In the Standard version of UV WinLab, the original method simply becomes unlocked. In the Enhanced Security version, the Audit trail is initiated on unlocking a method.

Reviewed: This is only available in the Enhanced Security version of UV WinLab. It is the method status after it has been reviewed.

Approved: This is only available in the Enhanced Security version of UV WinLab. It is the method status after it has been approved. IPV methods (for Medium Performance Methods) have a status of Approved.

How do I lock a method?

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the correct permission to lock a method. Permissions are defined by the UV WinLab Administrator. Please contact them for further information.

A method is locked from within the Workspace.

NOTE: The report template associated with a Method should be approved before the Method is locked. If not, the Print to Database reporting option will not be available.

1. Open the method that you wish to lock.
Double-click on the name of the method in the Explorer.
2. From the Tools menu select **Lock**.
The Lock Method dialog is displayed.
3. If you are using the Enhanced Security version of UV WinLab, enter your **User name**, **Password**, **Reason** and **Comment**.
The fields that appear on this dialog depend on the settings previously defined by the UV WinLab Administrator.
4. Click **OK**.
The dialog closes and the method is locked. The fields on the Data Collection page are grayed as they can no longer be edited. The Status bar in the Workspace shows that the method is locked. The Lock command on the Tools menu toggles to Unlock. The status of the method in Explorer is now displayed as locked and the Revision number is 1. The Audit Trail is started.

How do I unlock a method?

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the correct permission to unlock a method. Permissions are defined by the UV WinLab Administrator. Please contact them for further information.

A method is unlocked from within the Workspace.

1. Open the method that you wish to unlock.
Double-click on the name of the method in the Explorer.
2. From the Tools menu select **Unlock**.
The following message is displayed: The Method is locked. Samples may be run, but modification of method parameters will not be possible unless it is first unlocked.
3. Click **OK**.
The message is removed and the method is opened in the Workspace.
4. From the Tools menu select **Unlock**.
The Unlock Method dialog is displayed.
5. Enter your User name, Password, Reason and Comment.
The fields that appear on this dialog depend on the settings previously defined by the UV WinLab Administrator.
6. Click **OK**.
The dialog closes. The fields on the Data Collection and Accessory pages can now be edited. The status bar in the Workspace shows that the method is unlocked. The **Unlock** command on the Tools menu toggles to **Lock**. The status of the original method is still locked (and retains any reviewed and/or approved status) and the name is appended with -Revision 1. A new unlocked copy of the method is created with the same name but appended with -Revision 2.

NOTE: Once Revision 1 has been unlocked once it is not possible to unlock it again. The following message is displayed if you try to unlock it again: 'This method has already been unlocked and a later revision created. Use Save Settings As New Method to create a new editable method'.

Can I run a locked method on a different instrument or the same instrument with a different accessory?

No, once a method has been locked it must use the instrument and accessory for which it has been set-up. If you try to use a different configuration, you will get an error message and the method will not run. However, if the method is not locked, the software will detect a change to the instrument and/or accessory and update the settings accordingly.

Reviewing and approving Methods

What is meant by reviewing and approving?

The ability to formally review and approve methods, tasks, IPV setups and sample results is a function of the Enhanced Security version of UV WinLab.

It is up to the UV WinLab Administrator to set the correct privileges to ensure that only the appropriate people can 'sign off' data.

We use the term 'Review' to mean that the person has looked at the data and has agreed that it is correct. This is along the lines of a peer review and any number of people can review data as determined by your internal procedures.

We use the term 'Approve' to mean that a person with 'authority' has signed off the data as fit-for-purpose and again details of who is allowed to do this should be documented in your internal procedures. A Method can only be approved once.

How do I review a method?

A method is reviewed from within the Workspace. A method must be locked before it can be reviewed.

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the correct permission to review a method. Permissions are defined by the UV WinLab Administrator. Please contact them for further information. If you are using the Standard version of UV WinLab, reviewing methods is not available.

1. Open the method that you wish to review.
Right-click on the name of the method in the Explorer and select **View** from the menu.
2. From the Tools menu select **Review** and from the submenu select **Method**.
The Review Method dialog is displayed.
3. Enter your User name, Password, Reason and Comment.
The fields that appear on this dialog depend on the settings previously defined by the UV WinLab Administrator.
4. Click **OK**.
The dialog closes and the method status updates to Reviewed.
The method event log is updated. To view the method event log, select **Event Log** and then **Method** from the Tools menu in the Workspace.

How do I approve a method?

A method is approved from within the Workspace. A method must be locked before it can be approved. A method does not have to be reviewed before it is approved.

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the correct permission to approve a method. Permissions are defined by the UV WinLab Administrator. Please contact them for further information. If you are using the Standard version of UV WinLab, approving methods is not available.

1. Open the method that you wish to approve.
Right-click on the name of the method in the Explorer and select **View**.
2. From the Tools menu select **Approve** and from the submenu select **Method**.
The Approve Method dialog is displayed.
3. Enter your User name, Password, Reason and Comment.
The fields that appear on this dialog depend on the settings previously defined by the UV WinLab Administrator.

4. Click **OK**.

The dialog closes. The status of the method updates to Approved.

The method event log is updated.

Viewing, editing and running methods

How do I run a method?

- Select the required Method, and then from the File menu select **Run**.
The Method opens in the Workspace and can be edited.

OR

Right-click on the Method and select **Run** from the pop-up menu.

OR

Select the Method and then press ENTER.

OR

Click  to run the Method.

What is the difference between viewing, editing and running a method?

Viewing

If you select to view a method, all the method settings are grayed and cannot be altered. You can select **Save Settings As New Method** from the File menu and save the method with a new name, thereby creating a new method. To use this new method, you will need to close the current method (with the grayed settings) and then open the new method from the Explorer.

If you close the Workspace when you are viewing a method, you are not prompted to save the method as it is not possible to make any changes.

NOTE: It is not possible to select **Run** (to collect data) when you are viewing a method.

Editing

If you select to edit a method, you can make changes to the method and save these changes. When you close the Workspace, a prompt will ask whether you wish to save the method. This prompt will appear regardless of whether you have made any changes. If you have not run the method (that is, not collected any data), the Save Results To Task and Save Results As New Task commands on the File menu will remain grayed as they are not applicable.

It is possible to run the method (to collect data and so create a task) when you are editing a method. When you close the Workspace, a prompt will appear asking if you wish to save the method and the task respectively. If you choose not to save the method, any changes you made since it was last saved are lost. If you are using the Standard version of UV WinLab and you choose not to save the task, all the data will be lost. If you are using the Enhanced Security version of UV WinLab, you must save the task before exiting the Workspace. The Save Settings To Method, Save Settings As New Method, Save Results To Task, and Save Results As New Task commands on the File menu are all available.

Running

If you have run the method and collected data (and so created a task), when you close the Workspace, a prompt will appear asking if you wish to save the task. If you do not save the method, any changes you made since it was last saved are lost. If you are using the Standard version of UV WinLab and you choose not to save the task, all the data will be lost. If you are using the Enhanced Security version of UV WinLab, you must save the task before exiting the Workspace. The Save Settings As New Method, Save Results To Task, and Save Results As New Task commands on the File menu are all available.

If you select to run a method, you can make changes to the method and save these changes using the Save Settings command. When you close the Workspace, a prompt will ask whether you wish to save the task. If you have not run the method (that is, not collected any data), the Save Results To Task and Save Results As New Task commands on the File menu will remain grayed as they are not applicable.

How do I save baselines with a method and how does this affect the corrections collected when I run a task?

NOTE: You must open a method in Edit mode.

1. Define the corrections you require.

2. Click  .
Clicking Autozero will perform the selected corrections.

3. From the File menu select **Save Settings Method**.

4. On the Save Method dialog, select **Save Corrections**.
The corrections will be stored with the method.

NOTE: You should not save corrections with the method if you select **Always at task start** or **Always before next measurement**, as by definition, previous corrections are discarded.

When you run the task from this method:

If **As required at task start** is selected, the corrections that were saved with the method are used and only any additional corrections that are required are collected at the start of the task provided the expiry time has not elapsed.

If **As required before next measurement** is selected, the corrections that were saved with the method are used and only any additional corrections that are required are collected before each measurement.

NOTE: If you press  when you are in a task that contains corrections that were saved as part of the method, these previously saved corrections are discarded.

Can I setup the sample table and save this as part of my method?

Yes, you can specify the sample table including number of samples, replicates/measurements, and columns and will be saved when the method is saved. This will be saved when the method is saved.

Example Methods for Medium Performance Instruments

The following example methods are provided for Medium performance instruments. They are divided into groups based on their application. You can select which group(s) of methods to install when adding a new instrument. See below for further information about a particular method:

Biochemistry & Molecular Biology

- | | |
|---|--|
|  Bradford Protein Assay |  Nucleic Acid Concentration |
|  Lowry Protein Concentration |  Warburg-Christian Nucleic Acid and Protein Assay |
|  Protein Scan | |

Clinical & Health Care

- | | |
|--|---|
|  Bilirubin in CSF (UK Method) |  Cytochrome P450 Determination |
|  G6PD Kinetics |  Porphyrin Scan |

Environmental

- | | |
|---|---|
|  Chlorophyll determination (Trichromatic Method) |  Chlorophyll (US 10200H) |
|  Nitrate |  Phosphate in Water |

Food & Drink

- | | |
|--|--|
|  ASBC Color and Turbidity in Beer |  EU Olive Oil Determination |
|  Sulfite in Food by Enzymatic Determination |  Wine Analysis |

General QC

- | | |
|---|--|
|  Lambda Max |  Naphthalenes (ASTM D1840-03) |
|  Percentage Strength |  Scan and Second Derivative |

To run an example method:

1. Open the **Example Methods** folder and then open the **Medium Performance Instruments** methods folder.
2. Open the appropriate method subfolder and highlight the required method.
3. Select **Run** from the File menu.
The method opens in Workspace.

ASBC Color and Turbidity in Beer

This method is a Beer color and hue method produced by the American Society of Brewing Chemists (ASBC). The ASBC scale is used in the measurement of the color of beer, malt worts, caramel and similar solutions. This scale should be measured with a cell pathlength of 10 mm. The example method is based on a Wavelength Program.

To run the method:

1. Open the **Food & Drink** folder.
2. Highlight the **ASBC Color and Turbidity in Beer** method and then select **Run** from the File menu.

The method opens in Workspace.

3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.

4. Ensure the following parameters are entered:

Wavelengths 420 nm, 440 nm, 700 nm.

Ordinate Mode %T

5. Select the Sample Info page.
6. Enter the number of **Samples**.

7. Select the Processing page.

Equations are defined within the Processing page.

Select an Equation and then click **Settings** to view the Equation.

8. Select the Results page.

The Results Table is defined to display the following values for each sample: **T 420nm**, **T 440nm**, **T 700nm**, **Color** and **Turbidity**.

9. Select the Output page.

The default report template is **ASBC Color and Turbidity** and this is set to print **On demand**.

10. If required, alter the report frequency or output.

11. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the tree and click **Preview**.
The report is displayed.

Bilirubin in CSF (UK Method)

This method is designed to determine the levels of bilirubin in cerebrospinal fluid (CSF), an important diagnostic tool for those with suspected subcranial haemorrhage.

Oxyhaemoglobin has an absorbance maximum between 410 and 418 nm. Bilirubin has a broad peak in the range 450 to 460 nm, or a shoulder adjacent to an oxyhaemoglobin peak, if this is present.

This method is based on a Scan method. A scan is collected for the wavelength range of interest. A baseline is determined, and the absorbance above this baseline at 476 nm (the net bilirubin absorbance, NBA) is calculated. In addition, the absorbance of any oxyhaemoglobin peak above this baseline (the net oxyhaemoglobin absorbance, NOA) at nm is also determined.

To run the method:

1. Open the **Clinical & Health Care** folder.
2. Highlight the **Bilirubin in CSF (UK Method)** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan	600–350 nm
------	------------

Ordinate Mode	A
---------------	---

Slit Width	1 nm
------------	------

Data Interval	0.1 nm
---------------	--------

Number of cycles	1
------------------	---

Cycle time	1 second
------------	----------

5. Select the Sample info page.

Patient ID, **Lo Baseline Point** and **Hi Baseline Point** columns are added to the table by default.

6. Enter the number of **Samples**.

7. Select the Processing page.

Equations are defined within the Processing page.

(Select an Equation and then click **Settings** to view the Equation.)

8. Select the Results page.

The Results Table is defined to display the following values for each sample: **Patient ID**, **Lo Baseline Point**, **Hi Baseline Point**, **NBA**, **Abs Low Base**, **Abs Hi Base**, **Abs at Peak**, **OHb Abs**, **OHb Peak** and **NOA**.

9. Select the Output page.

The default report template is **Bilirubin**.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Bradford Protein Assay

This example method is for the determination of protein concentration following the Bradford method. It is based on a Wavelength Quant method.

The determination is based on the fact that the dye Coomassie Brilliant Blue G-250 exists in two different color forms, red and blue. It is converted from the red form to the blue form on binding with a protein, with a corresponding shift in the absorption maximum from 465 nm to 590 nm. In this method the absorption at 590 nm is measured. Four calibration standards are run.

Reference: Bradford M. M. (1976), *Anal Biochem.*, 72, pp. 248–254.

To run the method:

1. Open the **Biochemistry & Molecular Biology** folder.
2. Highlight the **Bradford Protein Assay** method and then select **Run** from the File menu.

The method opens in the Workspace.

3. Select **Data Collection** in the Folder List.

The Data Collection page is displayed.

4. Ensure that the following parameters are entered:

Wavelength 590 nm

Ordinate Mode A

Number of cycles 1

Cycle time 1 second

5. Select the Sample info page.

The sample table is configured to have four Standards and a sample.

6. Enter the number of **Samples**.

7. Select the Beer's Law Quant page.

8. Ensure the following parameters are entered:

Component Protein

Units $\mu\text{g mL}^{-1}$

Calibration curve Cubic

Correlation ⁽²⁾ 0.980000

Force recalibration On start-up

Allow 10% extrapolation

Proceed on error

9. Select the Parameters page and ensure that the **Baseline Correction** is set to **None**.

10. Ensure that the **Wavelength** is set to 590 nm.

11. Select the Results page.

The Results Table is defined to display the following values for each sample: **Protein** ($\mu\text{g ml}^{-1}$), **Ordinate (A)**.

12. Select the Output page.

The default report template is **Simple Quant Report** and this is set to print **On demand**.

13. If required, alter the report frequency or output.

14. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.
The report is displayed.

Chlorophyll a, b and c (Trichromatic) Determination

This method is used for the measurement of chlorophyll using the Trichromatic method (SCOR-UNESCO).

This is a simple multi-component analysis based on measurement at three different wavelengths. Coefficients are then applied to calculate the three chlorophyll types. The extinction coefficients used are from: Jeffery S.W., Humphrey G.F., (1975) "New spectrophotometric equation for determining chlorophyll a, b, c1 and c2," *Biochem. Physiol. Pflanz.*, 167, pp. 194–204

$$C_a [\text{mg m}^{-3}] = (11.85 D_{663-665} - 1.54 D_{647} - 0.08 D_{630}) v \text{l}^{-1} \text{V}^{-1}$$

$$C_b [\text{mg m}^{-3}] = (-5.43 D_{663-665} + 21.03 D_{647} - 2.66 D_{630}) v \text{l}^{-1} \text{V}^{-1}$$

$$C_c [\text{mg m}^{-3}] = (-1.67 D_{663-665} - 7.6 D_{647} + 24.52 D_{630}) v \text{l}^{-1} \text{V}^{-1}$$

Data is collected as a scan for speed and diagnostic purposes but it is reported as an expanding table.

To run the method:

1. Open the **Environmental** folder.
2. Highlight the **Chlorophyll (Trichromatic) method** and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 700–600 nm

Ordinate Mode A

Data interval 1 nm

5. Select the Sample info page.
Extract Volume (ml), **Filtered Water Vol (litres)** and **Pathlength (cm)** columns are added to the table by default.
6. Enter the number of **Samples**.
7. Enter the **Extract Volume (ml)**, **Filtered Water Vol (litres)** and **Pathlength (cm)** values for each sample.

8. Select the Processing page.

Equations are defined within the Processing page.

(Select an Equation and then click **Settings** to view the Equation.)

9. Select the Results page.

The Results Table is defined to display the following values for each sample: **Extract Volume (ml)**, **Filtered Water Vol (litres)**, **Abs 630nm**, **Abs 647nm**, **Abs 664nm**, **chl a (mg m⁻³)**, **chl b (mg m⁻³)**, **chl c (mg m⁻³)** and **Pathlength (cm)**.

10. Select the Output page.

The default report template is **Chlorophyll SCOR** and this is set to print **On demand**.

11. If required, alter the report frequency or output.

12. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.

The report is displayed.

Chlorophyll Determination (US10200H)

This method is used for the measurement of chlorophyll using the US10200H method.

To run the method:

1. Open the **Environmental** folder.
2. Highlight the **Chlorophyll determination (US 10200H)** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelengths 700 nm, 665 nm, 664 nm, 647 nm, 630 nm, 480 nm.

Ordinate Mode A

Number of cycles 1 nm

Cycle time 1.5 minutes

5. Select the Sample info page.
Volume Extracted and **Volume Filtered** columns are added to the table by default.
Each Sample row in the Sample Table has two Measurements associated with it.
6. Enter the number of **Samples**.
7. Enter the **Volume Extracted** and **Volume Filtered** for each sample.
8. Select the Processing page.
Equations are defined within the Processing page.
(Select an Equation and then click **Settings** to view the Equation.)
9. Select the Results page.
10. Ensure that the Results Table is defined to display the following values for each sample:
Volume Extracted and **Volume Filtered**.
11. On the Output page, ensure that the report template is **Chlorophyll US Method 10200H** and that this is set to print **On demand**.
12. If required, alter the report frequency or output.
13. Click  to run the Method.

14. When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.
 - To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.
The report is displayed.

Cytochrome P450 Determination

Cytochrome p450 is an important family of mixed-function oxygenases that are responsible for the metabolism of compounds such as steroids and fatty acids, and also xenobiotics such as aromatic hydrocarbons, therapeutic drugs and abused drugs.

Measurement of cytochrome activity is important in biochemistry, cytotoxicology and pharmacological studies.

This method can be used for determining cytochrome p450 in tissue samples. The measurement is based on the difference between the absorption maxima (at 450 nm) of the reduced and native forms of cytochrome p450.

Scatter correction at 490 nm is carried out to correct for stray light. Tissue preparations are often quite turbid, so the scatter correction will eliminate the high background values produced by turbid samples.

The reference wavelength used is 490 nm, since this is an isobestic point (common to both forms), effectively eliminating the effect of turbidity scattering.

The extinction coefficient is assumed to be 91 mM cm⁻¹.

The reference for the method is Omura, T. and Sato, R., (1964) *J. Biol. Chem.*, 239, pp. 2370–2378

To run the method:

1. Open the **Clinical & Health Care** folder.
2. Highlight the **Cytochrome P450 Determination method** and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 500–400 nm

Ordinate Mode A

Data interval 0.5 nm

5. Select the Sample info page.
Dil factor and **Resuspension factor** columns are added to the table by default.
6. Enter the number of **Samples**.
7. Enter the **Resuspension factor** and **Dilution factor** for each sample.

8. Select the Processing page.

Equations are defined within Processing page.

(Select an Equation and then click **Settings** to view the Equation.)

9. Select the Results page.

The Results Table is defined to display the following values for each sample: **Max Abs**, **Lambda Max**, **Abs 490 nm**, **Difference**, **p450 Conc (nm/mg)**.

10. Select the Output page.

The default report template is **Cytochrome P450 Determination** and this is set to print **On demand**.

11. If required, alter the report frequency or output.

12. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.

The report is displayed.

G6PD Kinetics

This method determines the activity of the enzyme Glucose-6-phosphate Dehydrogenase (G6PD). The rate is measured at 340 nm (NAD/NADH⁺ type of kinetics). This is a Timedrive method with a duration of 5 minutes. The default interval is 30 s.

This method can be automated to use with a cell changers (6, 8 or 9 positions).

To run the method:

1. Open the **Clinical & Health Care** folder.
2. Highlight the **G6PD Kinetics** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelength	340 nm
Ordinate Mode	A
Time Interval	30 s
Total time	300 s

5. Select the Sample info page.
The Sample Table has the column **Hb (g dL)** added as default.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within Processing page.

(Select an Equation and then click **Settings** to view the Equation.)

8. Select the Results page.
The Results Table is defined to display the following value for each sample: **Rate**, **Correlation Coefficient**, **G6PD (IUg Hb)**.
9. Select the Output page.
The default report template is **G6PD Kinetics** and this is set to print **On demand**.
10. If required, alter the report frequency or output.

11. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

EU Olive Oil Determination

This is an EU method to determine the purity of olive oil in accordance with CAC/RM 26-1970. The extinction is measured at two wavelengths (232 and 270 nm) and the shape of the baseline is assessed around 270 nm to calculate ΔE .

Reference: Codex standard for olive oil, virgin and refined, and for refined olive-pomace oil. Codex Stan 33-1981 (Rev 1-1989)

The ordinate values at 232 nm (Equation 1), 270 nm (Equation 2) are calculated individually. These individual values are then multiplied by concentration to give Equations 3 and 4. A peak is then searched for over specified range 268–272 nm.

To run the method:

1. Open the **Food & Drink** folder.
2. Highlight the **EU Olive Oil Determination** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range	300–220 nm
Ordinate Mode	A
Data interval	0.5 nm
Number of cycles	1
Cycle time	1 second

5. Select the Sample info page.
Conc (g/100 mL) and **Cell** columns are added to the table by default.
6. Enter the number of **Samples**.
7. Enter the **Conc (g/100 mL)** and **Cell** information for each sample.
8. Select the Processing page.
Equations are defined within the Processing page.

(Select an equation and then click **Settings** to view the equation)

9. Select the Results page.

The Results Table is defined to display the following values for each sample: **Conc (g/100mL), A232, A270, E232, E270, Max E, Lambda Max, Peak -4, Peak +4, Delta E.**

10. Select the Output page.

The default report template is **Olive Oil** and this is set to print **On demand**.

11. If required, alter the report frequency or output.

12. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.

The report is displayed.

Lambda Max

This method can be used to find the maximum absorbance and wavelength over a scan range 700–350 nm.

NOTE: If a different wavelength is required, it will be necessary to modify the data collection settings and both equations to reflect this.

To run the method:

1. Open the **General QC** folder.
2. Highlight the **Lambda Max** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure that the following parameters are entered:

Scan range 700–350 nm

Scan speed 960 nm/min

Ordinate Mode A

Slit width 1 nm

Data interval 1 nm

Number of cycles 1

Cycle time 1 second

5. Select the Sample info page.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within the Processing page.
(Select an Equation and then click **Settings** to view the Equation.)
8. Select the Results page.
The Results Table is defined to display the following value for each sample:
Absorbance and Wavelength at Maxima.

9. Select the Output page.

The default report template is **Lambda Max** and this is set to print **On demand**.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Lowry Protein Concentration

This example method is for Protein concentration determination following the Lowry method. It uses a Scanning quant method. The Scan range is 400–900 nm and 6 standards are run in duplicate. The peak at 750 nm is used for the calibration of protein concentration up to concentrations of 40 mg/ml. Above this, change the calibration to single point at 500 nm.

To run the method:

1. Open the **Biochemistry & Molecular Biology** folder.
2. Highlight the **Lowry Protein Concentration method** and then select **Run** from the File menu.

The method opens in Workspace.

3. Select **Data Collection** in the Folder List

The Data Collection page is displayed.

4. Ensure the following parameters are entered:

Scan range 900–400 nm

Scan speed 1920 nm/min

Ordinate Mode A

Slit width 1 nm

Data interval 1 nm

Number of cycles 1

Cycle time 1 second

5. Select the Sample info page.

6. Enter the number of **Samples**.

7. Select the Beer's Law Quant page.

8. Ensure the following parameters are entered:

Component Protein

Units mg/ml

Calibration curve linear

Correlation (²) 0.980000

9. Select the Parameters page and ensure the **Ordinate Mode** is set to **Height**.

10. In the Settings, the **Wavelength** is set at 750 nm.

11. Select the Results page.

The Results Table is defined to display the following values for each sample: **[Protein] (mg/ml), Residual, Ordinate (A)**.

The default report template is **Lowry** and this is set to print on demand.

12. If required, alter the report frequency or output.

13. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

➤ To preview the report that will be generated, select Output on the Folder List and click **Preview**.

The report is displayed.

Naphthalene (ASTM D1840-03)

This example method is for naphthalene concentration determination following the ASTM D1840-03. It uses a Wavelength Program method.

To run the method:

1. Open the **General QC** folder.
2. Highlight the **Naphthalene (ASTM D1840-03)** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelength	285 nm
Ordinate Mode	A
Slit Width	1 nm
Data Interval	1 nm
Number of cycles	1
Cycle time	1 second

5. Select the Sample info page.
The columns **Absorptivity (L/g-cm)**, **Equiv Vol of Solvent K**, **Sample Weight (g)**, **Rel Density of Fuel** and **Rel Density of Naphthalenes** are added to the table by default.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within Processing page.
(Select an Equation and then click **Settings** to view the Equation.)
8. Select the Results page.
The Results Table is defined to display the following value for each sample:
Naphthalenes by Mass (%) (%) and **Percent Naphthalene by Volume (%) (%)**.
9. Select the Output page.
The default report template is **Naphthalenes** and this is set to print **On demand**.
10. If required, alter the report frequency or output.
11. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Nitrate Determination (EPA 352.1/US Test Method 9200)

This method is used for the measurement of nitrate at between 0.1 mg/L and 10 mg/L nitrate. Brucine sulfate in 13N H₂SO₄ combines with nitrate ion at 100 °C to produce a yellow complex which is measured at 410 nm. The example is based on a Wavelength Quant method.

To run the method:

1. Open the **Environmental** folder.
2. Highlight the **Nitrate (EPA 352.1/US Test Method 9200)** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelength 410 nm

Ordinate Mode A

Data Interval 1 nm

Number of cycles 1

Cycle time 1 second

5. Select the Sample info page.
6. Enter the number of **Samples**.
For this example, add 5 samples.
7. Select the Beer's Law Quant page.
8. Ensure the following parameters are entered:

Component Nitrate

Units mg L⁻¹

Calibration curve linear

Correlation (²) 0.980000

Force recalibration On start-up

Allow 10% extrapolation

Proceed on error

9. Select the Parameters page and ensure that the **Baseline Correction** is set to **None** and the **Wavelength** is set to 410.00 nm.
10. Select the Calibration page and ensure that the table is defined to display the following values for each Standard: **Concentration**, **Ordinate (A)** and **Residual**.
11. Make sure there are 5 Standards, with concentration values of 0.000, 0.1250, 1.0000, 1.5000 and 2.0000.
12. Select the Results page.
The Results Table is defined to display the following values for each sample: **Nitrate (mg L⁻¹)** and **Ordinate (A)**.
13. Select the Output page.
The default report template is **Simple Quant Report** and this is set to print **On demand**.
14. If required, alter the report frequency or output.

15. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

➤ To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Nucleic Acid Concentration and Purity

This Nucleic Acid Quantification and Purity method is a Wavelength Program method based on a 260/280 ratio, with scatter correction at 320 nm. The result will 'PASS' if the purity value is between 1.7 and 2.1. The method also calculates the concentration according to the type of nucleic acid being used. This example method uses the sample tag feature of UV WinLab.

To run the method:

1. Open the **Biochemistry & Molecular Methods** folder.
2. Highlight the **Nucleic Acid Concentration and Purity** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelengths 260 nm, 280 nm, 320 nm

Ordinate Mode A

Slit Width 1 nm

5. Select the Sample info page.

A Sample Tag column is included in the Sample Table.

A sample tag is a special custom column that can be added to assign an 'alias' to a sample. For instance if you have a group of samples that each have their own particular identifier that you need to use as the sample name, but you also want to pick out samples as belonging to special groups, you can use a sample tag.

In this method, three sample tags have been defined – dsDNA, ssDNA, and RNA.

6. Enter the number of **Samples**.

7. For each sample, select the required sample tag – dsDNA, ssDNA or RNA.

Click in the Sample Tag field to display a drop-down arrow . Click on the arrow to display a drop-down list of dsDNA, ssDNA and RNA.

8. Select the Processing page.

Equations are defined within the Processing page and should be as shown below:

(Select an Equation and then click **Settings** to view the Equation.)

Equation Purity Corrected Ration Calculation (the 7th Equation) is used to determine whether the purity of the sample is acceptable.

$purity = ((A_{260}-A_{320}) / (A_{280}-A_{320}))$. The purity result is accepted if it lies between 1.7 and 2.1.

Conditional Formatting is used on the Equation as shown in the Settings dialog.

If the result lies between 1.7 and 2.1, the Ratio column (Equation 4) in the Results table will contain the word **PURE** in addition to the value. If the result is any other value, the Ratio column will contain the word **IMPURE** in addition to the value.

The Results Table is defined to display the following values for each sample – Abs 260, Abs 280, Abs 320, Ratio, Corr 260, dsDNA Conc, ssDNA Conc, and RNA Conc.

The default report template is **Nucleic Acid Quantification and Purity** and this is set to print on demand.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.

The report is displayed.

Percentage Strength at 500 nm

The Percentage Strength method compares the absorbance at 500 nm with a control standard (which is set to 100%). The raw absorbance and percentage strength are then calculated. The example method is based on a Wavelength Program method.

To run the method:

1. Open the **General QC** folder.
2. Highlight the **Percentage Strength at 500 nm** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelength 500 nm

Ordinate Mode A

Response 2 s

Slit width 1

Number of cycles 1

Cycle time 1 second

5. Select the Sample info page.
By default there is one **Control** sample and one **Sample**.
6. Select the Processing page.
Equations are defined within Processing page.
(Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following values for each sample – **Absorbance** and **Percent Strength**.

The default report template is **Percentage Strength at 500nm** and this is set to print **On demand**.

7. If required, alter the report frequency or output.

8. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.

Phosphate in Water (EPA 365.2 & 365.3/DIN 38 405)

This example method is for determining the concentration of phosphate compound in water. It measures phosphate in the form of orthophosphate and total phosphate after digestion using a Wavelength Quant method. This analysis is suitable for determining orthophosphate concentrations at 0101 mg/ml to 5 mg/ml PO_4^{3-} .

Orthophosphate ions in acid solution combine with molybdate ions in the presence of antimony ions to produce a complex that is reduced by ascorbic acid to phosphor molybdenum blue.

Environmental Monitoring and Support Laboratory (1983), "Method 365.2 Phosphate", *Methods for Chemical Analysis of Water and Water*, Environmental Monitoring and Support Laboratory (EMSL), Cincinnati OH, USA.

DIN 38 405 Teil 1

To run the method:

1. Open the **Environmental** folder.
2. Highlight the **Phosphate in Water** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelength 710 nm

Ordinate Mode A

Slit Width 1 nm

Data Interval 1 nm

Number of cycles 1

Cycle time 1 second

5. Select the Sample info page.
The column **Type** is added to the table as default.
6. Enter the number of **Samples**.
7. Select the Beer's Law Quant page.

8. Ensure that the following parameters are entered:

Component	Phosphate
Units	mg L^{-1}
Calibration curve	linear
Correlation (²)	0.980000
Force recalibration	On start-up
Allow 10% extrapolation	
Proceed on error	

9. Select the Parameters page and ensure that the **Baseline Correction** is set to **None** and the **Wavelength** is set to 710 nm.
10. Select the Calibration page and ensure that the table is defined to display the following values for each Standard: **Concentration**, **Ordinate (A)** and **Residual**.
11. Select the Results page.
The Results Table is defined to display the following values for each sample:
Phosphate (mg L⁻¹) and **Ordinate (A)**.
12. Select the Output page.
The default report template is **Phosphate Quant** and this is set to print **On demand**.
13. If required, alter the report frequency or output.

14. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Porphyrin Scan

It uses a Scan method.

To run the method:

1. Open the **Clinical & Health Care** folder.
2. Highlight the **Porphyrin Scan** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range	700–350 nm
Scan speed	120 nm/min
Ordinate Mode	A
Slit width	1 nm
Data interval	0.5 nm
Cycle time	1 second

5. Select the Sample info page.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within Processing page.

(Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following values for each sample: **Patient ID**, **Porphyrin (nmol /L)**, **Lo Baseline Point**, **Hi Baseline Point**, **Peak Height**, **Abs 405 nm**, **Abs RH Base**, **Abs LH Base**, and **Conc Factor**.

8. The template is **Porphyrin Scan** method.

9. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Sulfite in Food by Enzymatic Determination

This is a Timedrive method used for the determination of the sulfite (sulfurous acid) in food.

Sulfite is oxidized to sulfate by sulfite oxidase in the presence of oxygen. The hydrogen peroxide (H_2O_2) formed in this reaction is reduced by NADH-peroxidase (NADH-POD) in the presence of reduced nicotinamide-adenine dinucleotide (NADH). The NADH is oxidized. The amount of NADH oxidized is equivalent to the amount of sulfite. NADH absorbs at 334, 340 and 365 nm.

To run the method:

1. Open the **Food & Drink** folder.
2. Highlight the **Sulfite Determination** method and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelength	340 nm
Ordinate Mode	A
Slit Width	1 nm
Time Interval	1 s
Total time	45 minutes

5. Select the Sample info page.
Time Point A1 (minutes), and **Time Point A2 (minutes)** columns are added to the table by default.
6. Enter the number of **Samples**.
7. Enter the values for **Time Point A1** and **Time Point A2** for each sample.
8. Select the Processing page.
Equations are defined within Processing page.
(Select an Equation and then click **Settings** to view the Equation.)
9. Select the Results page.
The Results Table is defined to display the following values for each sample: **Time Point A1 (minutes)**, **A1**, **Time Point A2 (minutes)**, **A2**, **Delta A**, **Sulfite Conc (g/L)**.

10. Select the Output page.

The default report template is **Sulfite Determination** and this is set to print **On demand**.

11. If required, alter the report frequency or output.

12. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Warburg–Christian Assay for Protein and Nucleic Acid Concentration

This example method is for the determination of protein concentration following the method described by Warburg and Christian. It is based on a Scan method.

This method is used to eliminate the interference, by calculation, of nucleic acid absorbance in the determination of protein concentration. Proteins exhibit a distinct UV absorption band with a maximum at around 280 nm. Nucleic acids also have a strong UV absorbance at 280 nm, but absorb more strongly at 260 nm. The reverse is true of proteins. Therefore a two-component mixture can be quantified using two linear equations:

$$C_{\text{Protein}} [\text{mg/ml}] = 1.55 * A_{280} - 0.757 * A_{260}$$

$$C_{\text{Nucleic Acid}} [\mu\text{g/ml}] = 62.9 * A_{260} - 36.0 * A_{280}$$

This is based, however, on the application of generic molar absorption coefficients for proteins and for nucleic acids, where in practice, the individual coefficient is often unknown.

Warburg O, and Christian W. (1941), *Biochem Z*, 310, p. 384.

To run the method:

1. Open the **Biochemistry & Molecular Biology** methods folder.
2. Highlight the **Warburg–Christian Assay for Protein and Nucleic Acid Concentration method** and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure that the following parameters are entered:

Scan range	330–230 nm
Scan speed	240 nm/min
Ordinate Mode	A
Slit width	1 nm
Data interval	1 nm
Number of cycles	1
Cycle time	1 second

5. Select the Sample info page.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within the Processing page.
Select an Equation and then click **Settings** to view the Equation.

8. Select the Results page.

The Results Table is defined to display the following values for each sample – **Abs 260nm**, **Abs 280nm**, **Protein Conc (mg/ml)**, and **Nucleic Acid (µg/ml)**.

9. Select the Output page.

The default report template is **Warburg Christian Assay as list** and this is set to print on demand.

10. If required, alter the report frequency or output.



11. Click to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.

The report is displayed.

Wine Color Analysis

This is a combination of two methods to measure Wine Color Intensity using Glories 1984 method, and Wine Color Hue using Sudraud 1958 method.

Color reference: Glories Y. (1984), "La couleur des vins. 2e partie," *Connaissance Vigne et Vins*, 18, pp. 253–271.

Hue reference: Sudraud P. (1958), "Interpretation des courbes d'absorption des vins rouges," *An. Technol. Agric.*, 7, pp. 203–208.

Samples of undiluted wine should be run in a 1 mm pathlength glass or quartz cell. The method will adjust the absorbance to 10 mm.

To run the method:

1. Open the **Food & Drink** folder.
2. Highlight the **Wine Analysis** and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 780–380 nm

Scan speed 960 nm/min

Ordinate Mode A

Slit width 1 nm

Data interval 1 nm

Number of Cycles 1

Cycle time 1 second

5. Select the Sample info page.
The column **Pathlength (mm)** is added to the table by default.
 6. Enter the number of **Samples**.
 7. Enter the **Pathlength** for each sample.
 8. Select the Processing page.
Equations are defined within the Processing page.
- (Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following values for each sample – **A420 nm**, **A520 nm**, **A620 nm**, **Color Intensity**, **Wine Hue**, **Tristim X**, **Tristim Y**, **Tristim Z**, **xvalue**, **yvalue**, **L* a* b***, **Cab***, **S**, **Q**, **hab***, **hab*1** and **hab*2**.

The default report template is **Wine Analysis Report** and this is set to print **On demand**.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Scan and Second Derivative

This method is used to collect a scan and superimpose the calculated second derivative spectrum. The second derivative spectrum is multiplied by a factor to allow overlay of both spectra on a similar scale.

The associated template (scan with processing and conditions) will show the raw and processed spectrum superimposed on one page. The template also displays the processing functions and the instrument settings.

To run the method:

1. Open the **General QC** folder.
2. Highlight the **Scan and Second Derivative method** and then select **Run** from the File menu.
The method opens in Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range	700–600 nm
Scan speed	960 nm/min
Ordinate Mode	A
Slit width	1 nm
Data interval	1 nm
Number of cycles	1
Cycle time	1 second

5. Select the Sample info page.
6. Enter the number of **Samples**.
7. Select the Processing page.
The following processing is defined:

Process	Settings
Derivative	Order: Second; Width: 9
Arithmetic	Multiply; 20

The default report template is **Scan with Second Derivative**, and this is set to print **On demand**.

8. If required, alter the report frequency or output.

9. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Example Methods for High Performance Instruments

The following Example Methods are provided for High performance instruments. They are divided into groups based on their application. You can select which group(s) of methods to install when adding a new instrument. For further information about a particular method, see:

Color

 Color Difference (Delta E)

 HP Reflectance Color User Selectable

 Simple Transmittance Color Method 5 nm

 Whiteness & Yellowness Index

Materials

 Camouflage Using Integrating Sphere

 Measurement of Glass and Architectural Materials to EN410

 Opacity

 Haze ASTM D-1003

Optics

 Bandpass Filter Characterization

 Cut-off Filter Characterization

 Film Thickness Using Interference Fringes

Solar Reflectance

 Terrestrial Solar Irradiance

To run an example method:

1. Open the **Example Methods** folder and then open the **High Performance Instruments** methods folder.
2. Open the appropriate method subfolder and highlight the required method.
3. Select **Run** from the File menu.
The method opens in the Workspace.

Bandpass Filter Characterization

To run the method:

1. Open the **Optics** folder.
2. Highlight the **Bandpass Filter** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 420–380 nm

Ordinate Mode %T

Data interval 0.10 nm

5. Select the Sample info page.
Nominal Wavelength (nm), **Out of Band LH Point (nm)** and **Out of Band LH Point (nm)** columns are added to the table by default.
6. Enter the number of **Samples**.
7. Select **Corrections** in the Folder List and select the appropriate options.
For more accurate results, it is recommended that you select **0%T/ Blocked Beam Baseline** and **Use internal attenuator**.
8. Enter the **Nominal Wavelength** and **Out of Band** wavelength for each sample.
9. Select the Processing page.
Equations are defined within Processing page.
(Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following values for each sample: **Nominal Wavelength (nm)**, **Peak Wavelength (nm)**, **Peak Transmittance (%)**, **Center Wavelength (nm)**, **Out of Band LH Point (nm)**, **Out of Band LH Point (nm)**, **Mean Out of Band Transmittance (%)** and **Wavelength Pass/Fail**.

The default report template is **Bandpass Filter** and this is set to print on demand.

10. If required, alter the report frequency or output.

11. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Camouflage Using Integrating Sphere

To run the method:

1. Open the **Materials** folder.
2. Select the **Camouflage Using Integrating Sphere** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 1200–300 nm

Ordinate Mode %R

Slit width 1 nm

Data interval 5 nm

5. Select the Sample info page.

The column **Type** is added to the table by default. The options are **Blank**, **Control** and **Sample**.

6. Enter the number of **Samples**.

The default report template is **Camouflage Report** and this is set to print **On demand**.

7. If required, alter the report frequency or output.

8. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Color Difference (Delta E)

The Color Difference is a method used to calculate the color difference between two sets of L*, a*, b* values using illuminate D65 (daylight with a color temperature of 6500 K) and a 10 degree angle, standard observer. A reference spectrum (in either %T or %R ordinate mode) with a wavelength range of 380–780 nm and a data interval of 5 nm is added to the method.

To run the method:

1. Open the **Color** folder.
2. Highlight the **Color Difference (Delta E)** method and then select **Run** from the File menu.

The method opens in the Workspace.

3. Select **Data Collection** in the Folder List.

The Data Collection page is displayed.

4. Ensure the following parameters are entered:

Scan	780–380 nm
range	
Ordinate	%R
Mode	
Data	10 nm
interval	

5. Select the Sample info page.

The columns Reference Name, KL (1=default, 2=textiles), Kc, Kh, K1 (0.045 graphic arts/0.048 textiles) and K2 (0.015 graphic arts/0.014 textiles) are added to the table by default.

6. Enter the number of **Samples**.

7. Select the Processing page.

Equations are defined within the Processing page.

The data is interpolated over the scan range with an interval of 5 nm.

8. Ensure that the Results Table is defined to display the following values for each sample – X D65/10, Y D65/10, Z D65/10, Chrom x D65/10, Chrom y D65/10, L* D65/10, a* D65/10, b* D65/10, Cab* D65/10, hab*, X Ref, Y Ref, Z Ref, Chrom x ref, Chrom y Ref, L* Ref, a* Ref, b* Ref, Cab* Ref, hab* Ref, Delta L*, Delta a*, Delta b*, Delta Cab*, Delta hab*, Delta H, and Delta E CIE 1976 and DCIE 1994.

The default report template is **Delta E** and this is set to print **On demand**.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.
The report is displayed.

Measurement of Glass and Architectural Materials to EN410

This method for High Performance Instruments is for measuring solar, light and UV transmission (not transmittance) and reflectance for a sample measured in transmittance and reflectance using a sphere (either 150 or 60mm) according to EN410.

The settings defined in this method are for a 150 mm integrating sphere. If you have a 60 mm integrating sphere, the NIR gain may need reducing.

To run the method:

1. Open the **Materials** folder.
2. Highlight the **Measurement of Glass and Architectural Materials to EN410** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 2500–250 nm

Ordinate Mode %T

Data interval 5 nm

5. Select the Sample info page.
Pol. Angle and **Mode** columns are added to the table by default.

6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within the Processing page. Select an Equation and then click **Settings** to view the Equation.
The Results Table is defined to display the following values for each sample: **Solar Transmission (Ts)**, **Visible Transmission (Tv)**, **UV Transmission (Tuv)**, **Solar Reflectance (Rs)**, **Vis Reflectance (Rv)** and **UV Reflectance (Ruv)**.
The default report template is **EN410 Report**.
8. If required, alter the report frequency or output.

9. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Cut-Off Filter Characterization

This example method is used to characterize interference and uses a Scan method. This example is based on a 350 nm cut-off filter, but can be modified by changing the From and To wavelengths.

To run the method:

1. Open the **Optics** folder.
2. Highlight the **Cut-Off Filter Characterization** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Wavelengths 420–300 nm

Ordinate Mode %T

Slit Width 1 nm

5. Select the Sample info page.
Nominal Wavelength (nm) column is added to the table by default.
6. Enter the number of **Samples**.
7. Enter the **Nominal Wavelength** for each sample.
8. Select the Processing page.
Equations are defined within the Processing page. Select an Equation and then click **Settings** to view the Equation.

The Results Table is defined to display the following values for each sample: **Nominal Wavelength (nm)**, **Maximum Transmission (%)**, **Minimum Transmission (%)** and **Cutoff Wavelength**.

The default report template is **Cutoff Filter Report** and this is set to print on demand.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Film Thickness using Interference Fringes

This method uses a Scan method.

To run the method:

1. Open the **Optics** folder.
2. Highlight the **HP Film Thickness Using Interference Fringes** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 800–300 nm

Ordinate Mode %T

Data interval 0.5 nm

5. Select the Sample info page.
Angle of Incidence, **Refractive Index**, **Low Eval WI** and **High Eval WI** columns are added to the table by default.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within the Processing page.

Select an Equation and then click **Settings** to view the Equation.

The Results Table is defined to display the **Thickness (Microns)** of each sample.

The default report template is **Film Thickness** and this is set to print when the task is completed.

8. If required, alter the report frequency or output.

9. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Opacity Calculation for Paints and Coatings

This example method is used to calculate the opacity of paints and coatings using ISO 2814 using a 0°/d sphere collection geometry. The method measures the tristimulus Y value of a sample coated on both a black and a white substrate.

The percentage opacity is calculated using:

$$\% \text{ Opacity} = (Y_{\text{black}} * 100) / Y_{\text{white}}$$

To run the method:

1. Open the **Materials** folder.
2. Select the **Opacity** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 780–380 nm

Ordinate Mode %R

Data interval 10 nm

5. Select the Sample Info page.

The columns **Type** and **Description** are added to the table by default. The default type is Measurement.

The Description column for Measurement 1 states Coating on Black Substrate.

The Description column for Measurement 2 states Coating on White Substrate.

6. Enter the number of **Samples**.

7. Select the Processing page.

Equations are defined within Processing page.

(Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following for each sample: **Color Analysis**, **Description**, **Yblack**, **Ywhite**.

The default report template is **Opacity** and this is set to print **On demand**.

8. If required, alter the report frequency or output.

9. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

Haze ASTM D-1003

This method is designed to calculate the haze of transparent plastics according to ASTM D-1003.

The method is based on a series of measurements on an integrating sphere (usually 150 mm type). It uses illuminate A (Tungsten lamp) or C (Filtered tungsten lamp to emulate daylight) and an observer angle of 2 (CIE 1931) or 10 degrees (CIE 1964).

The transmittance spectra are converted to luminous transmission and the haze calculated.

The data used in the method is obtained between 780 and 300 nm, with a 5 nm data interval.

To run the method:

1. Open the **Materials** folder.
2. Highlight the **Haze ASTM D-1003** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 780–300 nm

Ordinate Mode %R

Data interval 5 nm

5. Select the Sample Info page.
The columns **Illuminant (A=1,C=2)** and **Std Obs (2 or 10 degrees)** are added to the table by default.
6. Enter the number of **Samples** and, for each sample, the appropriate **Illuminant** and **Std Obs** values.
7. Select **Corrections** in the Folder List and select the appropriate options.
For more accurate results, it is recommended that you select **100%T/0A Baseline (Autozero)**, and **Use internal attenuator**.
8. Select the Processing page.
Equations are defined within Processing page.
(Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following values for each sample: **Haze (%)**, **Diffuse Luminous Transmittance (%)**, **Y Sample (Light Transmittance)**, **Y Sample and Instrument Scatter**, **Y Instrument Scatter**, **Illuminant (A=1,C=2)**, **Std Obs (2 or 10 degrees)**.

The default report template is **Haze Report** and this is set to print on demand.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select Output on the Folder List and click **Preview**.
The report is displayed.

HP Reflectance Color User Selectable

The HP Reflectance Color User Selectable method is based on the CIE (Commission Internationale de l'Eclairage) conventions. This method uses illuminate A (Tungsten lamp), C (Filtered tungsten lamp to emulate daylight) or D65 (daylight with a color temperature of 6500 K) and an observer angle of 2 (CIE 1931) or 10 degrees (CIE 1964). It is designed for use with diffuse reflectance using an integrating sphere. The first part of the method calculates the tristimulus values X, Y and Z (corresponding to the primary using the illuminate and observer tables issued by CIE. X, Y and Z represent the primary colors – red, green and blue.

The method calculates L*, a* and b* coordinates and the chroma (Cab*) and hue (Hab*).

Data is collected as a scan for speed and diagnostic purposes but it is reported as an expanding table.

It can be adapted for use with a transmittance accessory, but a thickness correction may be needed for samples of a non-standard thickness. See Simple Transmittance Color Method.

To run the method:

1. Open the **Color** folder.
2. Highlight the **HP Reflectance Color User Selectable** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 780–380 nm

Data Interval 10 nm

Ordinate Mode %R

5. Select the Sample info page.
The columns **Illuminant (A=1,C=2,D65=3)** and **Std Obs (2 or 10 degrees)** are added to the table by default.
6. Enter the number of **Samples** and, for each sample, the appropriate **Illuminant** and **Std Obs** values.

7. Select the Processing page.

Equations are defined within the Processing page.

First the data is interpolated over the scan range with an interval of 5 nm.

(Select an Equation and then click **Settings** to view the Equation.)

8. Ensure that the Results Table is defined to display the following values for each sample

– **Tristim X, Tristim Y, Tristim Z, xvalue, yvalue, L*, a*, b*, Cab*** and **Hab***,
Illuminant (A=1,C=2,D65=3), Std Obs (2 or 10 degrees).

The default report template is **User Defined Color Report** and this is set to print **On demand**.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.

The report is displayed.

Simple Transmittance Color Method 5 nm

The Simple Color Method is based on the CIE (Commission Internationale de l'Eclairage) conventions. This method uses illuminate D65 (daylight with a color temperature of 6500 K) and a 10 degree observer angle (CIE 1964). It is designed for use with diffuse reflectance using an integrating sphere.

The first part of the method calculates the tristimulus values X, Y and Z (corresponding to the primary using the illuminate and observer tables issued by CIE. X, Y and Z represent the primary colors – red, green and blue.

The method calculates L*, a* and b* coordinates and the chroma (Cab*) and hue (Hab*).

Data is collected as a scan for speed and diagnostic purposes but it is reported as an expanding table.

To run the method:

1. Open the **Color** folder.
2. Highlight the **Simple Color Method 5nm** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range	780–380 nm
Ordinate Mode	%R

5. Select the Sample info page.

The columns **Color Analysis**, **Actual Thickness**, **Std Thickness** and **Reflectance Value (%)** are added to the table by default.

6. Enter the number of **Samples**.

7. Select the Processing page.

Equations are defined within the Processing page.

First the data is corrected for thickness; then the data is interpolated over the scan range with an interval of 5 nm.

(Select an Equation and then click **Settings** to view the Equation.)

8. Ensure that the Results Table is defined to display the following values for each sample – **Tristim X**, **Tristim Y**, **Tristim Z**, **xvalue**, **yvalue**, **L***, **a***, **b***, **Cab*** and **Hab***.

The default report template is **Color Analysis Report** and this is set to print **On demand**.

9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.

The report is displayed.

Terrestrial Solar Irradiance

This method is based on ASTM E 892-87, and calculates the terrestrial solar spectra irradiance at Air Mass 1.5 for a tilted 37 degree surface. This method can be used to measure parameters that are used to determine the efficiency of solar panels and photovoltaics. The method uses external data supplied in the ASTM. Wavelength data is rounded to the nearest integer. The data is interpolated to every 1 nm, allowing for data to be scanned at data intervals greater than 1 nm. If required, this step can be deleted and data scanned at 1 nm data intervals.

To run the method:

1. Open the **Materials** folder.
2. Highlight the **ASTM E 892-87 Terrestrial Solar Irradiance** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 2500–300 nm

Ordinate Mode %R

Data interval 5 nm

5. Select the Sample info page.
The **Description** column is added to the table by default.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within the Processing page.
(Select an Equation and then click **Settings** to view the Equation.)

The Results Table is defined to display the following value for each sample: **Solar Response**.

The default report template is **ASTM E892 Solar Response** and this is set to print on demand.

8. If required, alter the report frequency or output.

9. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.
The report is displayed.

Whiteness & yellowness Index

This method is used to calculate the Whiteness Index according to ASTM E 313-96 and the Yellowness Index according to ASTM D 1925-88. This example calculates data using a 2 degree observer (CIE 1932) and Illuminant C (North Sky Daylight with a color temperature of 6774 K). The method also calculates L*a*b*C*ab (Chroma) and h*ab (hue) for this observer angle and illuminant.

To run the method:

1. Open the **Color** folder.
2. Highlight the **Whiteness & Yellowness Index** method and then select **Run** from the File menu.
The method opens in the Workspace.
3. Select **Data Collection** in the Folder List.
The Data Collection page is displayed.
4. Ensure the following parameters are entered:

Scan range 780–380 nm

Ordinate Mode %R

5. Select the Sample info page.
The column **Color Analysis** is added to the table by default.
6. Enter the number of **Samples**.
7. Select the Processing page.
Equations are defined within the Processing page.
First the data is interpolated over the scan range with an interval of 5 nm.
(Select an Equation and then click **Settings** to view the Equation.)
8. Ensure that the Results Table is defined to display the following values for each sample
– **Tristim X**, **Tristim Y**, **Tristim Z**, **xvalue**, **yvalue**, **L***, **a***, **b***, **Cab***, **Hab***, **YI**, **WI** and **Tint**.

The default report template is **Whiteness Yellowness Index** and this is set to print on demand.

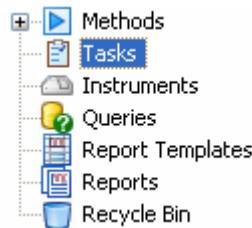
9. If required, alter the report frequency or output.

10. Click  to run the Method.

When the Method is run and the task has been generated, the results are calculated and inserted into the Results Table.

- To preview the report that will be generated, select **Output** on the Folder List and click **Preview**.
The report is displayed.

Tasks



Tasks are created using methods. A method (for example, a quant method) is used as a template to create a task (for example, a quant task). When data, such as sample information, is added and the samples are run, all the information (the template and the results) are saved together as a task.

Viewing tasks in the Explorer

What is displayed in the Main pane of the Explorer when a task folder is selected in the Folder List?

When a task folder is selected, the top section of the Main pane shows the folders and tasks on the relevant section of the Folder List.

The Full details view displays the columns: Name, Type, Modified by, Modified on, and Status.

The available task types are:  Scan,  Wavelength Program,  Scanning Quant,  Wavelength Quant,  Timedrive, and  Polarization Scan.

NOTE: Polarization Scan Task is only available for High performance instruments.

A task may have any of the following status types: In progress, Complete, Aborted, Reviewed, Approved.

What is displayed in the Display pane of the Explorer when a task is selected in the Main pane?

When a task is selected in the Main pane, the Display pane contains the details of the task: Name, Description, Type, Created by, Created on, Modified by, Modified on, Instrument serial no, Instrument type and Status (of the task).

What menu items are available when I right-click on a task from within the Explorer?

The following menu items are available:

View	Opens the selected task as read-only.
Continue	Continues the task. Allows you to collect data and add new samples to a task. NOTE: It is not possible to continue a task if it has been locked, reviewed or approved.
Re-process	Re-processes the selected task. Allows you to apply new processing commands to the data. NOTE: It is not possible to re-process a task if it has been locked, reviewed or approved.
Cut	Cuts the task to the clipboard.
Delete	Deletes the task to the recycle bin.

NOTE: If any menu item is disabled it means that you do not have the necessary permission to perform the operation.

What is displayed when I hover the mouse over a task?

When the mouse is hovered over the task in the Main pane, a tooltip displays the name of the task, the type of task and any description that was entered.

Is it possible to add folders to the tasks node?

Yes.

1. Select the Tasks node in the Folder List.
2. From the File menu select **New** and then select **Folder**.
A folder is added and can be renamed.

For more information about folders, please see folders.

How do I move tasks between folders?

1. Select the required Task and then from the Edit menu select **Cut**.
2. Select the new destination within the Explorer Tasks (for example, a different folder) and then click **Paste**.

Creating tasks from methods

Do I have to use a method to create a task?

Yes, you always have to start from an existing method. A task is a method that has been run and the data saved. A method can be used to create many tasks.

How do I create a task from a method?

1. Select the method upon which the task will be based and from the File menu (or by clicking the right mouse button on the method) select **Run**.

The method will open in the Workspace.

OR

Create a new method using the New Method Wizard, and select Edit on opening.

The method will open in the Workspace.

2. If necessary, complete the Sample Table, and any other nodes of the Folder List that were not completed (or need editing) when the method was created.

NOTE: You must have permission to edit methods to be able to alter the method settings.
For further information about your permissions, please contact your UV WinLab Administrator.

3. Click  .
The task is run.
4. From the File menu select **Save Results** and then select **ToTask**.
The Save Task dialog is displayed. A default name for the task is provided.
5. If required, edit the task **name** and enter a **Description**.
6. Click **Save**.
7. If you are using the Enhanced Security version of UV WinLab you are then prompted for your **User name**, **Password**, **Reason** and **Comment**.
8. Click **OK**.
The task is saved to the tasks folder in the Explorer.

If a method is superceded, can a previous version be used to create a task?

A locked, earlier version of a method can be used to create a task. However, none of the method parameters can be modified. When you attempt to open a locked, superceded method, the following message is displayed:

Method is locked, samples may be run but modification of method parameters will not be possible unless it is first unlocked.

Does a method have to be locked and approved before it can be used to create a task?

No, a method does not have to be locked and approved before it is used to create a task. However, for good practice we recommend that you do lock and approve methods before using them to create tasks.

Running and saving tasks

How does an analyst run a task?

By running a method. A method can be run from the Explorer or from a shortcut on the Desktop or the Start menu.

1. In the Explorer double-click on the method to be used to create the task.

OR

If you have created a shortcut to the method on the Desktop, double-click on the method icon.

The method opens in the Workspace.

2. Complete the Sample Table on the Sample tab on the Sample Info page.

3. Click .

The samples are run in the order listed in the Sample Table. The Graphs tab displays automatically at the start of the run and the spectrum appears in the graph display. A green tick appears next to the sample name in the Sample Table when it has been run.

For information on reporting and data export see Output.

NOTE: If the method being used is locked, and the incorrect accessory is fitted, an error message will be displayed informing the user that the method cannot be performed with the current configuration. The accessory configuration must be correct.

How do I save a task?

1. From the File menu select **Save Results**, and then from the submenu select **ToTask**.
The Save Task dialog is displayed.
2. Enter a **Name** if you wish to change the default Name, and a **Description** of the Task.
The default name is based on the method used to create the task, the date and the time.
3. Click **Save**.
The Save Results dialog is displayed.
4. Enter your User name, Password, Reason and Comment.
The fields displayed on this dialog depend on the settings previously determined by the UV WinLab Administrator.
5. Click **OK**.
The Task is saved under Task in the Explorer.

NOTE: If you try to exit a task without saving a message will be displayed asking if you wish to save the task before exiting. If you are using the Enhanced Security version of UV WinLab you must save the task before you exit.

NOTE: In the Enhanced Security version of the software, samples are automatically saved to the database between collections. You cannot save over an existing Task and so **Save As Task** will be inactive. You must select **Save Results** from the File menu and then select **As New Task** from the submenu.

NOTE: You can set up the Standard version of the software to always save a Task on closing, without displaying a message.

How do I set up UV WinLab to always save tasks on closing?

In the Standard version of the software, on closing a task a message is displayed asking if you wish to save the task before closing the program. If you wish UV WinLab to always save a task on closing, without displaying a message, you can select the **Always save task on close** option.

1. From the Administration menu select **Options**.
The Options dialog is displayed.
2. Select the check box **Always save task on close**.
3. Click **OK**.
The Save Results dialog is displayed.

NOTE: This option is only available with the Standard version of UV WinLab. If you are using the Enhanced Security version of UV WinLab you **must** save the task before you exit the method. A message will be displayed prompting you to save the task before exiting.

NOTE: If you are unable to see the Administration menu it is because you do not have the required permission. Permissions are defined by the UV WinLab Administrator. Please contact your UV WinLab Administrator for further details about your permissions.

Is it possible to overwrite an existing task?

In the Enhanced Security version of UV WinLab, each task must be saved with a unique name and it cannot be overwritten.

In the Standard version of UV WinLab, it is possible to overwrite an existing task if a duplicate name is specified when saving.

*Can I save my spectra as *.sp files?*

Yes, you can save the raw sample spectra in *.sp format for analysis in software such as **UV WinLab Data Processor and Viewer (DPV)**.

1. From the File menu select **Save Spectra**.
The Save Spectra dialog is displayed.

2. Select the files you wish to export.

A check mark indicates that a file is selected.

OR

Click **Select All** to select all the samples in the Sample Table.

3. Click **Save**.

The Save Results dialog is displayed.

4. Click **Browse**.

The Browse for Folder dialog is displayed.

5. Select the folder to contain the data.

OR

Click **Make New Folder**.

A new folder is added and you can name the folder.

6. Click **OK**.

7. To overwrite spectra already exported from the task, select **Overwrite Samples**.

NOTE: If you select any samples that have already been saved, without selecting **Overwrite Samples**, only the new spectra will be saved. The existing spectra will NOT be overwritten. A dialog will ask if you wish to continue.

8. Click **Save**.

- To deselect all the samples, click **Clear All**.

NOTE: You can also send your raw and processed spectra, as well as the spectral results of any equations, directly to UV WinLab DPV selecting **Send To UV WinLab DPV** on the File menu. The application will start with your data open in the Viewing Area (see the UV WinLab DPV Help file for information on UV WinLab DPV).

Can I save a Task and then Continue it at a later date?

Yes, you can continue any task that has not been locked, reviewed or approved. You can run samples that were not completed previously, or add new samples to a task that was completed.

- To continue a task, select the task and then from the File menu select **Continue**.
The task is opened in the Workspace.

Can I make changes to a task if I am part way through a Task?

Yes. When you close the Workspace, you will be asked if you wish to save the task. If you are using the Standard version of UV WinLab, and you choose not to save the task, the data that has just been collected will be lost. If you are using the Enhanced Security version of UV WinLab, you must save the task before exiting the Workspace. The **Save Settings As New Method**, and **Save As Task** commands on the File menu are all available.

Viewing tasks and re-processing data

How do I view the details and results of a task at a later date?

If you know the name of the task, double-clicking on it in the Explorer will open the task in the Workspace where you can view the settings and the results. The Data Collection and Instrument pages are grayed as they cannot be edited.

If you do not know the name of the task, you can search for it using a Query. The results of the query are displayed in the Query window and double-clicking on a result opens the task that generated the result in the Workspace.

How do I view a task that has previously been run?

It is possible to view all the information relating to a task that has previously been run.

- Select the task and from the File menu (or by right-clicking on the task) select **View**.
The task opens in the Workspace as read-only. All fields are grayed and cannot be edited.

The Save Task and Save As Task commands on the File menu are not available.

NOTE: Double-clicking on a task also opens it in the Workspace as read-only.

The task can be reviewed and approved (if you have the correct permissions) when it is opened as read-only.

NOTE: It is also possible to save all the settings (excluding the data) as a new method. From the File menu select **Save As** and then select **Method**. This applies to the Standard and Enhanced Security versions of UV WinLab.

Is it possible to re-process results?

Yes, it is possible to select different processing options and apply them to the collected data.

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have permission to re-process results.

1. From the Explorer Window, open the previously saved task by right-clicking on it and selecting **Re-process** or by selecting **Re-process** from the File menu.
2. From the Folder List in the Workspace, select **Processing**.
3. Select the new processing options.
The result of the processing can be seen in the graph pane below. Select **Processed** to see the processed spectra or **Both** to see the raw and processed spectra.
4. From the File menu select **Save As** and then select **Task**.
The re-processed task must be saved with a new name.
5. Enter a **Name** and **Description** and then click **Save**.
The Save Results dialog is displayed.

6. Enter your User name, Password, Reason and Comment.

The fields displayed on the dialog depend on whether you are using the Standard or Enhanced Security version of UV WinLab, and if you are using the Enhanced Security version – whether electronic signatures have been switched on by the UV WinLab Administrator.

7. Click **OK**.

The re-processed task is saved under Tasks in the Explorer Window.

NOTE: It is also possible to save all the settings (excluding the data) as a new method.

From the File menu select **Save As** and then select **Method**. This applies to both the Standard and Enhanced Security versions of UV WinLab.

How do I re-process a task?

NOTE: To re-process a task, you must have the necessary permission. Please contact your UV WinLab Administrator for further information about your permissions.

Re-process allows you to apply different processing commands to the data saved.

NOTE: It is not possible to add comments to the sample or standards table when re-processing a task.



1. Select the required task and then from Toolbar click .

2. The task opens.

Data collection is not available but the processing page is available for editing.

3. Edit the processing as required.

The results will be updated accordingly.

If you are using the Standard version of UV WinLab you can overwrite the previous data or save the task with a new name.

4. From the File menu select **Save** and then select **Task** to overwrite the original data.

OR

To save the task with a new name, select **Save As** and then select **Task**.

You will be prompted for a new name.

If you are using the Enhanced Security version of UV WinLab, you cannot overwrite the original data. The re-processed task must be saved with a new name.

5. From the File menu select **Save As** and then select **Task**.

6. If required, edit the task **name** and enter a **Description**.

7. Click **Save**.

8. If you are using the Enhanced Security version of UV WinLab you are then prompted for your **User name**, **Password**, **Reason** and **Comment**.

9. Click **OK**.

The reprocessed task is saved to the tasks folder in the Explorer.

NOTE: It is also possible to save all the settings (excluding the data) as a new method. From the File menu select **Save As** and then select **Method**. This applies to the Standard and Enhanced Security versions of UV WinLab.

How do I view a summary of the task?

- From the Tools menu within the Workspace, select **Summary**.
A summary of the Task is displayed in the Communiqué print preview window.

For further information on printing the summary, see the Communiqué Report Creator section of the Help.

Can I save a Task and then Continue it at a later date?

Yes, you can continue any task that has not been locked, reviewed or approved. You can run samples that were not completed previously, or add new samples to a task that was completed.

- To continue a task, select the task and then from the File menu select **Continue**.
The task is opened in the Workspace.

How can I easily locate a task I have previously run?

A query can be performed to search for a particular task.

For further information please see Queries.

A query can be performed to search for a particular task.

For further information please see Queries.

Can I make changes to a method if I am part way through a task?

Yes. When you close the Workspace, you will be asked if you wish to save the task. If you are using the Standard version of UV WinLab, and you choose not to save the task, the data that has just been collected will be lost. If you are using the Enhanced Security version of UV WinLab, you must save the task before exiting the Workspace. The **Save Method**, **Save As Method**, **Save Task**, and **Save As Task** commands on the File menu are all available.

Reviewing and approving tasks

What is meant by reviewing and approving?

The ability to formally review and approve methods, tasks, IPV setups and sample results is a function of the Enhanced Security version of UV WinLab.

It is up to the UV WinLab Administrator to set the correct privileges to ensure that only the appropriate people can 'sign off' data.

We use the term 'Review' to mean that the person has looked at the data and has agreed that it is correct. This is along the lines of a peer review and any number of people can review data as determined by your internal procedures.

We use the term 'Approve' to mean that a person with 'authority' has signed off the data as fit-for-purpose and again details of who is allowed to do this should be documented in your internal procedures.

How do I review a task?

When a task has been run, it is 'locked' by definition – that is, all fields are disabled and cannot be edited unless you have permission to Reprocess results.

A task is reviewed in the Workspace.

1. Open the task that you wish to review.
Double-click on the name of the task in the Explorer.
2. From the Tools menu select **Review** and from the submenu select **Task**.
The Review dialog is displayed.
3. Enter your User name, Password, Reason and Comment.
The fields that appear on this dialog depend on the settings previously defined by the UV WinLab Administrator.
4. Click **OK**.
The task event log is updated accordingly.

How do I approve a task?

When a task has been run, it is 'locked' by definition – that is, all fields are disabled and cannot be edited unless you have permission to Reprocess results.

NOTE: A task does not have to be reviewed before it is approved.

A task is approved in the Workspace.

1. Open the task that you wish to approve.
Double-click on the name of the task in the Explorer.
2. From the Tools menu select **Approve** and from the submenu select **Task**.
The Approve dialog is displayed.
3. Enter your User name, Password, Reason and Comment.
The fields that appear on this dialog depend on the settings previously defined by the UV WinLab Administrator.
4. Click **OK**.
The status of the task is **Approved**.

What are the available statuses of a task?

A task may have any of the following status types:

In progress (that is, incomplete)

Complete (all samples have been measured)

Aborted (abandoned by the user)

Reviewed (reviewed by the user)

Approved (approved by the user).

Instruments



When you select Instruments from the Folder List, the Main pane displays the instruments available.

The icon  indicates a Medium performance instrument, and the icon  indicates a High performance instrument.

The Main pane also allows you add a new instrument. The instrument can be a real instrument that you are attached to or a simulated instrument if you are not attached to an instrument. A simulated instrument will allow you to run tasks and "collect" simulated data.

NOTE: If you are unable to perform any of the procedures described below it is likely that you do not have the necessary permission. Permissions are assigned by the Administrator; contact them for further information on your permissions within UV WinLab.

NOTE: It is not possible to add folders to the Instrument node of the Folder List.

Viewing Instruments in the Explorer

What menu items are available when I right-click on an instrument?

The following menu items are available when you right-click on an instrument:

Edit		Displays the Edit Instrument dialog. NOTE: The user must have the necessary permission to edit an instrument.
Delete		Deletes the instrument.
Instrument Verification		NOTE: This menu item and all its submenu items are only available for Medium performance instruments.
	Display last IPV	Displays the IPV summary results. NOTE: This is only available if an IPV has previously been run.
	Create IPV Setup	Displays the IPV setup.
	Apply To Instrument	Displays the Apply IPV Setup dialog and associates that IPV setup with the default instrument.
	Perform Now	Displays a drop-down list of all available IPV Setups that will run on the default instrument. NOTE: Perform Now is grayed unless it is selected from the right-mouse menu on the default instrument.

Set as default instrument		Makes the currently selected instrument the default instrument. NOTE: It is possible to have both a High performance and Medium performance instrument as default simultaneously.
Manual Control		Opens the Manual Control task and applies it to the selected instrument.
Instrument Event Log		Displays the Event Log dialog for the Instrument. NOTE: This is only available for High performance instruments.
Calibrate Instrument		Displays the Calibration dialog for the Instrument. NOTE: This is only available for High performance instruments.
Instrument Properties		Displays the Properties dialog for the Instrument. NOTE: This is only available for High performance instruments.

What is shown in the Display pane when an instrument is selected?

The Display pane lists the following information:

- Name of the instrument
- Type of instrument
- Serial number of the instrument
- Status
- Last IPV Setup
- Port
- Installed on.

This information can be selected and copied to the clipboard using CTRL+C.

Adding and deleting instruments

What instruments are available?

The available Medium performance instruments are Lambda 25, 35, 45, 20, 40, 40P, 20Bio and 40Bio.

The available High performance instruments are Lambda 650, 750, 800, 850, 900, 950 and 1050.

Simulated versions of each instrument are also available. When a simulated instrument is added, the **Serial Number** is set to **Simulation** (and cannot be changed).

How do I add a new instrument?

Before collecting spectra you must add your instrument.

NOTE: You must have the configure instruments permission to be able to add an instrument.

NOTE: You must switch the instrument on and allow it to initialize before adding it to the software.

Adding a Medium performance instrument (Lambda 25, 35, 45, 20, 40, 40P, 20Bio and 40Bio)



1. In the Main pane double-click the **Add New Instrument**  icon.
The New Instrument Wizard starts.
2. Select Medium performance UV/Vis instrument from the drop-down list.
The description below the drop-down list details the available instruments.
3. Click **Next**.
4. Select the instrument type from the drop-down list, and if required, select **Make this the default instrument**.
5. Click **Next**.
6. If you wish to add a Medium performance simulated instrument, select **Simulation**.
A simulation allows you to simulate data collection.

NOTE: If you wish to use a Simulated Medium performance instrument, you must also setup the Simulator. See the on-screen Help for further information

7. From the drop-down list, select the **Port** the instrument is connected to.
This is grayed if Simulation is selected.
8. Click **Next**.

NOTE: Clicking **Next** will automatically perform a check to see if the correct instrument is attached to the selected port and switched on, and will display a Warning message if not.

9. Enter the **Name** and **Serial number** of the instrument.
The Serial Number can be found on the rear of the instrument. The Name will be displayed in the Explorer beneath the instrument icon. It can be edited in future if required.
10. Click **Next**.
The Methods Shortcuts page is displayed.
11. Select your shortcut options. If you wish to add shortcuts to the base methods for your instrument to the Start Menu or the Desktop, select the appropriate options.
All the options are selected as default.
12. Click **Next**.
The Example Methods dialog is displayed.

13. Select the group(s) of Example Methods that you would like to install.
The groups include: Biochemistry & Molecular Biology, Clinical & Healthcare, Environmental, Food & Drink, and General QC.
14. Click **Next**.
The Finished page displays all the selected settings.
15. If all the settings are correct click **Finish** to close the wizard and add the instrument.
The Wizard closes, a confirmation message that the instrument has been successfully installed is displayed, and the instrument is displayed in the Explorer. If you have selected to make this the default instrument, a tick mark is displayed next to the instrument icon.

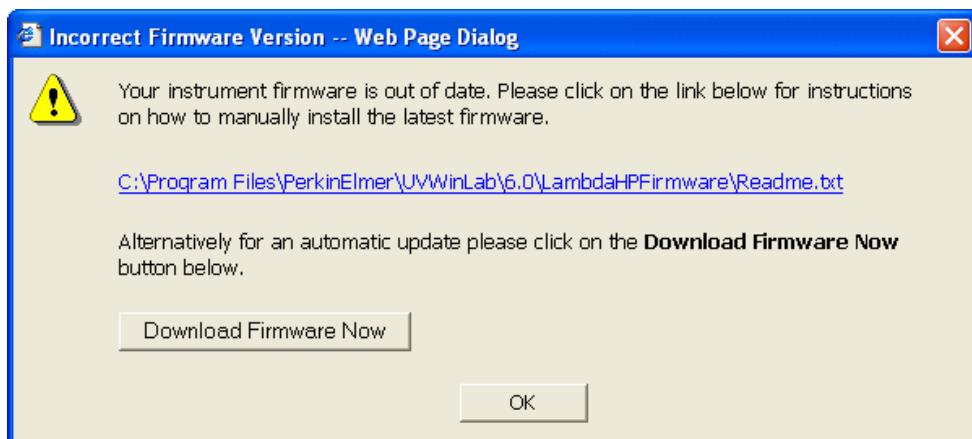
Adding a High performance Instrument (Lambda 650, 650R, 650S, 750, 750S, 850, 850R, 800, 900, 950, 1050 NB and 1050 WB)

1. In the Main pane double-click **Add New Instrument**.
The New Instrument Wizard starts.
2. Select High performance UV/Vis/NIR instrument from the drop-down list.
The description below the drop-down list details the available instruments.
3. Click **Next**.
4. Select the instrument type from the drop-down list, and if required, select **Make this the default instrument**.
5. Click **Next**.
6. From the drop-down list, select the **Port** to which the instrument is connected.

OR

If you are not connected to an instrument, select **Offline instrument**.

7. Click **Next**.
The software will check that the specified instrument type is connected. The software will also check the instrument firmware version. If this is out of date the following message is displayed.

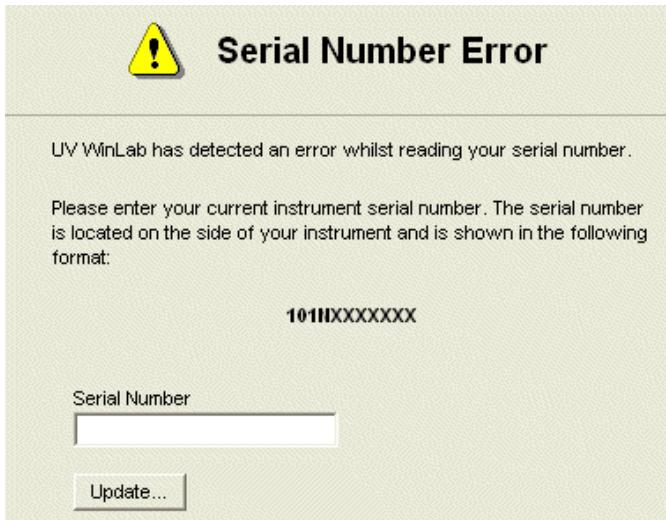


If you wish to download the firmware, click **Download Firmware Now**. When the firmware update is complete the instrument installation will continue.

NOTE: It will not be possible to proceed unless the new firmware has been installed.

The software will try and detect the serial number of the attached instrument. The next wizard screen you see depends on whether the serial number is successfully detected.

If the Serial Number is not correctly detected:



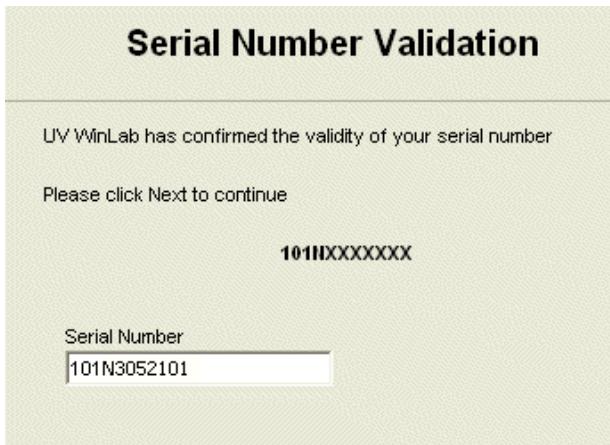
8. You must enter the **Serial Number** and then click **Update**.

NOTE: You will not be able to proceed with the instrument installation until you have updated the Serial Number.

9. When the Serial Number has been updated, click **Next** to continue.

OR

If the Serial Number is correct:



10. Click **Next** to continue.

11. Enter the **Name** of your instrument.
The Name will be used for identification throughout UV WinLab.
The Serial Number previously detected or entered is displayed but cannot be edited.
12. Select whether Common Beam Depolarizer, Double Polarizer/Depolarizer, and/or Sample/Reference Beam Attenuators are installed.
13. Click **Next**.
The Methods Shortcuts page is displayed.
14. Select your shortcut options. If you wish to add shortcuts to the base methods for your instrument to the Start Menu or the Desktop, select the appropriate options.
All the options are selected as default.
15. Click **Next**.
The Example Methods dialog is displayed.
16. Select the group(s) of Example Methods that you would like to install.
The groups include: Color, Optics, Materials and Solar Reflectance.
17. Click **Next**.
The Finished page displays all the selected settings.
18. If all the settings are correct click **Finish** to close the wizard and add the instrument.

The Wizard closes, a confirmation message that the instrument has been successfully installed is displayed, and the instrument is displayed in the Explorer. If you have selected to make this the default instrument, a tick mark is displayed next to the instrument icon.

How do I delete an instrument?

- Select the instrument to be deleted and then from the File menu select **Delete**.

OR

Right-click on the instrument and from the menu select **Delete**.

OR

Press the **Delete** key.

You are asked to confirm the deletion before the instrument is deleted.

How do I select an instrument to be the default instrument?

1. To select an instrument, click on the required instrument in the Main pane of the Explorer.
The instrument is highlighted.
2. From the Tools menu select **Set As Default Instrument**.

OR

Right-click on the selected instrument and select **Set As Default Instrument** from the menu.



A tick indicates the default instrument

How do I add a simulated instrument?

NOTE: Simulated instruments are only available for Medium performance instruments. For High performance instruments you can select to install an offline instrument. This will allow you to create methods for a particular method type with being attached to the instrument. However, you cannot simulate data collection.

Follow the instructions above for adding a new instrument, but make sure that **Simulation** is selected on the second screen of the New Instrument Wizard. When a simulated instrument is added, the name of the instrument in Explorer is Sim X (where X is the type of instrument), and the Serial number is Simulation.

Before the Simulated instrument can be used, the Simulator must be set up.

1. From the Start menu select Programs – PerkinElmer Applications – UV WinLab – Setup Simulator.
The UV WinLab Simulator Control dialog is displayed.
2. Select the **Simulation Name** from the drop-down list.
3. In the **Folder for Data** field, enter the full path for the folder containing the data to be used for the simulation.
The data to be used in the simulation must have a *.asc extension.
4. Click **Close**.
The simulator is setup.

Instrument Performance Verification

What are the different types of IPV status?

The different status types are:

Pass	The IPV is in date and was passed.
Overdue	The IPV is out of date.
Fail	The last IPV failed.
None	Not applicable for simulated instrument or High performance instrument.

NOTE: IPV is not available for High performance instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, 1050 NB and 1050 WB). The status will be set to **None**.

Instrument Details

How do I open or edit instrument details?

Double-click on an Instrument to display the Edit Instrument dialog.

Instrument type	The instrument type is displayed but cannot be edited.
Name	Enter a Name for the instrument. The instrument will be listed in the Explorer using this Name.
Serial Number	Enter the Serial Number of the instrument.
Port	Select the Port the instrument will communicate through from the drop-down list of available ports. The software will perform an automatic check to ensure that the correct type of instrument is connected to that port.

Instrument Properties

How do I view the properties of an instrument?

NOTE: This is only available for High performance instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 850R, 900, 950, 1050 NB and 1050 WB). In addition, you must have permission to configure instruments in order to view this dialog. For further information about your permission please contact your UV WinLab software Administrator.

1. In Explorer, highlight the instrument whose properties you wish to view.

2. From the Tools menu select **InstrumentProperties**.

OR

Right-click on the Instrument and select **Instrument Properties**.

The Properties dialog is displayed.

Lamps tab

3. Select the lamp type from the list of available internal lamps for the selected instrument.

The **Current Usage** field displays the usage of the selected lamp. This field is for information only and cannot be edited.

4. To reset the lamp usage after a lamp has been replaced, select **Reset Usage**.

NOTE: If the lamp has expired, it is not possible to just reset the **Current Usage**. The lamp must be replaced.

The **Current Usage** time is reset to zero when you click OK and exit the dialog.

NOTE: You must have the necessary permission to be able to reset the usage time. In addition, this action is a signature point and the information (user name, password, reason, comment) will be recorded in the audit trail.

5. Enter the **Lamp Life** of the selected lamp.

We recommend 2000 hours for a PerkinElmer-supplied deuterium lamp, and 1000 hours for a PerkinElmer-supplied tungsten lamp.

A reminder message to change the lamp will be displayed when the Current Usage approaches the Lamp Life.

6. To add an external lamp click **Add**.

A row is added to the External Lamps table.

7. Click in the **Name** field and enter the name of the lamp.

8. Click in the **Description** field and enter a description of the lamp.

9. Select whether the lamp is **Enabled** or **Disabled** from the drop-down list in the Enabled column of the table.

NOTE: It is not possible to have more than one lamp enabled.

- To remove a lamp from the table, click anywhere in the required row and then click **Remove**.

The lamp is removed.

NOTE: An external lamp will not appear on the Data Collection page within the Workspace unless it has been enabled from this Properties page.

- To enable an external lamp, select **Enable other lamps**.

This will enable the External lamp on the Data Collection page within the Workspace.

Instrument tab

The instrument tab displays the current filter table settings which can be modified, and also which items of firmware are enabled. This tab is mainly for use by PerkinElmer Service Engineers. You should not alter anything unless you fully understand the implications of doing so.

NOTE: The **Add** and **Remove** buttons are only available if you have Service Permission. For further information about your permissions, please contact your UV WinLab Administrator.

Filters are held on a disc within the instrument, which moves to the correct position when the appropriate wavelength is reached. The disc can only rotate in one direction.

The table below shows how the filter is related to the position (defaults). Positions 8, 9, and 10 are NIR filters and are only available on the Lambda 900, 950 and 1050.

Position	Wavelength	Filter
1	150	Glass Filter
2	319.2	T=100%
3	379.2	UG11
4	562.4	BG38
5	690.4	OG550
6	810.4	RG665
7	1190.4	RG780
8	1670.4	T-LPG-1.0
9	2680.8	T-LPG-1.5
10	3350	T-LPG-2.5
11		T=0%

A maximum of 20 rows can be specified in the Filter table.

1. To add a row, click **Add**.
A row is added to the table.
2. Click in the **Wavelength** field, and enter the required wavelength.
It is not possible to enter the same wavelength twice.

3. Click in the **Position** field and select the position from the drop-down list. The **Filter Type** is automatically updated.

OR

Click in the **Filter Type** field and select the filter from the drop-down list. The **Position** field is automatically updated.

The same filter position and filter type can be specified in the table more than once.

NOTE: The wavelengths can be added in any order. If you wish to reorder the wavelengths, you can click on the wavelength column header. When the dialog is closed the software will automatically reorder the table into wavelength order if they are not already.

- To delete a row, click anywhere in the row and click **Remove**.
 - To return the table to the default values, click **Defaults**.
1. For the firmware to recognize that a common beam depolarizer is installed, select **Enable common beam depolarizer**.
 2. For the firmware to recognize that sample and reference beam attenuators are installed, select **Enable internal attenuators**.
This option is not available on the Lambda 650, 650R, 650S, 750 and 750S.

How do I reset the lamp usage after I have changed a lamp?

1. To reset the lamp usage after a lamp has been replaced, select **Reset Usage**.
2. Click **OK**.
The usage is reset to 0:0.

What is the lamp life?

We recommend 2000 hours for a PerkinElmer-supplied deuterium lamp, and 1000 hours for a PerkinElmer-supplied tungsten lamp.

How do I add external lamps?

1. To add an external lamp click **Add**.
A row is added to the External Lamps table.
2. Click in the **Name** field and enter the name of the lamp.
3. Click in the **Description** field and enter a description of the lamp.
4. Select whether the lamp is **Enabled** or **Disabled** from the drop-down list in the Enabled column of the table.

NOTE: It is not possible to have more than one lamp enabled.

- To remove a lamp from the table, click anywhere in the required row and then click **Remove**.
Note that the lamp may still appear in the External Lamp drop-down list on the Instrument page.

How do I enable an external lamp?

NOTE: An external lamp will not appear on the Data Collection page within the Workspace unless it has been enabled from this Properties page.

- To enable an external lamp, select **Enable other lamps**.

This will enable the drop-down list of available External lamps on the Data Collection page within the Workspace. To use this lamp, the External lamp check box on the instrument page must be selected and the lamp selected from the drop-down list.

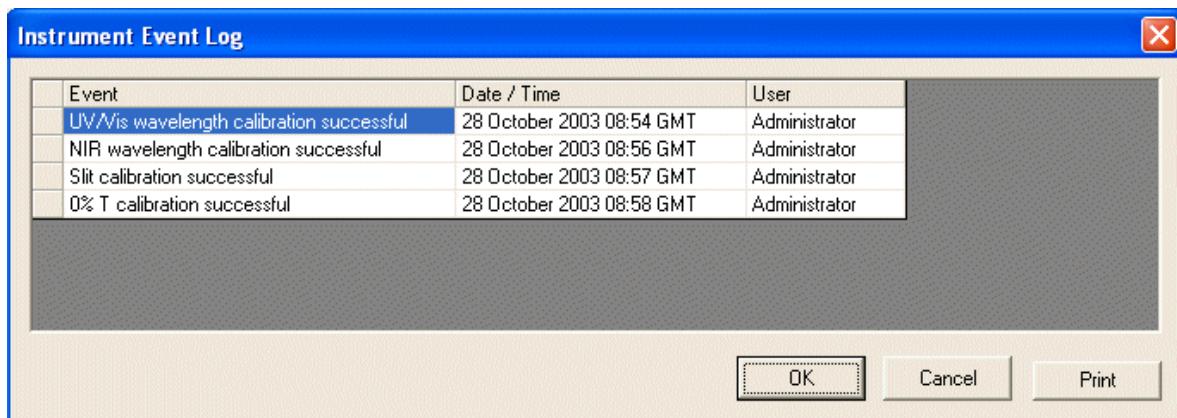
Instrument Event Log

How do I view the Instrument Event Log?

- Right-click on the instrument and select **Event Log**.

The Instrument Event Log dialog is displayed.

The Instrument Event Log records all calibration details, and all instrument property changes (for example, changing a lamp or detector):



How do I print the Instrument Event Log?

1. Right-click on the instrument and select **Event Log**.

The Instrument Event Log dialog is displayed.

2. Click **Print**.

All the Instrument Event Log information is printed to the default printer.

Calibrate Instrument

How do I calibrate my instrument?

NOTE: You must check that the beam path is clear before performing a calibration.

1. Right-click on the instrument and select **Calibrate Instrument**.

The Calibration Utility dialog is displayed.

There are three (four for Lambda 750/900/950/1050) calibration routines available – **UV/Vis Wavelength**, **NIR Wavelength**, **Slits**, and **0%T**.

NIR Wavelength is only available for the Lambda 750/900/950/1050. All, or a combination of these routines can be performed.

The dialog displays the names of the routines, a description, the last calibration date of the routine, and whether the test is enabled.

2. To view (and change) the settings for a particular routine, highlight the routine in the table and then click **Settings**.

The settings dialog for the selected routine is displayed.

NOTE: **Settings** is not available for **Slits**.

UV/Vis Wavelength

This calibration calibrates the UV/Vis wavelength range.

- Select **Manual set**, **Manual set 1 peak**, **Manual Set 2 peaks**, **Auto search 1 peak**, or **Auto search 2 peaks**.

Manual set – allows you to enter the **offset** and **factor** (without the need to determine the values in advance).

Manual set 1 peak – allows you to change the offset. A well known peak can be shifted to an exact wavelength.

- Enter the values of **Old Peak 1** and **New Peak 1**.

For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

Manual set 2 peaks – allows you to change the offset and factor by specifying the old and new values of 2 well known peaks.

1. Enter the values of **Old Peak 1** and **New Peak 1**.

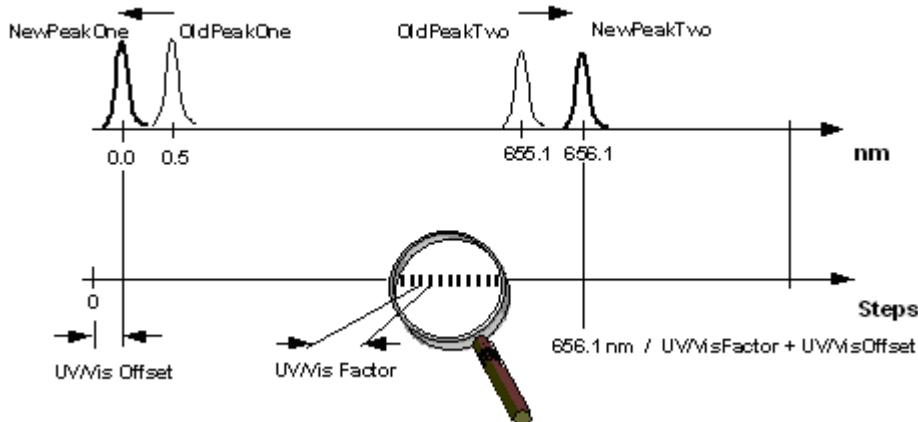
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

2. Enter the values of **Old Peak 2** and **New Peak 2**.

For example (as shown below), change of Old Peak 2 at 655.1 nm to New Peak 2 at 656.1 nm.

Auto search 1 peak – allows you to change the offset. The software performs an automatic search for the D₂ peak at 656.1 nm. The measured peak is then shifted to the exact wavelength.

Auto search 2 peaks – allows you to change the offset and factor. The software performs an automatic search for the D₂ peak at 656.1 nm and the peak at 0.0 nm. The measured peaks are then shifted to the exact wavelengths.



NIR Wavelength

This calibration calibrates the NIR wavelength range.

- Select **Manual set**, **Manual set 1 peak**, **Manual Set 2 peaks**, **Auto search 1 peak**, or **Auto search 2 peaks**.

Manual set – allows you to enter the **offset** and **factor** (without the need to determine the values in advance).

Manual set 1 peak – allows you to change the offset. A well known peak can be shifted to an exact wavelength.

- Enter the values of **Old Peak 1** and **New Peak 1**.
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

Manual set 2 peaks – allows you to change the offset and factor by specifying the old and new values of 2 well known peaks.

1. Enter the values of **Old Peak 1** and **New Peak 1**.
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.
2. Enter the values of **Old Peak 2** and **New Peak 2**.
For example (as shown below), change of Old Peak 2 at 1310.2 nm to New Peak 2 at 1312.2 nm.

Auto search 1 peak – allows you to change the offset. The software performs an automatic search for the D₂ peak at 1312.2 nm. The measured peak is then shifted to the exact wavelength.

Auto search 2 peaks – allows you to change the offset and factor. The software performs an automatic search for the D₂ peak at 1312.2 nm and the peak at 0.0 nm. The measured peaks are then shifted to the exact wavelengths.

0%T

This calibrates the electronic offsets.

- Select **Auto** to enable the software to determine the required offset.

OR

Enter the required **Calibration offset** value to be used.

1. Select the routines to be performed by the calibration.
A check mark indicates the routine is selected and will be performed.
2. Click **Calibrate**.
The calibrations are performed in the order listed in the table. Messages below the table show the progress of the calibration.

NOTE: An example message is 'D2-Peak 656.1 Pass 1: pass[Peak:654.90nm; Energy: 0.575E, Slit:1.00nm, Gain 63]'. This means that for the calibration test to find the D2 peak at 656.1nm, the test passed. For the first pass of the test (that is, the first time the test was run, with a slit of 1.00 nm), the peak was found at 654.90 nm. The energy and gain are reported. The peak found at 654.90 nm is then set to 656.1 nm.

When a calibration has passed, a green tick is displayed in the first column of the table. If the test fails, a red cross is displayed. In such an event please contact a PerkinElmer Service Engineer.

3. When the calibration has finished, click **Close** to close the dialog.

It is not possible to print the calibration results. The date and time of the calibration is recorded in the instrument event log.

NOTE: If you deselect calibration routines and then exit the dialog (either after performing the calibration or without performing the calibration), the next time you open the dialog all the options are selected by default.

Manual Control

Manual control is a special method that allows manual control of the default instrument and enables you to look at the ordinate result at a particular wavelength, or switch to alignment mode, without anything being saved to the database. It consists of only Instrument and Accessory settings and cannot be used to collect data. A typical use of Manual Control would be for aligning an accessory.

How do I select Manual Control?

- Ensure Instruments is selected on the Folder List in the Explorer, and then from the Tools menu select **Manual Control**.

The Manual Control workspace is displayed.

The top page, displayed by default or by selecting Manual Control from the Folder List, displays the **Name** and **Description**. These cannot be edited.

What menu items are available for Manual Control?

Only the following menus and menu items are available:

File menu	Exit	Closes the workspace.
Actions menu	Autozero	Zeros the instrument.
	Calibrate Instrument	Opens the calibration utility dialog that allows you to calibrate the attached default instrument. NOTE: This is only available for the Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950 and 1050.
	Instrument Properties	Opens the Instrument Properties dialog. NOTE: This is only available for the Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950 and 1050.
	Optimize Gains	Optimizes the gain based on the current settings. Allows you to choose whether to apply this to the open method. NOTE: This is only available for the Lambda 900, 950 and 1050. It is only available from the Actions menu when in the Workspace.
Help menu	Contents and Index	Displays the on-screen Help.

What commands are available on the Toolbar within Manual Control?

	Autozero
	Calibrate Instrument (only available for High performance instruments)
	Instrument Properties (only available for High performance instruments)

How do I perform Manual Control on an instrument that is not the default instrument?

You need to set the required instrument as the default instrument.

1. Select **Instruments** from the Folder List in Explorer.
2. Right-click on the instrument and select **Set As Default Instrument**.

You will now be able to perform Manual Control on the selected instrument.

Autozero

How do I perform an Autozero?

1. Enter the **Wavelength** at which you want to perform the Autozero and then click **Apply**.
The instrument goes to the selected wavelength.
2. From the Actions menu select **Autozero**.
The following message is displayed – 'Remove sample(s) and then press OK to perform a 100%T/OA correction'.
The Ordinate is set to zero absorbance.

Calibrate Instrument

How do I calibrate my instrument?

NOTE: Calibration is only available for high-performance instruments – Lambda 650, 650R, 650S, 750, 750S, 800, 850, 850R, 900, 950, 1050 WB and 1050 NB.

NOTE: You must check that the beam path is clear before performing a calibration.

1. From the Actions menu select **Calibrate Instrument**.
The Calibration Utility dialog is displayed.
There are three (four for Lambda 750/900/950/1050) calibration routines available – **UV/Vis Wavelength**, **NIR Wavelength**, **Slits**, and **0%T**. **NIR Wavelength** is only available for the Lambda 750/900/950/1050. All, or a combination of these routines can be performed.
The dialog displays the names of the routines, a description, the last calibration date and time of the routine, and whether the test is enabled.

- To view (and change) the settings for a particular routine, highlight the routine in the table and then click **Settings**.

The settings dialog for the selected routine is displayed.

NOTE: Settings is not available for **Slits**.

UV/Vis Wavelength

This calibration calibrates the UV/Vis wavelength range.

- Select **Manual set**, **Manual set 1 peak**, **Manual Set 2 peaks**, **Auto search 1 peak**, or **Auto search 2 peaks**.

Manual set – allows you to enter the **offset** and **factor** (without the need to determine the values in advance).

Manual set 1 peak – allows you to change the offset. A well-known peak can be shifted to an exact wavelength.

- Enter the values of **Old Peak 1** and **New Peak 1**.

For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

Manual set 2 peaks – allows you to change the offset and factor by specifying the old and new values of 2 well known peaks.

- Enter the values of **Old Peak 1** and **New Peak 1**.

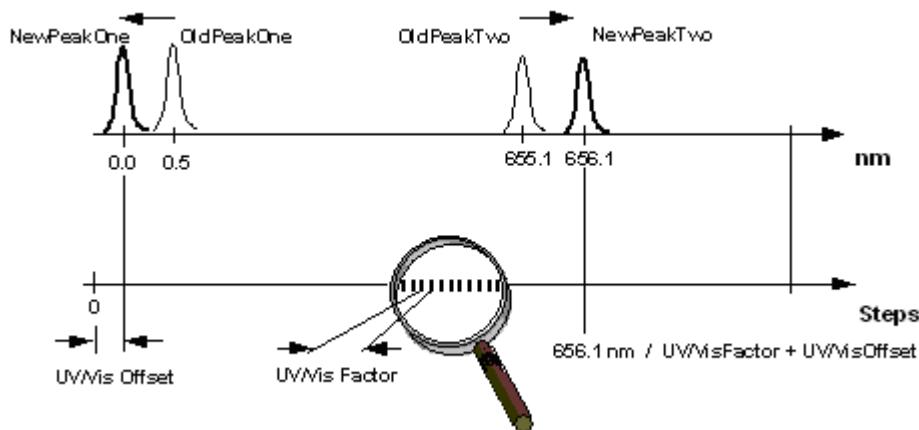
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

- Enter the values of **Old Peak 2** and **New Peak 2**.

For example (as shown below), change of Old Peak 2 at 655.1 nm to New Peak 2 at 656.1 nm.

Auto search 1 peak – allows you to change the offset. The software performs an automatic search for the D₂ peak at 656.1 nm. The measured peak is then shifted to the exact wavelength.

Auto search 2 peaks – allows you to change the offset and factor. The software performs an automatic search for the D₂ peak at 656.1 nm and the peak at 0.0 nm. The measured peaks are then shifted to the exact wavelengths.



NIR Wavelength

This calibration calibrates the NIR wavelength range.

- Select **Manual set**, **Manual set 1 peak**, **Manual Set 2 peaks**, **Auto search 1 peak**, or **Auto search 2 peaks**.

Manual set – allows you to enter the **offset** and **factor** (without the need to determine the values in advance).

Manual set 1 peak – allows you to change the offset. A well known peak can be shifted to an exact wavelength.

- Enter the values of **Old Peak 1** and **New Peak 1**.

For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

Manual set 2 peaks – allows you to change the offset and factor by specifying the old and new values of 2 well known peaks.

1. Enter the values of **Old Peak 1** and **New Peak 1**.

For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

2. Enter the values of **Old Peak 2** and **New Peak 2**.

For example (as shown below), change of Old Peak 2 at 1310.2 nm to New Peak 2 at 1312.2 nm.

Auto search 1 peak – allows you to change the offset. The software performs an automatic search for the D₂ peak at 1312.2 nm. The measured peak is then shifted to the exact wavelength.

Auto search 2 peaks – allows you to change the offset and factor. The software performs an automatic search for the D₂ peak at 1312.2 nm and the peak at 0.0 nm. The measured peaks are then shifted to the exact wavelengths.

$$R_{\text{actual}} = \frac{[R_{\text{measured}} - (R_0)] \times (R_{100})}{100 - (R_0)}$$

0%T

This calibrates the electronic offsets.

- Select **Auto** to enable the software to determine the required offset.

OR

Enter the required **Calibration offset** value to be used.

1. Select the routines to be performed by the calibration.

A check mark indicates the routine is selected and will be performed.

2. Click Calibrate.

The calibrations are performed in the order listed in the table. Messages below the table show the progress of the calibration.

When a calibration has passed, a green tick is displayed in the first column of the table. If the test fails, a red cross is displayed. In such an event please contact a PerkinElmer Service Engineer.

3. When the calibration has finished, click **Close** to close the dialog.

It is not possible to print the calibration results. The date and time of the calibration is recorded in the instrument event log.

NOTE: If you deselect calibration routines and then exit the dialog (either after performing the calibration or without performing the calibration), the next time you open the dialog all the options are selected by default.

How do I alter the settings of the calibration routines?

- To modify the settings of a routine, select the required row in the table and click **Settings**.

The appropriate calibration dialog is displayed.

NOTE: Settings is not available for Slit Calibration.

UV/Vis Two Peak

This calibration calibrates the UV/Vis wavelength range.

- Select **Manual set**, **Manual set 1 peak**, **Manual Set 2 peaks**, **Auto search 1 peak**, or **Auto search 2 peaks**.

Manual set – allows you to enter the **offset** and **factor** (without the need to determine the values in advance).

Manual set 1 peak – allows you to change the offset. A well known peak can be shifted to an exact wavelength.

- Enter the values of **Old Peak 1** and **New Peak 1**.

For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

Manual set 2 peaks – allows you to change the offset and factor by specifying the old and new values of 2 well known peaks.

1. Enter the values of **Old Peak 1** and **New Peak 1**.

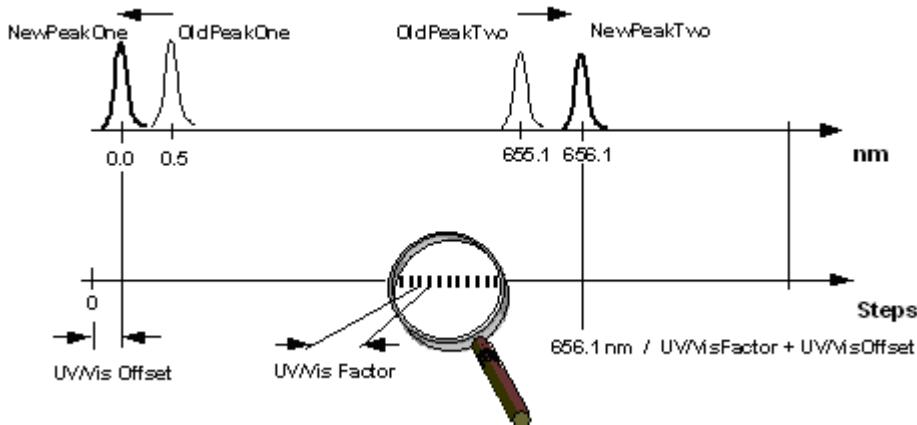
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

2. Enter the values of **Old Peak 2** and **New Peak 2**.

For example (as shown below), change of Old Peak 2 at 655.1 nm to New Peak 2 at 656.1 nm.

Auto search 1 peak – allows you to change the offset. The software performs an automatic search for the D₂ peak at 656.1 nm. The measured peak is then shifted to the exact wavelength.

Auto search 2 peaks – allows you to change the offset and factor. The software performs an automatic search for the D₂ peak at 656.1 nm and the peak at 0.0 nm. The measured peaks are then shifted to the exact wavelengths.



NIR Two Peak

This calibration calibrates the NIR wavelength range.

- Select **Manual set**, **Manual set 1 peak**, **Manual Set 2 peaks**, **Auto search 1 peak**, or **Auto search 2 peaks**.

Manual set – allows you to enter the **offset** and **factor** (without the need to determine the values in advance).

Manual set 1 peak – allows you to change the offset. A well known peak can be shifted to an exact wavelength.

- Enter the values of **Old Peak 1** and **New Peak 1**.
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.

Manual set 2 peaks – allows you to change the offset and factor by specifying the old and new values of 2 well known peaks.

1. Enter the values of **Old Peak 1** and **New Peak 1**.
For example, change of Old Peak 1 at 0.5 nm to New Peak 1 at 0.0 nm.
2. Enter the values of **Old Peak 2** and **New Peak 2**.
For example (as shown below), change of Old Peak 2 at 1310.2 nm to New Peak 2 at 1312.2 nm.

Auto search 1 peak – allows you to change the offset. The software performs an automatic search for the D₂ peak at 1312.2 nm. The measured peak is then shifted to the exact wavelength.

Auto search 2 peaks – allows you to change the offset and factor. The software performs an automatic search for the D₂ peak at 1312.2 nm and the peak at 0.0 nm. The measured peaks are then shifted to the exact wavelengths.

$$R_{\text{actual}} = \frac{[R_{\text{measured}} - (R_0)] \times (R_{100})}{100 - (R_0)}$$

0%T

This calibrates the electronic offsets.

- Select **Auto** to enable the software to determine the required offset.

OR

Enter the required **Calibration offset** value to be used.

Instrument Properties

How do I view the properties of an instrument?

NOTE: This is only available for high performance instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, WB 1050 and NB 1050). In addition, you must have permission to configure instruments in order to view this dialog. For further information about your permission please contact your UV WinLab software Administrator.

- From the Actions menu select **Instrument Properties**.

Lamps tab

1. Select the lamp type from the list of available internal lamps for the selected instrument.

The **Current Usage** field displays the usage of the selected lamp. This field is for information only and cannot be edited.

2. To reset the lamp usage after a lamp has been replaced, select **Reset Usage**.

NOTE: If the lamp has expired, it is not possible to just reset the **Current Usage**. The lamp must be replaced.

The **Current Usage** time is reset to zero when you click OK and exit the dialog.

NOTE: You must have the necessary permission to be able to reset the usage time. In addition, this action is a signature point and the information (user name, password, reason, comment) will be recorded in the audit trail.

3. Enter the **Lamp Life** of the selected lamp.

We recommend 2000 hours for a PerkinElmer-supplied deuterium lamp, and 1000 hours for a PerkinElmer-supplied tungsten lamp.

A reminder message to change the lamp will be displayed when the Current Usage approaches the Lamp Life.

4. To add an external lamp click **Add**.
A row is added to the External Lamps table.
 5. Click in the **Name** field and enter the name of the lamp.
 6. Click in the **Description** field and enter a description of the lamp.
 7. To enable the lamp, click in the **Enabled** check box.
Enabled lamps will appear in the External lamp drop-down list on the Instruments page.
- To remove a lamp from the table, click anywhere in the required row and then click **Remove**.
The lamp is removed.

NOTE: An external lamp will not appear on the Data Collection page within the Workspace unless it has been enabled from this Properties page.

Instrument tab

The instrument tab displays the current filter table settings and also which items of firmware are enabled.

NOTE: It is not possible to edit the Filter Table when the Instrument Properties have been accessed via the Actions menu from within Manual Control. You must right-click on the Instrument from within the Explorer and select Instrument Properties.

You should not alter the firmware options unless you fully understand the implications of doing so.

1. For the firmware to recognize that a common beam depolarizer is installed, select **Enable common beam depolarizer**.
2. For the firmware to recognize that sample and reference beam attenuators are installed, select **Enable internal attenuators**.
This option is not available on the Lambda 650, 650R, 650S, 750 and 750S.

How do I reset the lamp usage after I have changed a lamp?

1. To reset the lamp usage after a lamp has been replaced, select **Reset Usage**.
2. Click **OK**.
The usage is reset to 0:0.

What is the lamp life?

We recommend 2000 hours for a PerkinElmer-supplied deuterium lamp, and 1000 hours for a PerkinElmer-supplied tungsten lamp.

How do I add external lamps?

1. To add an external lamp click **Add**.
A row is added to the External Lamps table.
2. Click in the **Name** field and enter the name of the lamp.

3. Click in the **Description** field and enter a description of the lamp.
 4. To enable the lamp, click in the **Enabled** check box.
Enabled lamps will appear in the External lamp drop-down list on the Instruments page.
- To remove a lamp from the table, click anywhere in the required row and then click **Remove**.
The lamp is removed.

How do I enable an external lamp?

NOTE: An external lamp will not appear on the Data Collection page within the Workspace unless it has been enabled from this Properties page.

- To enable the lamp, click in the **Enabled** check box.
Enabled lamps will appear in the External lamp drop-down list on the Instruments page.

Optimize Gains

What is Optimize Gains?

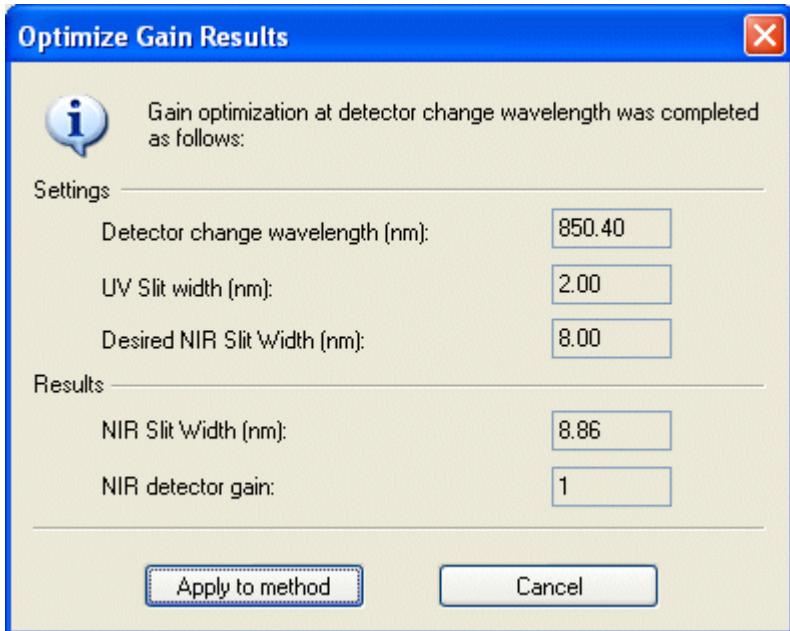
NOTE: Optimize Gains is only available for the Lambda 900, 950 and 1050. It is only available from the Actions menu when you are within the Workspace.

Optimize gain adjusts settings to avoid any discontinuity at the detector change point caused by differences in detector linearity. The primary method is to adjust the gain to ensure that the Servo slit width control will stay within its operating range (0.2–20 nm).

The energy (and thus the slit position) is affected by the accessory fitted. Therefore, balance should be done with the accessory installed.

1. The NIR slit mode is set to **Servo**.
2. Servo is based on the reference beam energy level.
3. The NIR gain is set to 0.
4. The **Monochromator** is set at a point just to the NIR side of the detector change.
5. The slit width is measured.
The slit width should be at the mid-point of its range. If it is too high, the gain is increased and step 3 onwards is repeated.
6. The UV slit width is set to one-quarter of the size of the NIR slit width, due to differences in the monochromator grating.
7. The UV gain is set to 100.

NOTE: When you select **Optimize Gains**, the Instrument Status Bar will update with the Instrument status. When the procedure is complete, a dialog will be displayed showing the result and asking whether you wish to apply the result to the current method:



Click **Apply to method** to apply the results to the current method, or click **Cancel** to reject the results and keep the current instrument settings.

See also

[Autosampler manual control](#)

[Sipper manual control](#)

[Cell changer manual control](#)

[Peltier manual control](#)

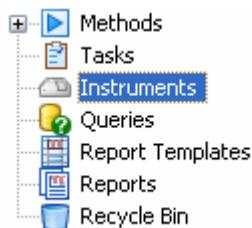
[Medium performance instruments](#)

[Manual Control – Data Collection](#)

[High performance instruments](#)

[Manual Control – Data Collection](#)

IPV Setup



NOTE: IPV is only available for Medium performance instruments – Lambda 25, 35, 45, 20, 40, 40P, 20Bio and 40Bio. The IPV commands on the Tools menu within the Explorer are only available when **Instruments** is highlighted in the Folder List.

There is a growing awareness among analytical chemists of the need to produce accurate and precise results. Quality control has become increasingly important in all areas of analytical chemistry. The quality of an analytical result depends on various factors among which instrumentation is one of the most important, and a number of regulations for single applications fields already exist.

To comply with national and international regulations and standards, such as Good Laboratory Practice (GLP), EN45000, or Pharmacopoeia, the performance of spectrometers must be checked at regular intervals.

An IPV is used to determine the fitness for purpose of an instrument. A large number of tests are available covering most appropriate regulatory bodies.

If you are using the Enhanced Security version of UV WinLab you must have the correct permission to allow you to create IPV Setups. Permissions are defined by the Administrator – please contact your UV WinLab Administrator if necessary.

NOTE: If you are using the Standard version of UV WinLab, many of the security features described below are not present as they are only necessary for Enhanced Security.

Creating an IPV Setup

What IPV tests are available?

The following IPV tests are available from the Tests page of the IPV Setup. For further information about a particular test, see:

Absorbance zero stability	Resolution with toluene
Baseline stability	Resolution toluene in methanol
Noise at wavelength 1	Stray light with potassium chloride (KCl)
Noise at wavelength 2	Stray light with potassium chloride (KCl) at 200 nm
Noise at wavelength 3	Stray light with sodium iodide (NaI) at 220 nm
Photometric accuracy with glass filters	Stray light with sodium nitrite (NaNO ₂)
Photometric accuracy with	Wavelength accuracy with glass filters

K2Cr2O7 solution

Photometric accuracy with K2Cr2O7 solution at 430 nm	Wavelength accuracy with solution
Photometric accuracy with potassium nitrate (KNO ₃) solution	Wavelength accuracy with third-party lamp
Photometric linearity grey glass	Wavelength repeatability D2
Photometric repeatability glass	
Resolution with benzene	

Who can create an IPV Setup?

Standard security – any user can setup, lock and unlock an IPV setup.

Enhanced security – a user must have permission to create IPV setups. The default group 'Developer' has permission to create IPV setups. In addition, if a user has permission to create IPV setups they can run an IPV on demand. Only users with permission to create IPV setups can run IPVs on demand.

How do I create a new IPV Setup?

1. From the Folder List, select **Instruments** and then click on (to select) the instrument whose performance you wish to verify.
2. From Tools menu select **Instrument Performance Verification** and then select **Create IPVSetup** from the submenu.
The IPV Setup is displayed. There are three pages in the Folder List for the IPV Setup – IPV Setup, Reporting and Tests. Each of these is described in turn below.
3. Select the required options on each page.

Setup

The Setup page defines the name of the IPV setup and the timing of the tests. It allows you to import and export an IPV setup. It also allows a Standard Operating Procedure (SOP) to be attached to the IPV setup.

Name	Select the Name of a Setup from the drop-down list of available IPV Setups. OR To create a new IPV Setup, click New. The New IPV Setup dialog is displayed. Enter a Name and click OK. The Name is displayed on the Setup page.
------	---

Description	If a previous IPV has been selected, the Description is displayed. You can edit this Description if required. If a new setup has just been created, you can enter a description for this new setup.
Save	Enables you to save the setup with the selected name.
Save As	Enables you to save the setup with a new name.
Import	Enables you to import an IPV setup created and exported from another PC.
Export	Enables you to export an IPV setup so that it can be used on another PC.
On demand only	Selecting On demand only disables all timing controls. The IPV tests can then only be run by selecting the Perform now menu command. See Timing. NOTE: This setup will only be used by someone with IPV authorization specifically choosing to do so.
Automatic tests occur at	Selecting Automatic tests occur at means that this setup will be triggered as specified for each test and the tests will be required the first time a task using this instrument is opened, after the time set here. See Timing.
Attach SOP	Enables you to attach a Standard Operating Procedure document that will be available to users when the tests are run.
Summary	Displays a summary of the tests selected and the settings chosen using a standard template.
Lock	Enables you to lock the setup.
Approve	Enables someone with the correct permission to approve the setup.
Review	Enables someone with the correct permission to review the setup.
Audit Trail	Displays the audit trail for the setup using a standard template.

Reporting

The IPV report can be printed as a hardcopy, or printed to the database. Either or both of these options can be selected.

Select the output(s) required.

Print hardcopy of report	Prints a hardcopy of the report to the default printer.
Save report to database	Puts a copy of the report into the Communiqué database. NOTE: Save report to database is not available as an option within the Enhanced Security version of UV WinLab as the report is always saved.

Report timing	Select to output a report After each test or At end of tests.
Report summary only	When selected, only a summary report is printed rather than the full report. The full report is available via a Query.

NOTE: If Report summary only is not selected, both the Full report and report summary are printed by default.

Tests

The Tests page displays all the available IPV tests and enables you to select the tests you wish to perform.

Available tests	A list of all the available tests.
Selected tests	A list of the tests to be included in the setup.
Add	Moves a test from the available list to the selected list.
Remove	Moves a test from the selected list to the available list.

The setup for each test is displayed by selecting the Test from the Folder List within the IPV Setup dialog. For further information on setting up tests see the help for the individual test.

Can I edit a previously created IPV Setup?

You can select and edit a previously created IPV setup.

1. From the Tools menu in Explorer select **Instrument Performance Verification** and then select **Create IPV Setup**.
The IPV Setup dialog is displayed.
2. Select the **Name** of the IPV setup that you wish to edit.
The parameters of the setup are displayed.
3. Edit the setup as required.
4. Click **Save**.
The setup is saved with the new parameters.

NOTE: If you are editing a locked IPV setup you will be forced to save the setup with a new name. The **Save** button is grayed; only the **Save As** button is available.

If you unlock a setup and edit it, the audit trail records the changes and the revision number of the setup, which is shown in the Audit Trail, is incremented.

Can I import an IPV Setup?

Yes. See Importing and exporting IPV setups.

Saving an IPV Setup

How do I save an IPV Setup?

- Once you have created the IPV Setup and have entered a **Name** on the Setup page, click **Save** or **Save As** on the Setup page.
If you are using the Enhanced Security version of UV WinLab you may be asked to select a reason and enter a comment. The dialog that appears depends on the settings previously defined by the UV WinLab Administrator.

NOTE: If you are using the Standard version of UV WinLab and you enter a name that has already been used you can overwrite the existing setup with the same name providing the setup has not been locked. If the original setup with the same name has been locked you will be prompted for a new name.

NOTE: If you attempt to save an IPV setup without entering all the necessary parameters a message is displayed. You can continue with the save or return to the test and complete the parameters.

Can I save an IPV Setup that is only partially completed?

Yes, it is possible to save the setup and return to it later to complete the test parameters. On re-opening the IPV setup you will be informed which tests are incomplete.

Importing and exporting IPV Setups

How do I import an IPV Setup?

1. On the Setup page of the IPV Setup dialog, click **Import**.
The Import setup dialog is displayed.
2. Select an IPV setup file (*.ipv) and click **Open**.
The IPV setup file is imported and the settings are displayed.

NOTE: If you try to import an IPV setup file with the same name as a setup file that already exists, you will be asked to rename the setup that is being imported.

NOTE: If a file has been changed outside of the UV WinLab software it cannot be imported.

NOTE: It is possible to import an incomplete IPV setup.

What happens if I try to import an IPV Setup with the same name as a Setup that already exists?

If you try to import an IPV setup with the same name as a setup that already exists, you will be asked to rename the setup that you are importing. An Import IPV Setup dialog is displayed.

1. Enter a new name and click **OK**.

If you are using the Enhanced Security version of UV WinLab, the User authentication dialog is displayed.

2. Enter your **User name** and **Password**. Select a **Reason** from the drop-down list, and if required enter a **Comment**.
3. Click **OK**.
The User authentication dialog closes.

NOTE: If you Cancel the User authentication dialog, the Setup file is not imported.

How do I export an IPV Setup?

1. From the IPV Setup page, select the **Name** of the Setup that you want to Export.
2. Click **Export**.
A Save As dialog is displayed.
3. Select the path where the file is to be saved.
The file must have the extension .ipv.
4. Click **Save**.
The dialog closes and a copy of the file is exported.

NOTE: If a file of the same name already exists in the place you are exporting to, you can overwrite the existing file or save the file with a different name.

NOTE: It is possible to export an incomplete setup.

IPV reports and summaries

How do I display the summary and what does it contain?

- On the IPV Setup dialog click **Summary**.
A summary report of the IPV setup is displayed in a Communiqué Print Preview window using the preset template.
The summary includes the name, description, Event log, the tests selected and test parameter details.

How do I print the summary report?

1. On the IPV Setup dialog click **Summary**.
A summary report of the IPV setup is displayed in a Communiqué Print Preview window using the preset template.



2. Click .
- The Print dialog is displayed.
- Define the Print settings as required.

NOTE: For further information on the Print settings dialog within Communiqué Print Preview see the Communiqué section of this Help.

4. Click **OK**.
The summary is printed as requested.

How do I view and print the IPV Setup audit trail?

NOTE: This is only applicable to the Enhanced Security version of UV WinLab.

5. Click **Audit Trail** on the setup page.

An Audit Trail report is displayed in a Communiqué Print Preview window using the preset template. The audit trail is combination of event log details and audit trail summary (changes documented at the point of locking).



6. Click .

The Print dialog is displayed.

7. Define the Print settings as required.

NOTE: For further information on the Print settings dialog within Communiqué Print Preview see the Communiqué section of this Help.

8. Click **OK**.

The summary is printed as requested.

NOTE: If you are using the Enhanced Security version of UV WinLab you will be prompted to confirm the Audit Trail printing. If electronic signatures have been turned on, you may be prompted for your user name, password, a reason and comment, depend on the settings previously defined by the UV WinLab Administrator.

How is the IPV report produced?

The type of output and the frequency of the report is defined on the Reporting page of the IPV Setup, and is saved as part of the IPV setup.

The IPV report can be printed as a hardcopy using the default printer or saved to the Communiqué database. Either or both of these options can be selected.

You can also select whether to output a report after each test or at the end of all the tests (where the number of tests run is defined by the timings settings of each test).

You can select to have only a summary report rather than the whole report. This summary report can be produced as a hard copy or saved to the database.

- Select **Report summary only** to only have a summary report.
The whole report is still available via a Query.

NOTE: If electronic signatures are switched on within the Enhanced Security version of UV WinLab, a report cannot be printed if it has not been successfully e-signed.

Running IPV tests

How frequently can I perform an IPV test?

You can select to perform the IPV tests **On demand only** or at a specified time.

NOTE: If you are using the Enhanced Security version of UV WinLab, only users who have permission to setup IPVs are able to perform IPVs on demand.

If you select **Automatic tests are due at**, enter the hour and select **am** or **pm** from the drop-down list.

NOTE: The hour must be in twelve hour format (0–12).

This setup will be triggered as specified for each test and the tests will be required the first time a task using this instrument is opened, after the time specified.

Automatic Tests Example – Setup1 has been configured with 2 tests: Test1 and Test2. Setup 1 has been configured to run at 7:00 am. Test 1 has been set to run every 2 days and Test 2 has been set to run every 5 days. The first time the setup is run, both the tests are run.

The next time the user logs into the application the system checks to see whether any of the tests are due. Since both tests were run, Test1 will now be due on the third day from the day it was run and Test2 will be due on the sixth day.

If the next time the user logs in it is the third day (2 days after the test was first run), Test1 will be due and the user will be prompted as such. Test2 is not due.

If the user next logs in on the sixth day, Test 1 and Test2 are due and the user will be prompted as such.

How do I apply an IPV Setup to an instrument?

1. Right-click on the default instrument, select **Instrument Performance Verification** and then select **Apply to instrument** from the submenu.
The Apply IPV Setup dialog is displayed. The dialog displays the name of the instrument that you are applying an IPV setup to.
2. Select the IPV setup to associate with the instrument from the drop-down list of available IPV setups.
3. Click **OK**.

The IPV setup is associated with the instrument.

NOTE: If an IPV setup that has not been fully setup is selected, a warning is displayed and the IPV setup dialog is then displayed with this setup loaded so that it can be completed.

NOTE: It is possible to run an IPV not associated with the instrument. Right-click on the instrument and select **Perform Now**. A dialog is displayed allowing you to select the IPV to run. This IPV will be run without the IPV setup being associated with the instrument.

How will I be reminded when tests are due if I have set them up to run automatically?

Selecting **Automatic tests occur at** on the setup page of the IPV setup means that this setup will be triggered as specified for each test and the tests will be required the first time a task using this instrument is opened, after the time specified. A message will be displayed telling you that IPV tests are now due on the instrument.

- Click **Perform now** to perform the required tests immediately.

OR

Click **Postpone** to close the dialog. The next time a task is opened that uses this instrument the IPV reminder dialog is redisplayed.

NOTE: if you are using the Enhanced Security version of UV WinLab it is only possible to postpone the tests if you have the correct permission. You will be asked to provide a reason for postponing the tests. If the tests are postponed, the IPV status of the instrument is set to NONE.

How many IPV Setups can I associate with an instrument?

Only one IPV setup can be associated with the connected instrument at any time.

How do I make an SOP available when the IPV is run?

It is possible to attach an SOP so that it is available when the IPV is run. If an SOP is attached, an SOP button is available when the IPV is run, allowing the user to click on it if they wish and display the SOP document.

1. On the IPV Setup page, select **Attach SOP**.
A check mark indicates that it is selected and that a button will appear at run time.
2. Click **Browse**.
The Open dialog is displayed.
3. Select the required document and click **Open**.
The full path and name of the document is displayed on the Setup page.

How do I add or remove a test from the IPV Setup?

The tests to be run are selected on the Tests page of the IPV Setup. The lists of tests in the Available tests column are tests that are available but are not currently part of the IPV Setup. The list of tests in the Selected tests column are the tests that are to be run as part of the IPV.

To add a test to the IPV

- Select the test from the list of Available tests and click **Add**.
The test is removed from the Available tests list and added to the Selected tests list.

To remove a test from the IPV

- Select the test from the list of Selected tests and click **Remove**.
The test is removed from the Selected tests list and added to the Available tests list.

How do I define the settings for each test?

The setup for each test is displayed by selecting the Test from the Folder List within the IPV Setup dialog. For further information on setting up tests see the help for the individual test.

NOTE: Each IPV test is configured independently of all the other tests.

Reviewing, approving and locking an IPV Setup

Does an IPV Setup have to be locked before it can be reviewed?

No. An IPV setup can be reviewed regardless of whether it is locked or unlocked.

If the IPV setup is locked and then reviewed, the Event log within the Audit Trail is updated.

How many times can an IPV Setup be reviewed before it is approved?

The IPV setup can be reviewed as many times as you like before it is approved.

How do I review an IPV Setup?

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the necessary permission to Review an IPV setup. If you do not have the correct permission, the Review button remains grayed.

An IPV setup does not need to be locked before it is reviewed.

1. Click **Review** on Setup page.
The Review IPV Setup dialog is displayed.
2. Enter your User name, Password, Reason and Comment.
The fields displayed on this dialog depend on whether you are using the Standard or Enhanced Security version of UV WinLab, and if you are using the Enhanced Security version – whether Electronic Signatures have been switched on by the Administrator.
3. Click **OK**.
The information is recorded in the Event log section of the Audit trail.

NOTE: It is not possible to review an approved IPV setup.

How do I approve an IPV Setup?

This applies to the Enhanced Security version of UV WinLab.

NOTE: An IPV setup can only be approved once it is locked.

1. Click **Approve** on Setup page.
The Approve IPV Setup dialog is displayed.
2. Enter your User name, Password, Reason and Comment.
The fields displayed on this dialog depend on whether you are using the Standard or Enhanced Security version of UV WinLab, and if you are using the Enhanced Security version – whether Electronic Signatures have been switched on by the Administrator.

3. Click **OK**.

The information is recorded in the Event history section of the Audit trail.

NOTE: You must have the necessary permission to Approve an IPV setup. If you do not have the correct permission, the Approve button remains grayed.

Is it possible to approve an IPV Setup more than once?

Yes, it is possible to approve an IPV setup more than once.

Each time the IPV setup is approved, the information is recorded in the Event log section of the Audit trail. This only applies to the Enhanced Security version of UV WinLab.

How do I lock an IPV Setup?

For the Standard version of UV WinLab:

- On the IPV Setup page, click **Lock**.
The IPV Setup is locked.

For Enhanced Security (ES):

1. On the IPV Setup page, click **Lock**.
The Locking IPV Setup dialog is displayed.
2. Enter your User name, Password, Reason and Comment.
The fields displayed on this dialog depend on whether you are using the Standard or Enhanced Security version of UV WinLab, and if you are using the Enhanced Security version – whether Electronic Signatures have been switched on by the Administrator.
3. Click **OK**.
The audit trail is activated.

NOTE: It is not possible to lock an incomplete IPV setup.

NOTE: Once the IPV setup has been locked it cannot be unlocked (and the Save button is grayed). However, you use the **Save As** command to create another version of the IPV Setup with a new name. This will be unlocked.

Can I view a locked IPV Setup?

Yes.

- On the IPV setup page, select the **Name** of the IPV setup that you want to view.

Does an IPV Setup have to be locked, reviewed and approved before it can be run?

No, an IPV setup can be run irrespective of its status.

NOTE: Reviewing and Approving is only applicable to the Enhanced Security version of UV WinLab.

Can I unlock an IPV Setup?

No. However, you can use the Save As command to create another version of the IPV setup. This will automatically be unlocked.

IPV Test Subsets

What are the USP Tests?

Photometric accuracy with glass filters (NIST 930/1930 or related);

Photometric accuracy with solutions (potassium dichromate);

Wavelength accuracy with deuterium lamp or Wavelength accuracy with glass filters or Wavelength accuracy with third-party lamp.

What are the EP/BP Tests?

Resolution with toluene solution;

Resolution with toluene in methanol;

Stray light with potassium chloride solution;

Photometric accuracy with solutions (potassium dichromate);

Wavelength accuracy with deuterium lamp or Wavelength accuracy with solution or Wavelength accuracy with third-party lamp.

What are the tests for the TGA Test (Australian code of good manufacturing practice for therapeutic goods test)?

Resolution with toluene solution;

Baseline flatness;

Stray light with potassium chloride solution;

Stray Light with potassium dichromate or Stray light with sodium nitrite;

Photometric accuracy with glass filters (NIST 930/1930 or related), or Photometric accuracy with solutions (potassium dichromate), or Photometric accuracy with potassium nitrate;

Wavelength accuracy with deuterium lamp or Wavelength accuracy with glass filters or Wavelength accuracy with solutions or Wavelength accuracy with third-party lamp.

See also

Running IPV tests

Absorbance Zero Stability Test

Absorbance zero stability is a measure of the degree of drift in the spectrometer reading from a set value within a specified period of time.

This test tests the absorbance stability using a Timedrive method over 60 minutes by calculating the trend at a specified wavelength and absorbance. The result must be less than or equal to the limit specified for the test to pass.

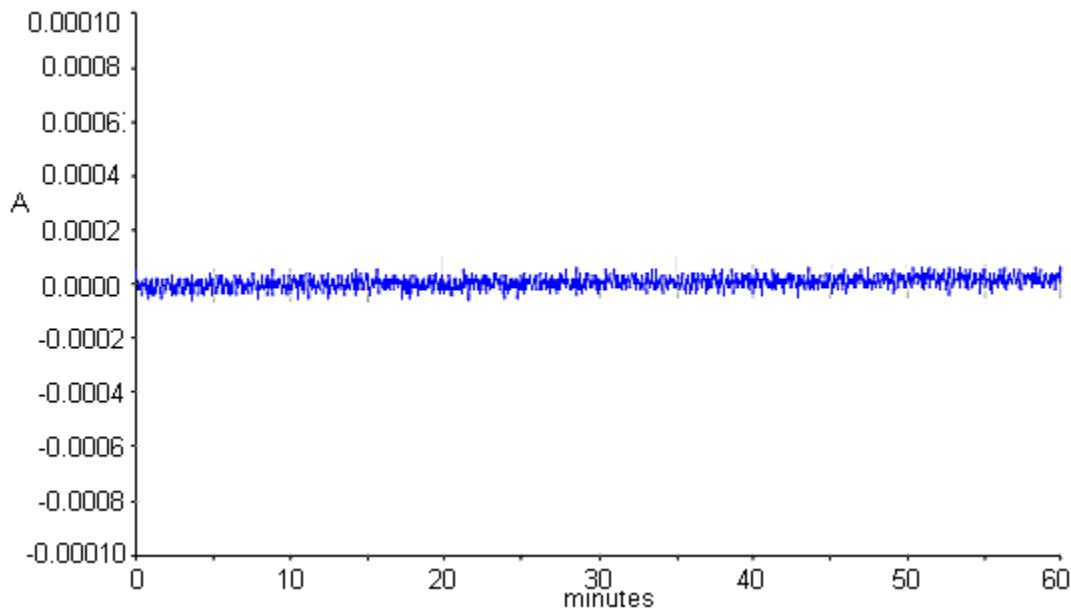
When might Absorbance Stability be an issue?

Absorbance stability is an issue when 100% transmission or zero absorbance continues to drift during a series of readings, affecting the value of each sample read, or causing inconvenience by constantly having to reset 100%T / zero absorbance.

Double beam spectrometers are usually very stable and have drift specifications of about 0.0002 A/h. Typical drift specifications for single beam instruments are about 10 times higher (0.02 A/h). A high stability (low drift) is of importance in all routine applications where the instrument is in continuous use. In kinetic analysis, the drift can be observed directly when enzymatic reactions are overlaid and the final result shows inaccuracy. A high drift may also affect quantitative analysis when a calibration has been performed and the baseline is shifting during the analysis of unknown samples.

In the absorbance stability test, a Timedrive method is used to measure the change of the absorbance reading during one hour. It is measured without samples and at the reference position (zero absorbance). For drift measurements it is especially important to let the instrument warm up. A double beam instrument needs a warm-up time of approximately 30 minutes, a single-beam instrument needs a warm-up time of about 1 hour.

The spectrum below shows a baseline measurement during a one hour period. Note the very high ordinate expansion with 0.002 A full scale.



How do I set up the Absorbance Zero Stability Test?

1. Select **Absorbance Zero Stability** from the list of tests in the Folder List on the IPV Setup page:



The Absorbance Zero Stability test page is displayed. The page contains all the default settings.

NOTE: The only entries that are required (if you wish to use the default values) are the prompts.

2. If required, change any of the default values.
3. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
4. Enter the text to appear before the test is run.
5. Repeat steps 3 and 4 for the **Sample**, **Pass** and **Fail** prompts.
6. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
7. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
8. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

9. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Absorbance Zero Stability to the list of tests I want to perform?

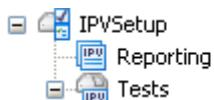
1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.

OR

Using the right mouse button, click on the instrument whose performance you wish to verify, and select **Create IPV Setup**.

The IPV Setup dialog is displayed.

3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:

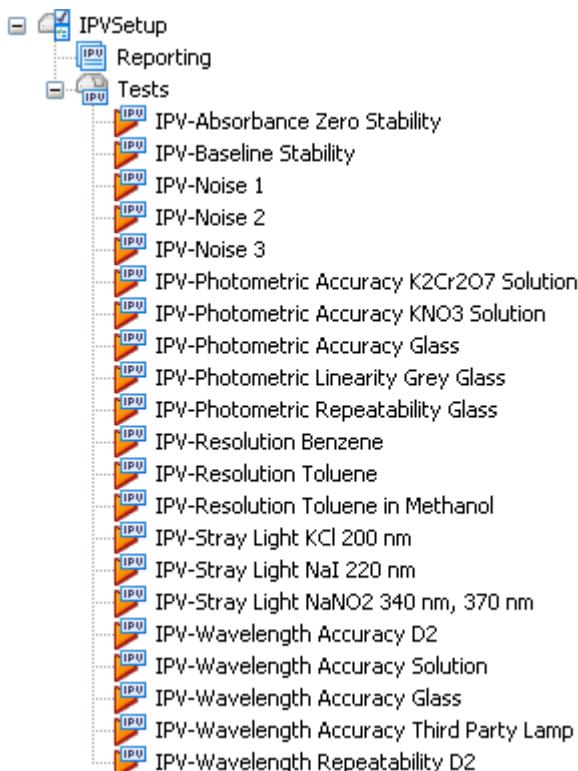


The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Absorbance Zero Stability** is not currently selected, it will appear in the **Available tests** list.

5. Select Absorbance Zero Stability and then click Add.

Absorbance Zero Stability moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Absorbance Zero Stability from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Absorbance stability in the Selected tests list and then click Remove.

Absorbance Zero Stability moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, maximum and minimum values?

	Default	Minimum	Maximum
Wavelength (nm)	500.00	190	1100
Absorbance (A)	0	0.0000	10.0000
Absorbance trend limit (A/hr)	< 0.001500	0.00	1.00
Perform every X days	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Slit width (nm)	1
Total Time (mins)	60
Time Interval (secs)	2
Ordinate Mode	A
Response (s)	2
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Timedrive method.

How does "Perform every" work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Baseline Stability Test

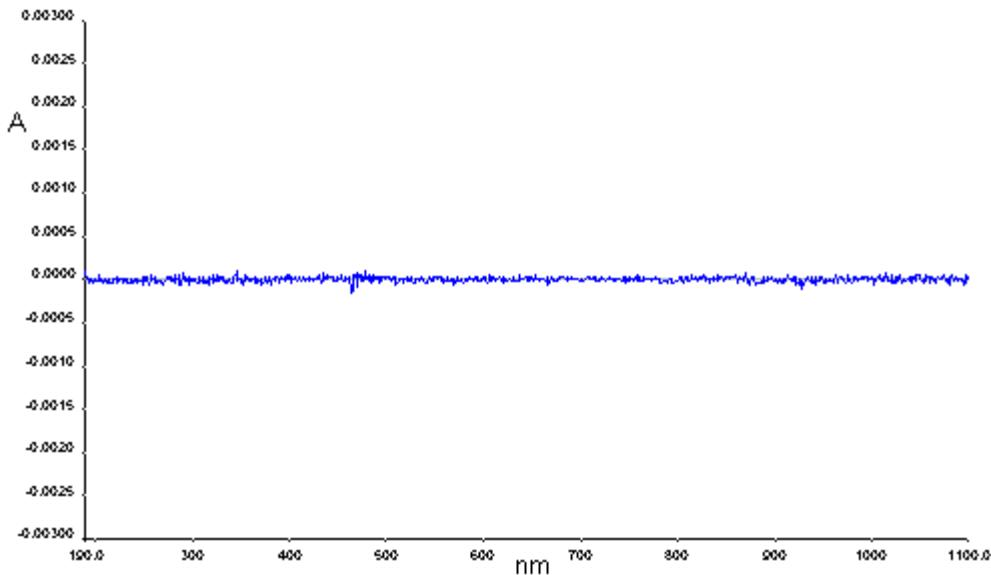
This test measures the baseline flatness at 0 A over the specification range of the instrument, against the limit specified.

When a spectrum is recorded over the entire wavelength range without sample or reference, the baseline stability specification can be defined. In practice, this line always has some noise from optical and electronic elements in the instrument, especially at the wavelength ends. In a high-quality, double-beam spectrometer, the baseline stability should be lower than 0.0001 A between 200 nm and 1100 nm at zero absorbance and a scan speed of 240 nm/min. Single beam instruments have about three times higher values for the corrected baseline. If the baseline stability is poorer, it may result in distorted spectra, especially when measuring at low concentrations.

How is Baseline Stability measured?

To measure baseline stability, a baseline correction is first performed. After baseline correction, the signal is recorded and the flatness can be determined from the resulting baseline. No standard material or solutions are required. When the test is performed, if the result is below or equal the upper limit, or higher than or equal to the lower limit, the test is passed.

The spectrum below is a typical graph of baseline flatness.



How do I define a Baseline Stability test?

1. Select **Baseline Stability** from the list of tests in the Folder List on the IPV Setup page:



The Baseline stability test page is displayed. The page contains all the default settings.

2. If you do not wish to use the default Baseline flatness limit, enter a new value.

3. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
4. Enter the text to appear before the test is run.
5. Repeat steps 3 and 4 for the **Pass** and **Fail** prompts.
6. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
7. If you do not want the tests to stop when a test fails, clear the check box beside **Stop tests on failure**.
8. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup with a new name

The Save or Save As dialog is displayed.

9. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Baseline Stability test to the list of tests I want to perform?

1. Select Instruments from the tree, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is show all the tests currently included in the IPV Setup
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the Tests folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV setup dialog displays the list of **Available tests** and **Selected tests**. If **Baseline Stability** is not currently selected, it will appear in the **Available tests** list.

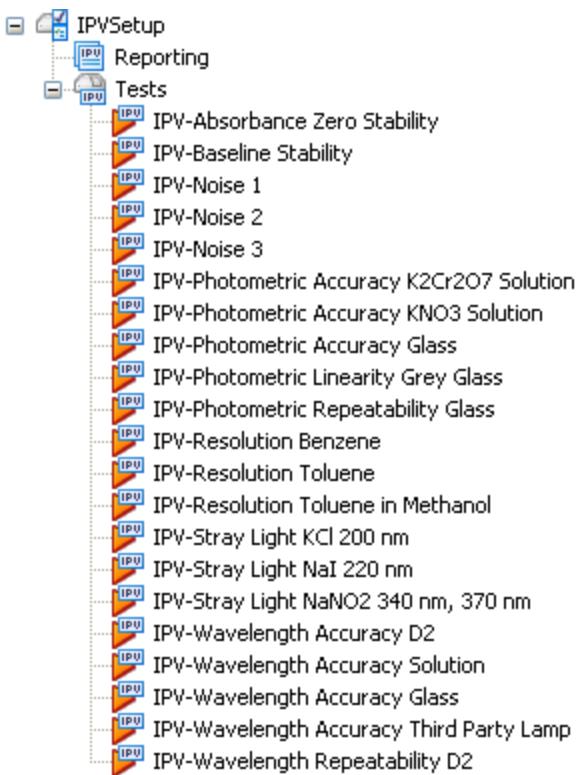
5. Highlight **Baseline Stability** and then click **Add**.

Baseline Stability moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Baseline Stability from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Baseline stability in the Selected tests list and then click Remove.

Baseline stability moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Baseline Flatness limits (A)	+/-0.001	0.0000	1.0000
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Start (nm)	1100 (1050 for Lambda 45)
End (nm)	200
Slit width (nm)	1
Data Interval (nm)	1.0
Ordinate Mode	A
Scan Speed (nm/min)	240
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop-down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

Noise

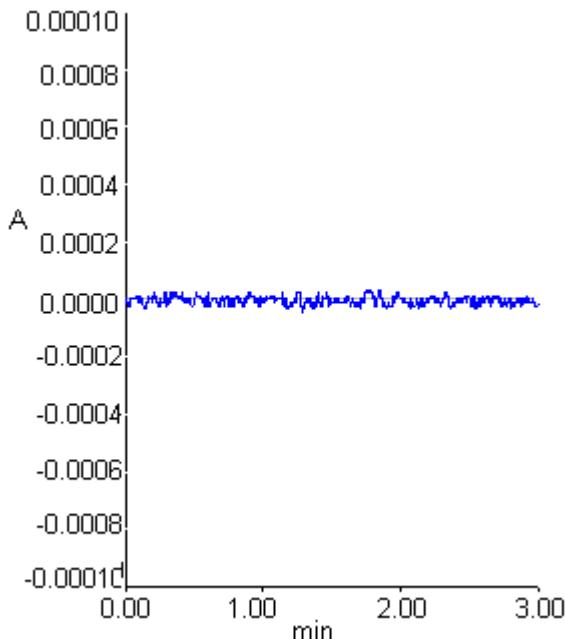
Noise means changes of the detector signal caused by optical and electronic effects and this interference causes uncertainty in the photometric information. The magnitude of the noise may vary across the spectrum and is most significant at low energy levels. The noise is defined at one wavelength compared to baseline stability. Low noise levels means high sensitivity measurements.

How is Noise measured?

Noise level is measured at a single wavelength for a short time period (3 minutes). No standard material or solutions are required. The noise level at the specified wavelength is determined from the resulting Timedrive spectrum by calculating the Standard Deviation (SD) at a specified wavelength and absorbance.

The noise level can be checked at different %T or absorbance levels.

The timedrive spectrum below shows the noise level measured at 500 nm over a 3 minute period. Note the high ordinate expansion. The maximum is 0.001 A and the minimum is –0.001 A.



There are three noise tests available:

Noise at Wavelength 1

Noise at Wavelength 2

Noise at Wavelength 3

Noise at Wavelength 1 Test

This test tests the noise over 180 seconds by calculating the Standard Deviation (SD) at a specified wavelength and absorbance. The result must be less than or equal to the limit specified.

When the test is performed, if the result is below or equals the upper limit, the test is passed.

How do I define a Noise at wavelength 1 test?

1. Select **Noise at wavelength 1** from the list of tests in the Folder List on the IPV Setup page:



The Noise at wavelength 1 test page is displayed. The page contains all the default settings.

2. If you do not wish to use the default limit, enter a new value.
3. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
4. Enter the text to appear before the test is run.
5. Repeat steps 3 and 4 for the **Sample**, **Pass** and **Fail** prompts.
6. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
7. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
8. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

9. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Noise at wavelength 1 test to the list of tests I want to perform?

1. Select Instruments from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the Tests folder on the Folder List within the IPV setup dialog:

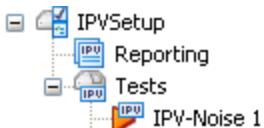


The folder expands to show all the tests currently included in the IPV Setup

The IPV setup dialog displays the list of **Available tests** and **Selected tests**. If **Noise at wavelength 1** is not currently selected, it will appear in the **Available tests** list.

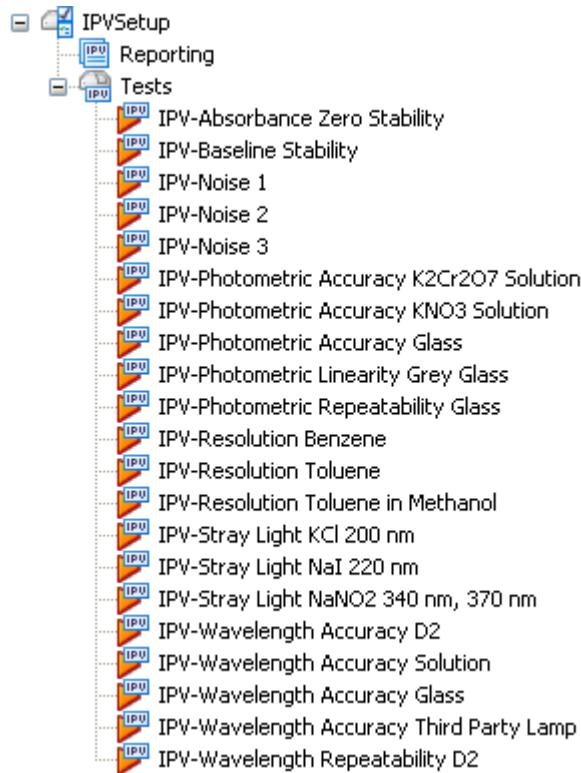
5. Highlight **Noise at wavelength 1** and then click **Add**.

Noise at wavelength 1 moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Noise at wavelength 1 from the list of tests I want to perform?

1. Select the Tests folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV setup dialog displays the list of **Available tests** and **Selected tests**.

2. Highlight Noise at wavelength 1 in the Selected tests list and then click Remove.
- Noise at wavelength 1** moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Wavelength (nm)	200	190	1100
Absorbance (A)	0	0.0000	10.0000
Noise SD limit (A)	0.00005	0.0000	1.0000
Perform every X days	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Wavelength (nm)	200
Slit width (nm)	1
Total Time (secs)	180
Time Interval (secs)	1
Ordinate Mode	A
Response (s)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Timedrive method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

IPV Setup

Running an IPV

Noise at Wavelength 2 Test

This test tests the noise over 180 seconds by calculating the Standard Deviation (SD) at a specified wavelength and absorbance. The result must be less than or equal to the limit specified.

When the test is performed, if the result is below or equals the upper limit, the test is passed.

How do I define a Noise 2 test?

1. Select **Noise 2** from the list of tests in the Folder List on the IPV Setup page:



The Noise at wavelength 2 test page is displayed. The page contains all the default settings.

2. If you do not wish to use the default **Wavelength**, enter a new value.
3. If you do not wish to use the default **Absorbance**, enter a new value.
4. If you do not wish to use the default **Noise SD** limit, enter a new value.
5. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
6. Enter the text to appear before the test is run.
7. Repeat steps 5 and 6 for the **Sample**, **Pass** and **Fail** prompts.
8. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **Days**, **Weeks** or **Months** from the drop-down list.
9. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
10. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV setup page and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

11. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Noise at wavelength 2 test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Noise at wavelength 2** is not currently selected, it will appear in the **Available tests** list.

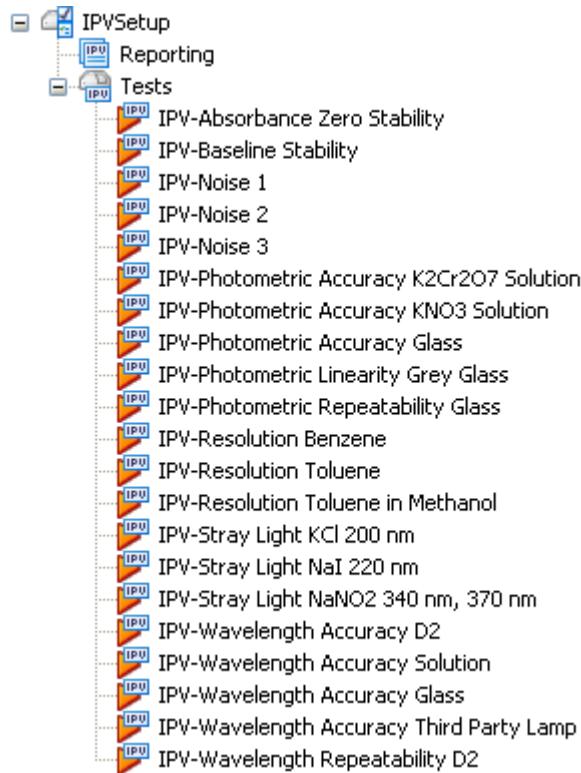
5. Select **Noise at wavelength 2** and then click **Add**.

Noise at wavelength 2 moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Noise at wavelength 2 from the list of tests I want to perform?

1. Select the Tests folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV setup dialog displays the list of **Available tests** and **Selected tests**.

2. Highlight Noise at wavelength 2 in the Selected tests list and then click Remove.
- Noise at wavelength 2** moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Wavelength (nm)	500	190	1100
Absorbance (A)	0	0.0000	10.0000
Noise SD limit (A)	≤ 0.00005 A	0.000000	1.000000
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Wavelength (nm)	500
Slit width (nm)	1
Total Time (secs)	180
Time Interval (secs)	1
Ordinate Mode	A
Response (secs)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Timedrive method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

IPV Setup

Running an IPV

Noise at Wavelength 3 Test

This test tests the noise over 180 seconds by calculating the Standard Deviation (SD) at a specified wavelength and absorbance. The result must be less than or equal to the limit specified.

When the test is performed, if the result is below or equals the upper limit, the test is passed.

How do I define a Noise at wavelength 3 test?

1. Select **Noise at wavelength 3** from the list of tests in the Folder List on the IPV Setup page:



The Noise at wavelength 3 test page is displayed. The page contains all the default settings.

2. If you do not wish to use the default limit, enter a new value.
3. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
4. Enter the text to appear before the test is run.
5. Repeat steps 3 and 4 for the **Sample**, **Pass** and **Fail** prompts.
6. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
7. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
8. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

9. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Noise at wavelength 3 test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Noise at wavelength 3** is not currently selected, it will appear in the **Available tests** list.

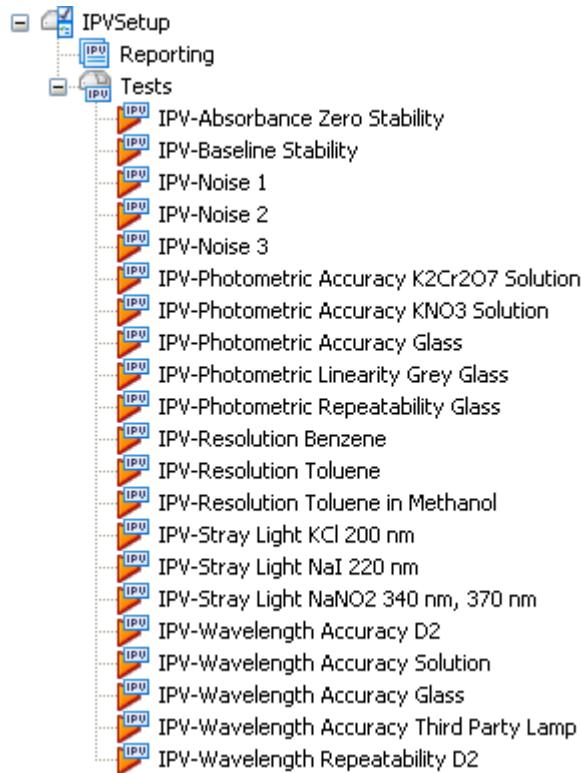
5. Select **Noise at wavelength 3** and then click **Add**.

Noise at wavelength 3 moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Noise at wavelength 3 from the list of tests I want to perform?

1. Select the **Tests** folder on the Tree within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Noise at wavelength 3 in the Selected tests list and then click Remove.

Noise at wavelength 3 moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Tree.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Wavelength (nm)	500	190	1100
Absorbance (A)	2	0.0000	10.0000
Noise SD limit (A)	≤ 0.00005	0.000000	1.000000
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Wavelength (nm)	500
Slit width (nm)	1
Total Time (secs)	180
Time Interval (secs)	1
Ordinate Mode	A
Response (secs)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Timedrive method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

IPV Setup

Running an IPV

Photometric Accuracy

Photometric Accuracy is defined as the closeness of the measured values of transmittance (%T) or absorbance (A) to the actual value, determined by measuring standards of known A and %T.

The photometric accuracy is a measure of the absolute correctness of an absorbance or transmittance value. It can only be specified with special calibration filters or calibration solutions.

There are three tests for photometric accuracy:

Photometric accuracy with glass filters

Photometric accuracy with K₂Cr₂O₇ solution

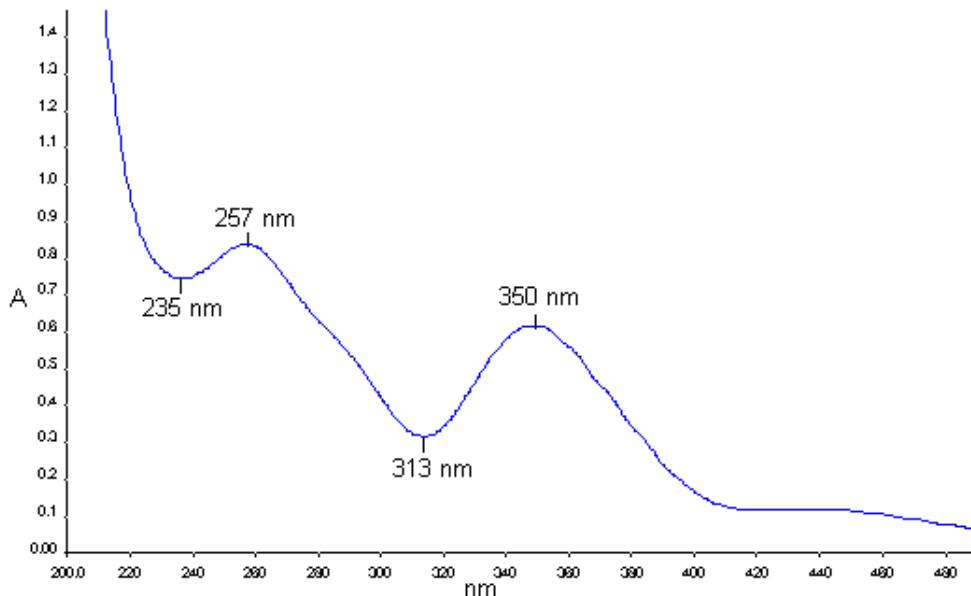
Photometric accuracy with Potassium Nitrate solutions

Photometric Accuracy K2Cr2O7 Solution Test

This test measures the photometric accuracy using the certified absorbance values at specified wavelengths of Potassium Dichromate ($K_2Cr_2O_7$) solution, against the limit specified. When the test is performed, if the result is below or equal the upper limit, or higher than or equal to the lower limit, the test is passed.

The solution from PerkinElmer's DAB reference material can be used to check photometric accuracy. The solutions are individually calibrated. They absorb over the entire wavelength range and so they are insensitive to stray radiation.

The spectrum below illustrates the spectral characteristics of potassium dichromate solution.



How do I define a Photometric accuracy K2Cr2O7 solution test?

1. Select **Photometric Accuracy K2Cr2O7Solution** from the list of tests in the Folder List on the IPV Setup page:



The Photometric Accuracy K2Cr2O7 Solution test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the reference material.
3. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

4. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Select the **Number of wavelengths** at which the photometric accuracy is to be tested. The default is 5. The minimum is 1 and the maximum is 10. The default wavelength values are displayed. These are dependent upon the Reference material selected. See the table below for further information.
6. Select the **Number of solutions** to be measured at each wavelength. The default is 2. The minimum is 1 and the maximum is 4.
7. Enter the **Reference serial numbers** for each of the solutions. The Reference serial number can be found on the certificate provided with the solution.
8. Enter the **Calibrated absorbances** for each of the solutions at each of the wavelengths. The values can be found on the certificate provided with the reference material. The number of values to be entered depends on the number of wavelengths and number of solutions. For example, if there are 4 wavelengths and 2 solutions, 8 values must be entered – as each solution has a value at each wavelength.
9. If you wish, enter the **Photometric accuracy limit** for each of the solutions. The minimum is 0.00 and the maximum is 10.00 A.
10. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
11. Enter the text to appear before the test is run.
12. Repeat steps 10 and 11 for the **Sample**, **Fail** and **Pass** prompts.
13. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
14. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
15. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the tree and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

16. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Photometric accuracy K2Cr2O7 solution test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Photometric Accuracy K2Cr2O7Solution** is not currently selected, it will appear in the **Available tests** list.

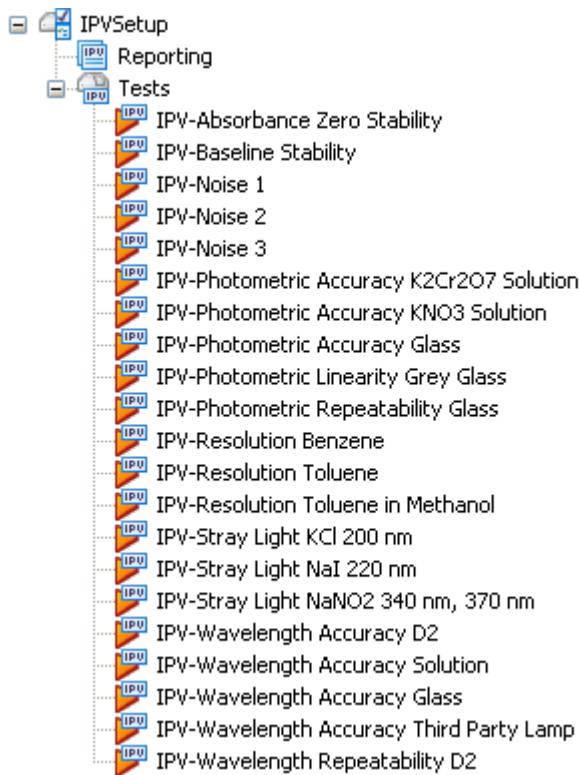
5. Select Photometric Accuracy K2Cr2O7Solution and then click Add.

Photometric Accuracy K2Cr2O7 Solution moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Photometric accuracy K2Cr2O7 solution from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Photometric Accuracy K2Cr2O7Solution in the Selected tests list and then click Remove.

Photometric Accuracy K2Cr2O7Solution is added to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (EP Potassium Dichromate in Sulphuric Acid, PerkinElmer Potassium Dichromate in Sulphuric Acid, PerkinElmer Potassium Dichromate in Perchloric Acid, NIST SRM 931 Liquid Filters, NIST SRM 935 Potassium Dichromate in Perchloric Acid)	EP Potassium Dichromate in Sulphuric Acid	–	–
Import reference material information	–	–	–
Reference material calibration date (from certificate)	–	–	–
Reference material re-calibration date (from certificate)	–	–	–

Warning X days before expiring	30	1	365
Number of solutions	2	1	4
Number of wavelengths	5	1	10
Reference serial numbers (number of serial numbers required is dependent on number of standards, max of 4)	–	–	–
X ₁ Wavelength (nm) (number dependent on number of wavelengths selected)	See table below	190	1100
Calibrated Absorbance (A) (from certificate)	–	0.0000	10.0000
Photometric Accuracy limits solution 1 (same for solutions 1 to 4) ($\pm A$)	This field is left blank in the software for you to enter your value. You must add the limit of the instrument (0.01 A) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0000	10.0000
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

What are the default wavelengths for EP Potassium Dichromate in Sulphuric Acid, PerkinElmer Potassium Dichromate in Sulphuric Acid, PerkinElmer Potassium Dichromate in Perchloric Acid, NIST SRM 931 Liquid Filters, and NIST SRM 935 Potassium Dichromate in Perchloric Acid?

Standards	Wavelengths (nm)	Standards
EP Potassium Dichromate in Sulphuric Acid	235, 257, 313, 350, 430	2
PerkinElmer Potassium Dichromate in Sulphuric Acid	235, 257, 313, 350	2
PerkinElmer Potassium Dichromate in Perchloric Acid	235, 257, 313, 350	2
NIST SRM 931 Liquid Filters	302, 395, 512, 678	4
NIST SRM 935 Potassium Dichromate in Perchloric Acid	235, 257, 313, 350	2

What are the fixed parameters for the test?

Slit width (nm)	1
Ordinate Mode	A
Response (secs)	5
Number of cycles	1
Cycle time (secs)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: The test uses a Wavelength program method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

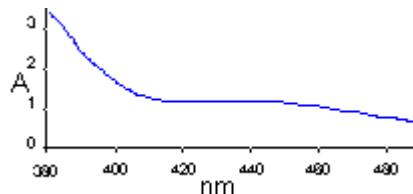
If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

Photometric Accuracy K2Cr2O7 Solution Test at 430 nm

This test measures the photometric accuracy using the certified absorbance values of Potassium Dichromate ($K_2Cr_2O_7$) solution at 430 nm, against the limit specified. When the test is performed, if the result is below or equal the upper limit and higher than or equal to the lower limit, the test is passed.

The spectrum below illustrates the spectral characteristics of potassium dichromate solution near 430 nm.



How do I define a Photometric accuracy K2Cr2O7 solution 430 nm test?

1. Select **Photometric Accuracy K2Cr2O7Solution 430 nm** from the list of tests in the Folder List on the IPV Setup page.



The Photometric Accuracy K2Cr2O7 Solution 430 nm test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the reference material.

3. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

4. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Select the **Number of wavelengths** at which the photometric accuracy is to be tested.

The default is 5. The minimum is 1 and the maximum is 10. The default wavelength value of 430 nm is displayed. This is dependent upon the Reference material selected. See the [table below](#) for further information.

6. Select the **Number of solutions** to be measured at each wavelength.
The default is 2. The minimum is 1 and the maximum is 4.
7. Enter the **Reference serial numbers** for each of the solutions.
The Reference serial number can be found on the certificate provided with the solution.
8. Enter the **Calibrated absorbances** for each of the solutions at each of the wavelengths.
The values can be found on the certificate provided with the reference material. The number of values to be entered depends on the number of wavelengths and number of solutions. For example, if there are 4 wavelengths and 2 solutions, 8 values must be entered – as each solution has a value at each wavelength.
9. If you wish, enter the **Photometric accuracy limit** for each of the solutions.
The minimum is 0.00 and the maximum is 10.00 A.
10. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
11. Enter the text to appear before the test is run.
12. Repeat steps 10 and 11 for the **Sample**, **Fail** and **Pass** prompts.
13. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
14. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
15. To define the next test, select the test from the list in the Folder List and enter the information as required.
OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the tree and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

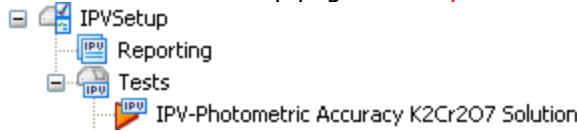
- The [Save or Save As dialog](#) is displayed.
16. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.
OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

1. Select **Photometric Accuracy K2Cr2O7Solution 430 nm** from the list of tests in the Folder List on the IPV Setup page – *Example*.



The Photometric Accuracy K2Cr2O7 Solution 430 nm test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the reference material.

The only reference material available for Photometric Accuracy K2Cr2O7Solution 430 nm is EP Potassium Dichromate in Sulphuric Acid.

3. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

4. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Select the **Number of wavelengths** at which the photometric accuracy is to be tested.

The default is 1. The minimum is 1 and the maximum is 10. The default wavelength value of 430 nm is displayed.

6. Select the **Number of solutions** to be measured.

The default is 1. The minimum is 1 and the maximum is 4.

7. Enter the **Reference serial numbers** for each of the solutions.

The Reference serial number can be found on the certificate provided with the solution.

8. Enter the **Calibrated absorbance** for the solution at 430 nm.

The value can be found on the certificate provided with the reference material.

9. If you wish, enter the **Photometric accuracy limit** for each of the solutions.

The minimum is 0.00 and the maximum is 10.00 A.

10. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

11. Enter the text to appear before the test is run.

12. Repeat steps 10 and 11 for the **Sample**, **Fail** and **Pass** prompts.

13. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
14. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
15. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the tree and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The [Save or Save As dialog](#) is displayed.

16. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Photometric accuracy K2Cr2O7 solution 430 nm test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or [create a new IPV setup](#) to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog.
The folder expands to show all the tests currently included in the IPV Setup.
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Photometric Accuracy K2Cr2O7Solution430 nm** is not currently selected, it will appear in the **Available tests** list.
5. Select Photometric Accuracy K2Cr2O7Solution 430 nm and then click Add.
Photometric Accuracy K2Cr2O7 Solution 430 nm moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List.

How do I remove Photometric accuracy K2Cr2O7 solution 430 nm test from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog.
The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.
2. Select Photometric Accuracy K2Cr2O7Solution 430 nm in the Selected tests list and then click Remove.
Photometric Accuracy K2Cr2O7Solution 430 nm is added to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material	EP Potassium Dichromate in Sulphuric Acid	–	–
Reference material calibration date (from certificate)	–	–	–
Reference material re-calibration date (from certificate)	–	–	–
Number of solutions	1	1	4
Number of wavelengths	1	1	10
Reference serial numbers (number of serial numbers required is dependent on number of standards, max of 4)	–	–	–
X ₁ Wavelength (nm) (number dependent on number of wavelengths selected)	430	190	1100
Calibrated Absorbance (A) (from certificate)	–	0.0000	10.0000

Photometric Accuracy limits solution 1 (same for solutions 1 to 4) (<u>+A</u>)	This field is left blank in the software for you to enter your value. You must add the limit of the instrument (0.01 A) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0000	10.0000
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Slit width (nm)	1
Ordinate Mode	A
Response (secs)	5
Number of cycles	1
Cycle time (secs)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: The test uses a Wavelength program method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select Days, Weeks or Months from the drop down list and then enter the appropriate value in the field to the left.

This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Photometric Accuracy with Potassium Nitrate Solution Test

This test measures the photometric accuracy using certified absorbance values at specified wavelengths of the potassium nitrate (KNO_3) solution, against the limit specified.

The test requires you to prepare a solution of potassium nitrate A.R. in distilled water and dilute this to give solutions with concentrations of 1.0650 %, 0.7100 % and 0.3550 % w/v.

When the test is performed, if the result is below or equal the upper limit, or higher than or equal to the lower limit, the test is passed.

How do I define a Photometric Accuracy with Potassium Nitrate Solution test?

1. Select **Photometric Accuracy with KNO3 Solution** from the list of tests in the Folder List on the IPV Setup page:



The Photometric Accuracy with KNO3 Solution test page is displayed. The page contains all the default settings.

2. Select or enter the name of the **Reference material**.
3. Enter the **Absorbance** value for each of the solutions.
4. If you wish to change the default **Photometric accuracy limit**, enter a new value in the field.
5. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
6. Enter the text to appear before the test is run.
7. Repeat steps 5 and 6 for the **Sample**, **Pass** and **Fail** prompts.
8. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
9. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
10. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

11. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Photometric Accuracy with Potassium Nitrate Solution test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

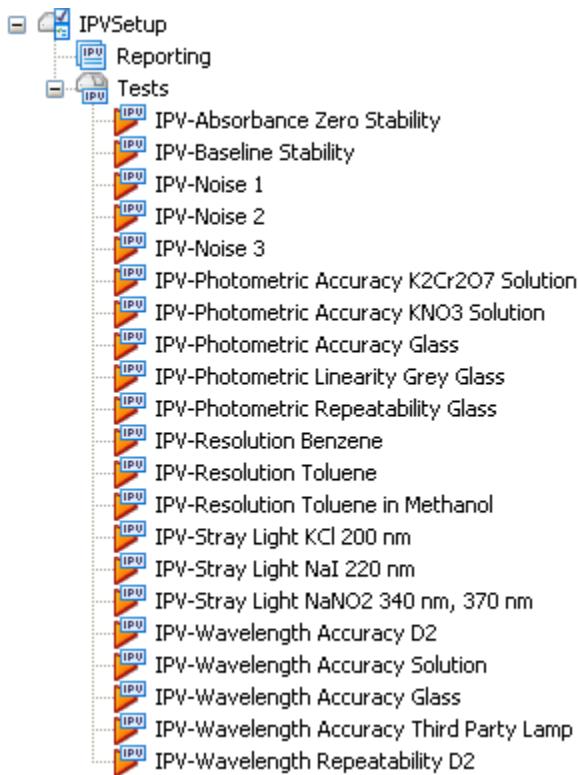
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Photometric Accuracy with KNO3 Solution** is not currently selected, it will appear in the **Available tests** list.

5. Select Photometric Accuracy with KNO3 Solution and then click Add.
Photometric Accuracy with KNO3 Solution moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Photometric Accuracy with Potassium Nitrate Solution from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Photometric Accuracy with KNO3 Solution in the Selected tests list and then click Remove.

Photometric Accuracy with KNO3 Solution moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Set absorbance (1.0650%) (A)	–	0.0000	10.0000
Set absorbance (0.7100%) (A)	–	0.0000	10.0000
Set absorbance (0.3550%) (A)	–	0.0000	10.0000

Photometric Accuracy limits (±A)	This field is left blank in the software for you to enter your value. You must add the limit of the instrument (0.01 A) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0000	10.0000
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What is the expected absorbance for each solution?

The expected absorbances are:

KNO ₃ (%)	Absorbance
1.0650	0.7510
0.7100	0.5000
0.3550	0.2500

What are the fixed parameters for the test?

Wavelength settings (nm)	302
Slit width (nm)	1
Ordinate Mode	A
Response (secs)	5
Number of cycles	1
Cycle time (secs)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Wavelength program method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Photometric Accuracy Glass

This test measures the photometric accuracy using certified absorbance values at specified wavelengths of a selected glass filter, against the limit specified.

The glass filters from PerkinElmer's set of NTRM spectrometric calibration reference material can be used to check photometric accuracy. The glass filters are individually calibrated. They absorb over the entire wavelength range and so they are insensitive to stray radiation. The influence of temperature on the transmission characteristics is small (-0.0003 Absorbance units per Kelvin). The influence of the spectral bandpass is $\pm 0.0005 \text{ A}$ in the range 0.5 nm to 4 nm slit.

When the test is performed, if the result is below or equal the upper limit, or higher than or equal to the lower limit, the test is passed.

How do I define a Photometric accuracy with glass test?

1. Select **Photometric Accuracy Glass** from the list of tests in the Folder List on the IPV Setup page—Example.



The **Photometric accuracy with glass** test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list of available reference materials.
3. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

4. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Select the Number of wavelengths.

The default is 5. The minimum is 1 and the maximum is 10.

6. Select the Number of filters.

The default is 1. The minimum is 1 and the maximum is 4.

7. Enter the **Reference serial numbers** for the filters.

8. If more than 5 wavelengths are specified, enter the additional wavelengths at which the test is performed.

9. Enter the **Calibrated absorbance** for each filter at each wavelength.
If there are 3 filters and 5 wavelengths, a total of 15 values will need to be entered.
10. Enter the **Photometric accuracy limit** of each filter.
11. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
12. Enter the text to appear before the test is run.
13. Repeat steps 11 and 12 for the **Sample**, **Fail** and **Pass** prompts.
14. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
15. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
16. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

17. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Photometric accuracy with glass test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.

4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Photometric accuracy with glass** is not currently selected, it will appear in the **Available tests** list.

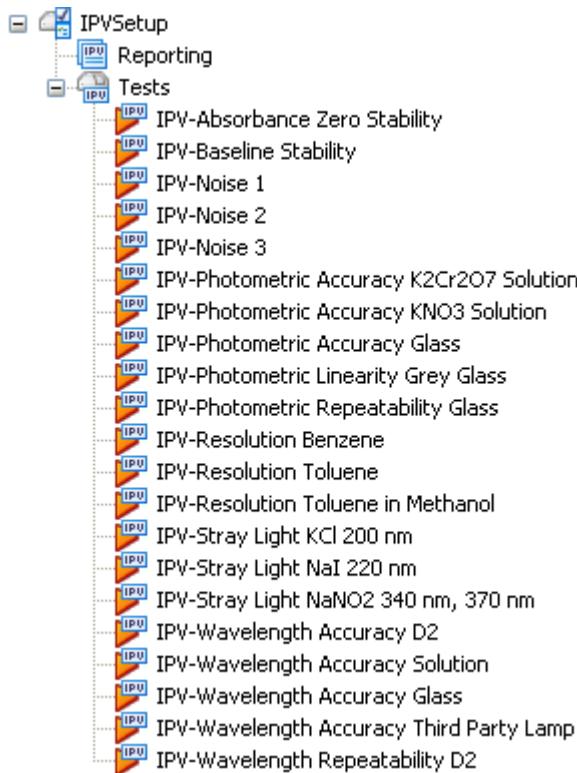
5. Select Photometric accuracy with glass and then click Add.

Photometric accuracy with glass moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Photometric accuracy with glass from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select **Photometric accuracy with glass** in the Selected tests list and then click **Remove**.

Photometric accuracy with glass moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer NTRM PKI 930 glass filters, PerkinElmer NTRM PKI 1930 glass filters, PerkinElmer Secondary Reference Material, NIST SRM 930 glass filters, NIST SRM 1930 glass filters, NIST SRM 2031 metal film on Quartz filters)	PerkinElmer NTRM PKI 930 glass filters	–	–
Calibration date (from certificate)	today	–	–
Re-calibration date (from certificate)	today	–	–
Number of wavelengths	5	1	10
Number of filters	1	1	4
Reference serial numbers (dependent on number of glass filters, max 4)	–	–	–
Wavelength (number dependent on number of wavelengths selected) (nm)	See table below	190	1100
Calibrated absorbance (A) (from certificate)	–	0.0000	10.0000
Photometric accuracy limits (repeated for filters 1 to 4) ($\pm A$)	This field is left blank in the software for you to enter your value. You must add the limit of the instrument (0.003 nm) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0000	10.0000
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

NOTE: The number of calibrated absorbances depends on number of filters and number of wavelengths. The maximum is 40 (10 wavelengths x 4 filters).

What are the available wavelengths for the different types of filters?

Filter	Wavelengths (nm)
NTRM PKI 930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
NTRM PKI 1930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
PerkinElmer Secondary Reference Material	440.00, 546.10, 635.00, 1700.00, 2300.00
NIST SRM 930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
NIST SRM 1930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
NIST SRM 2031	250.00, 280.00, 340.00, 360.00, 400.00, 465.00, 500.00, 546.10, 590.00, 635.00

What are the fixed parameters for the test?

Wavelengths (nm)	440
Slit width (nm)	1
Ordinate Mode	A
Response (secs)	5
Number of cycles	1
Cycle time (secs)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: The test uses a Wavelength program method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.

This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Photometric Linearity Grey Glass Test

Photometric linearity is a general check of the instrument's performance to confirm that a solution known to conform to the Beer–Lambert law will give a linear plot of absorbance versus concentration (A vs c) when measured by a spectrometer.

This test tests the photometric linearity using the certified absorbance values at specified wavelengths of a glass filter, against the limit specified. When the test is performed, if the result is below or equal the upper limit, or higher than or equal to the lower limit, the test is passed.

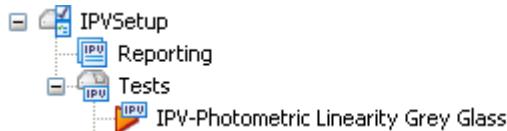
How do I validate the instrument's linearity?

You can validate the instrument's linearity using solutions of different concentrations or reference materials. During a PerkinElmer Validation Process check, glass reference materials (1 A + 2 A) are used. Measurements are taken for each glass reference material individually, then both reference materials are loaded into the sample holder at the same time. If the measurement for both reference materials together is the sum of the measurements for the reference materials individually, the instrument's response is linear.

If the response is not linear, additional checks should be performed on the instrument, particularly checking the instrument control setting (0%T / 100%T) and testing for stray radiation. It is not only the instrument performance which may limit linear measurements over a wide absorbance range. There are also some limits with certain samples. The sample itself must obey the Beer–Lambert law. Generally, the Beer–Lambert law is only valid for real and ideal solutions with no interactions between the dissolved sample molecules. The bigger the interaction between the dissolved molecules, the bigger is the deviation from the Beer–Lambert law. For example, association or dissociation in the sample may cause severe deviations.

How do I define a Photometric linearity with Grey Glass test?

1. Select **Photometric Linearity Grey Glass** from the list of tests in the Folder List on the IPV Setup page:



The Photometric linearity test page is displayed. The page contains all the default settings.

2. Select or enter the name of the **Reference material**.
3. If you wish, alter the default **Photometric linearity limit (A)**
The default is 0.02. The minimum is 0.00 and the maximum is 10.00 A.
4. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
5. Enter the text to appear before the test is run.
6. Repeat steps 4 and 5 for the **Sample,Pass** and **Fail** prompts.
7. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days, weeks or months** from the drop-down list.

8. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
9. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select IPV setup from the Folder List, and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

10. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

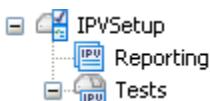
If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Photometric linearity with Grey Glass test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:

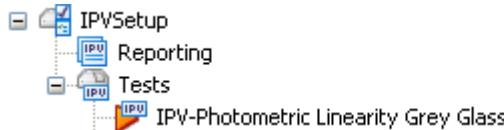


The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Photometric linearity with Grey Glass** is not currently selected, it will appear in the **Available tests** list.

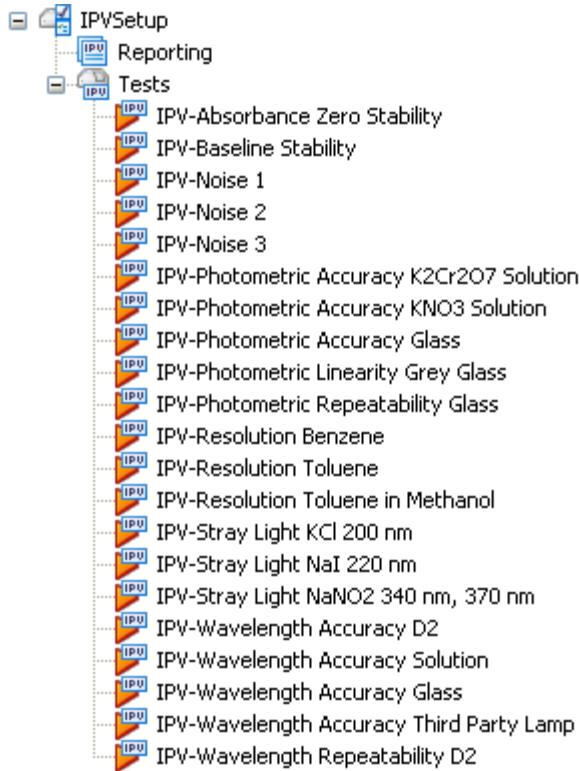
5. Select **Photometric linearity** and then click **Add**.

Photometric linearity with Grey Glass moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Photometric linearity with Grey Glass test from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Photometric linearity with Grey Glass in the Selected tests list and then click Remove.

Photometric linearity with Grey Glass is added to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material	–	–	–
Photometric linearity limits ($\pm A$)	0.020	0.0000	10.0000
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Wavelength Settings (nm)	546.1
Slit width (nm)	1
Ordinate Mode	A
Response (sec)	5
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Wavelength Program method.

What are other causes of non-linearity?

Stray radiation is not the only factor which influences linearity; linearity also depends on the characteristics of the detector. Photomultipliers as detectors have a much wider linear dynamic range than photodiodes. Spectrometers with a photomultiplier detector allow measurement over a wider absorbance range than spectrometers with diode detectors. In addition to this, you can use a reference beam attenuator in instruments with a photomultiplier; this is not possible in instruments with diode detectors. Nevertheless it is easily possible to measure up to about 4 Absorbance units with an instrument which has photodiodes as detector and respective low stray radiation. The linearity is somewhat dependent on the measurement wavelength, because stray light may be slightly different at different wavelengths.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Photometric Repeatability Glass Test

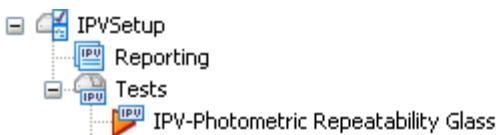
The photometric repeatability specifies the deviation of an absorbance value in repeated measurements. This parameter also has an influence on quantitative analysis and the comparability of photometric measurements. The smaller this deviation is, the more reproducible and comparable the results are. This is also called photometric reproducibility.

This test tests the photometric repeatability using certified absorbance values at specified wavelengths of a glass filter. The calculated standard deviation of the 10 measurements must be less than or equal to the limit specified for the test to pass.

Without good photometric repeatability, a spectrometer could not achieve photometric accuracy. If the instrument is to be trusted to produce reliable, reproducible data, a sample must read the same value repetitively (\pm the indicated tolerance). Photometric repeatability is specified in absorbance as a tolerance (e.g. $\pm 0.0002 \text{ A}$ at 1 A) and the absorbance level is stated as a condition of testing. As absorbance increases, photometric repeatability may deteriorate due to increased noise of the instrument at lower signal levels.

How do I define a Photometric repeatability test?

1. Select **Photometric repeatability glass** from the list of tests in the Folder List on the IPV Setup page:



The Photometric repeatability test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name.
The Serial number is on the certificate provided with the Reference material.
3. Enter the **Serial number** of the Reference material.
The Calibration date can be found on the certificate provided with the reference material.
Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.
4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .
The Re-calibration date can be found on the certificate provided with the reference material.
Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.
5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

6. Select the **Number of wavelengths** to be used in the test.

The number of wavelengths will be given on the certificate provided with the reference material.

The default is 5. The minimum is 1 and the maximum is 5. The default wavelength values are displayed.

7. Enter the **Calibrated absorbances** for the reference material.

The values can be found on the certificate provided with the Reference material.

8. If you wish, alter the default Photometric standard deviation limit (A)

The default is 0.001 A

9. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

10. Enter the text to appear before the test is run.

11. Repeat steps 10 and 11 for the **Sample**, **Pass** and **Fail** prompts.

12. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.

13. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.

14. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

15. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Photometric repeatability test to the list of tests I want to perform?

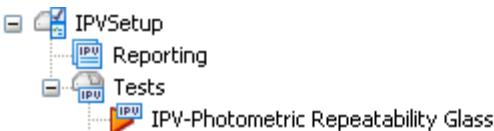
1. Select Instruments from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the Tests folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup

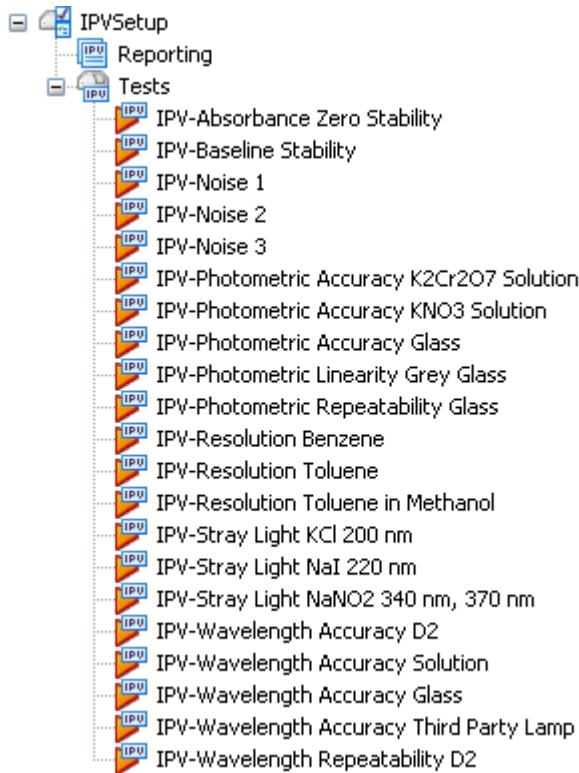
The IPV setup dialog displays the list of **Available tests** and **Selected tests**. If **Photometric Repeatability** is not currently selected, it will appear in the **Available tests** list.

5. Highlight Photometric repeatability and then click Add.
Photometric repeatability moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Photometric repeatability test from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Photometric repeatability in the Selected tests list and then click Remove.
- Photometric repeatability** is added to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer NTRM PKI 930 glass filters, PerkinElmer NTRM PKI 1930 glass filters, PerkinElmer Secondary Reference Material, NIST SRM 930 glass filters, NIST SRM 1930 glass filters, NIST SRM 2031 metal film on quartz filters)	PerkinElmer NTRM PKI 930 glass filters	–	–
Number of wavelengths	5	1	5
Serial number (from certificate)	–	–	–
Calibration date (from certificate)	(today)	–	–
Re-calibration date (from certificate)	(today)	–	–

Wavelength (nm)	See 'What are the available wavelengths for different types of filters'	190	1100
Calibrated absorbance (A) (from certificate)	–	0.0000	10.0000
Photometric standard deviation limit (A)	< 0.001 You must add the limit of the instrument (0.01 A) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0000	10.0000
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

What are the available wavelengths for the different types of filters?

Filter	Wavelengths (nm)
NTRM PKI 930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
NTRM PKI 1930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
PerkinElmer Secondary Reference Material	440.00, 546.10, 635.00, 1700.00, 2300.00
NIST SRM 930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
NIST SRM 1930 Glass Filters	440.00, 465.00, 546.10, 590.00, 635.00
NIST SRM 2031 metal film on Quartz filters	250.00, 280.00, 340.00, 360.00, 400.00

What are the fixed parameters for the test?

Wavelength (nm)	440
Slit width (nm)	1
Ordinate Mode	A
Time interval (sec)	5
Total time (sec)	45
Response (sec)	5
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: The test uses a Timedrive method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Resolution

The resolution in a spectral analysis depends upon the bandpass (spectral bandwidth) of the spectrometer, which in turn depends primarily on the physical slit width. The smaller the slit width, the higher the resolution of bands in a spectrum that can be achieved.

What is bandpass?

The bandpass is defined as the full width at half maximum of the radiation beam emerging from the monochromator. It is proportional to the mechanical slit width at high values. It decreases less rapidly than the mechanical slit at low values.

The bandpass of the instrument is of considerable importance, because it is an indication of the width of the spectrum passing through the sample. An instrument with a bandpass of 1 nm detects energy in a much narrower portion of the spectrum than an instrument with a 20 nm bandpass. While a 20 nm bandpass is sometimes usable in the visible region where chromophores exhibit absorption bands as wide as 100 nm or more, it is unacceptable in the UV region where absorption bands range from 60 nm to fine structures 1 nm wide. Furthermore, it is known that to yield absorbance values of a chromophore to within a desired accuracy of the theoretical absorptivity, the spectral bandpass of a spectrometer must be one-tenth or less of the width of the absorption peak measured at half peak height.

The instrument's bandpass should be 1/5 to 1/10 of the full width at half peak height of the band to be measured. Higher values can cause loss of both photometric accuracy and resolution of detail.

The energy decreases as the mechanical slit width squared. This causes increased noise at high resolution (that is, low bandpass).

There are three ways to check resolution:

Using benzene vapor

Using toluene solution

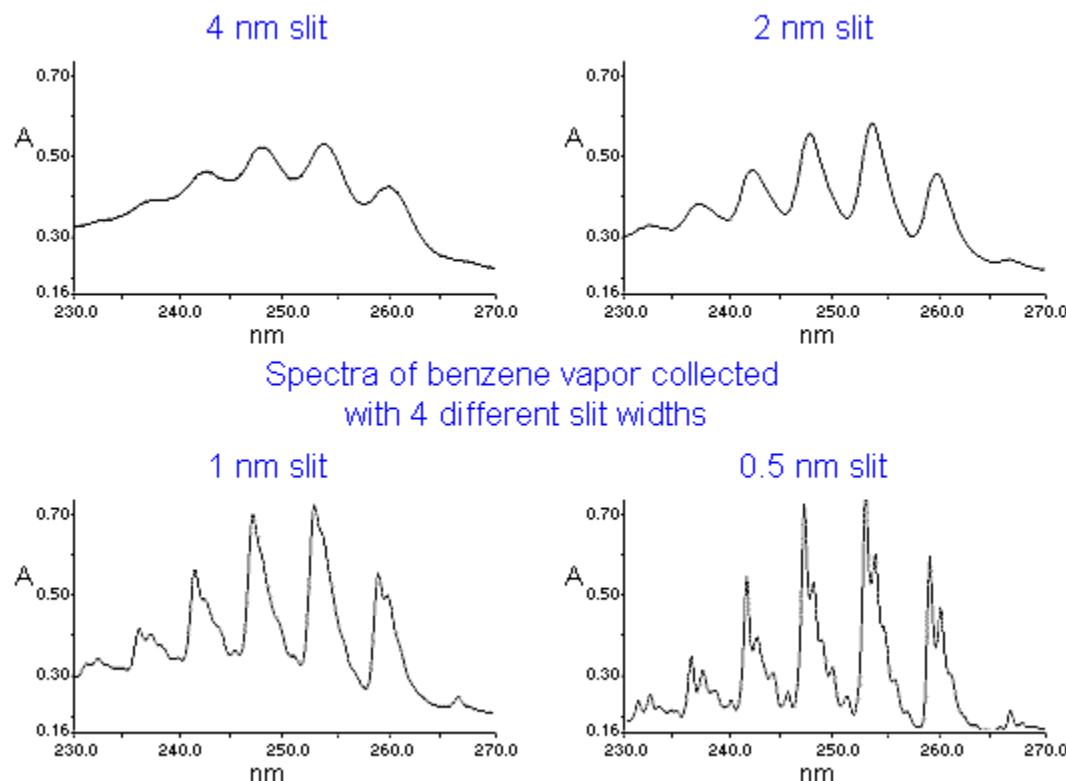
Using toluene in methanol.

Resolution with Benzene Vapor Test

This test tests the resolution using a benzene vapor sealed cell. This test is a visual assessment of the quality of the spectra and requires you to state whether the results are within specification.

To test the resolution, add one drop of benzene to a quartz cell and close the cell with a stopper.

As can be seen from the benzene spectra below, too high a slit width decreases both photometric accuracy and the resolution of spectral detail. Too low a slit width may increase noise. Under conditions of high energy, decreasing the slit causes only a small increase in noise. At low energy, the noise is inversely proportional to the slit width at large and medium slit widths. At very small slits, the noise increases faster than 1/slit width. A good compromise is usually made with a slit between 1/5 and 1/10 of the widths of the spectral features of interest.



How do I define a Resolution with Benzene Vapor test?

1. Select **Resolution Benzene** from the list of tests in the Folder List on the IPV Setup page:



The Resolution with Benzene Vapor test page is displayed. The page contains all the default settings.

2. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
3. Enter the text to appear before the test is run.
4. Repeat steps 2 and 3 for the **Sample**, **Pass** and **Fail** prompts.
5. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
6. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
7. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

8. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Resolution with Benzene Vapor test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Resolution Benzene** is not currently selected, it will appear in the **Available tests** list.

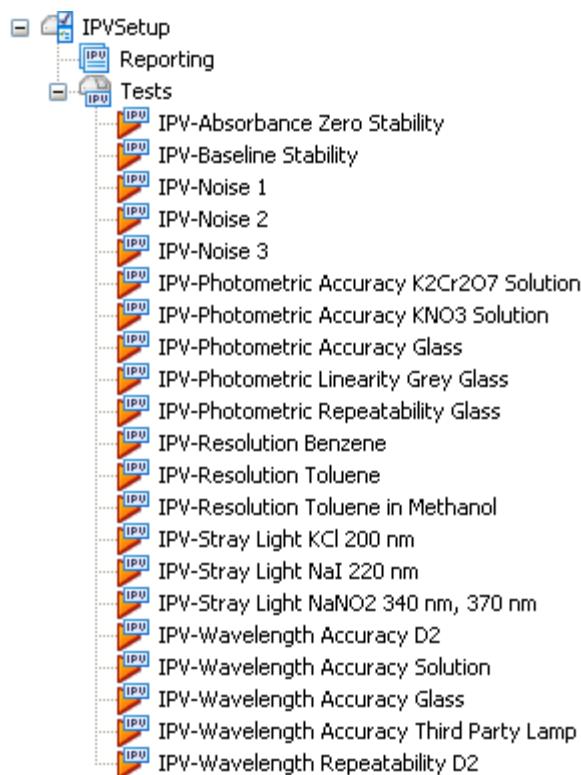
5. Select **Resolution Benzene** and then click **Add**.

Resolution Benzene moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Resolution with Benzene Vapor from the list of tests I want to perform?

1. Select the Tests folder on the Folder List within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV setup dialog displays the list of **Available tests** and **Selected tests**.

2. Highlight **Resolution Benzene** in the **Selected tests** list and then click **Remove**.
- Resolution Benzene** moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Perform every X days	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Start (nm)	270
End (nm)	230
Slit width (nm)	0.5, 1.0, 2.0, 4.0
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: Only a 1nm slit is used when performing the tests on a Lambda 25.

NOTE: An Autozero is performed between scans. This test uses a scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

How do I confirm if the test has passed or failed?

This test is a visual assessment of the quality of the spectra and requires you to state whether the results are within specification. When the test is complete, the spectrum is displayed in a dialog together with a **Pass** button and a **Fail** button. Make a visual inspection of the graph and then click **Pass** or **Fail** as required.

See also

[IPV Setup](#)

[Running an IPV](#)

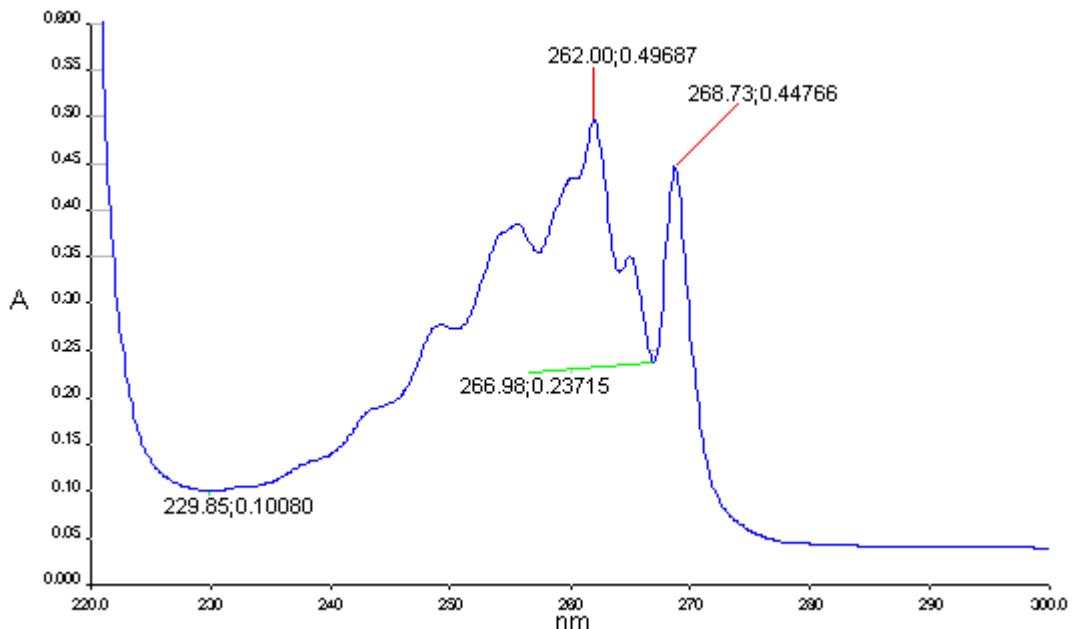
Resolution with Toluene Solution Test

This test tests the resolution using a toluene in hexane solution at a selected slit width. The calculated peak/valley ratio must be greater than the corresponding theoretical ratio for the test to pass.

Pharmacopoeia regulations state that only spectrometers which give good enough resolution of a toluene spectrum can be used for pharmaceutical applications. It is recommended in the literature that the absorbance ratio at 269 nm/266 nm is greater than 1.5.

The test solution is a 0.02% solution of toluene in hexane (v/v), in a 1 cm quartz cell. Alternatively, the PerkinElmer UV/Vis standard for EP/DAB may be used, which contains a ready-to-use toluene solution according to EP/DAB.

The spectrum below is of a 0.02% toluene solution in hexane recorded with slit width 1 nm. There is a maximum at 268.7 nm and a minimum at 266.9 nm. The respective absorbance readings are 0.4476 and 0.2371. The absorbance ratio of maximum and minimum is $0.4476/0.2371 = 1.89$



How do I define a Resolution with Toluene test?

1. Select **Resolution Toluene** from the list of tests in the Folder List on the IPV Setup page:



The Resolution with Toluene test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the Reference material.

3. Enter the **Serial number** of the Reference material.
This can be found on the certificate provided with the reference material.
4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .
The Calibration date can be found on the certificate provided with the reference material.
Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.
5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .
The Re-calibration date can be found on the certificate provided with the reference material.
Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.
6. Select which slit widths are to be used.
A check mark indicates the slit width is selected. The default is 1.0 nm.
7. For each selected slit width, enter the **Calibrated ratio**.
The Calibrated ratio can be found on the certificate provided with the reference material. The theoretical ratio is provided for reference.
8. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
9. Enter the text to appear before the test is run.
10. Repeat steps 8 and 9 for the **Pass** and **Fail** and **Sample** prompts.
11. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
12. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
13. To define the next test, select the test from the list in the tree and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the tree and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

14. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Resolution with Toluene Solution test to the list of tests I want to perform?

1. Select Instruments from the tree, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the Tests folder on the Tree within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup

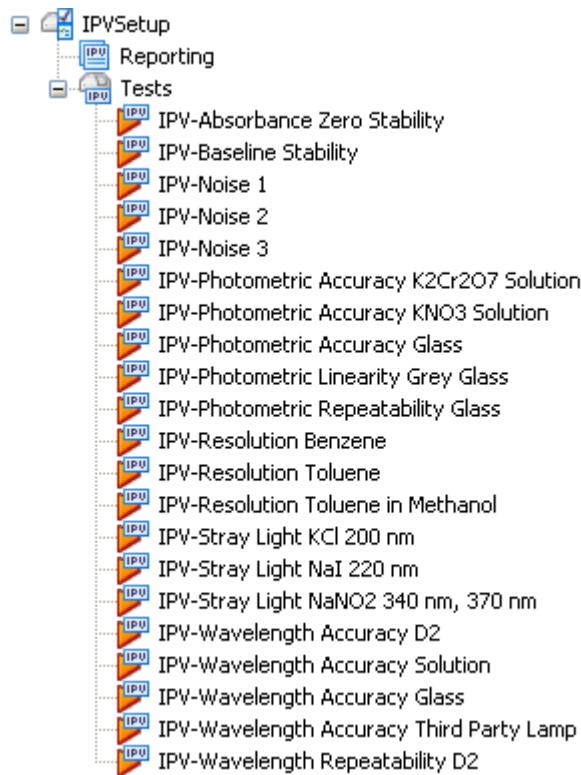
The IPV setup dialog displays the list of **Available tests** and **Selected tests**. If **Resolution Toluene** is not currently selected, it will appear in the **Available tests** list.

5. Highlight Resolution with Toluene Solution and then click Add.
Resolution Toluene moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Tree:



How do I remove Resolution with Toluene Solution from the list of tests I want to perform?

1. Select the Tests folder on the Tree within the IPV setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV setup dialog displays the list of **Available tests** and **Selected tests**.

2. Highlight Resolution Toluene test in the Selected tests list and then click Remove.

Resolution Toluene to the **Available tests** list. It is also removed from the Tests folder in the Tree.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material	PerkinElmer Toluene in Hexane	–	–
Serial number (from certificate)	–	–	–
Calibration date (from certificate)	–	–	–
Re-calibration date (from certificate)	–	–	–
Slit (0.5 and/or 1.0 and/or 2.0) (nm)	1 nm	0.5 nm	2.0 nm

Calibration ratio 0.5 nm (from certificate)	–	0.0	10.0
Calibration ratio 1.0 nm (from certificate)	–	0.0	10.0
Calibration ratio 2.0 nm (from certificate)	–	0.0	10.0
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

NOTE: If a slit width is specified that is not compatible with those available for a particular spectrometer, an error message will be displayed when you try to run the test.

What are the fixed parameters for the test?

Start (nm)	305
End (nm)	265
Slit width (nm)	0.5, 1.0, 2.0
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: Only a 1nm slit is used when performing the tests on a Lambda 25.

NOTE: If a slit width is specified that is not compatible with those available for a particular spectrometer, an error message will be displayed when you try to run the test.

NOTE: An Autozero is performed between scans. This test uses a scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

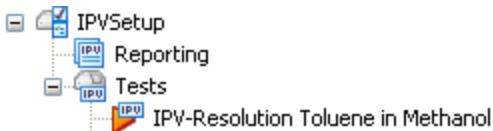
Resolution with Toluene in Methanol Test

This test tests the resolution using a toluene in methanol solution at the selected slit width. The calculated peak/valley ratio must be greater than the corresponding theoretical ratio for the test to pass.

Pharmacopoeia regulations state that only spectrometers which give good enough resolution of a toluene in methanol spectrum can be used for pharmaceutical applications. The test solution is a 0.02 % solution of toluene in methanol (v/v), in a 1 cm quartz cell. The spectrum shows a small negative peak at 265 nm located between two large negative peaks at 261 nm and 268 nm. It is recommended in the literature that the absorbance ratio at 265 nm / 261 nm is not less than 0.2.

How do I define a Resolution with Toluene in Methanol test?

1. Select **Resolution Toluene in Methanol** from the list of tests in the Folder List on the IPV Setup page:



The Resolution with Toluene in Methanol test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the Reference material.
3. Enter the **Serial number** of the Reference material.

This can be found on the certificate provided with the reference material.

4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. Select which **Slit widths** are to be used.

A check mark indicates the slit width is selected. The default is 0.5 nm.

7. For each selected slit width, enter the **Minimum Ratio**.

8. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

9. Enter the text to appear before the test is run.
10. Repeat steps 8 and 9 for the **Sample**, **Fail** and **Pass** prompts.
11. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
12. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
13. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

14. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Resolution with Toluene in Methanol test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
 2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
- The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
 4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Resolution Toluene in Methanol** is not currently selected, it will appear in the **Available tests** list.

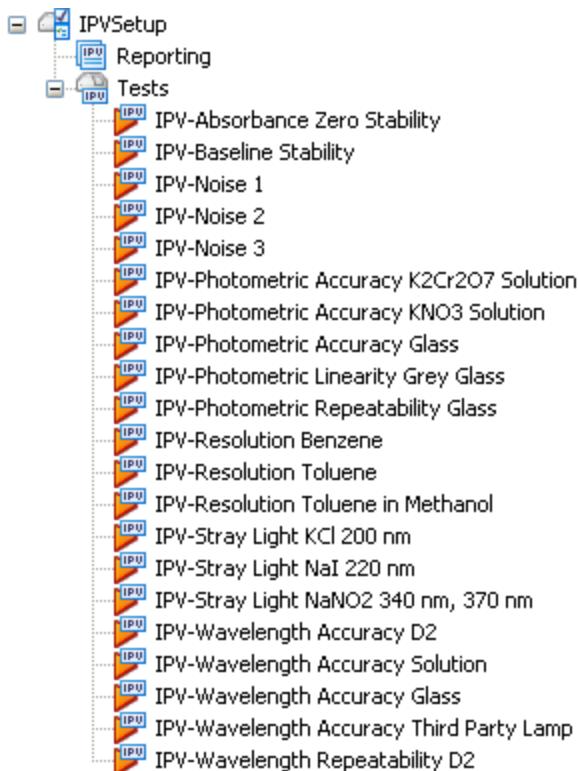
5. Select Resolution with Toluene in Methanol and then click Add.

Resolution Toluene in Methanol moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Resolution with Toluene in Methanol from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Resolution Toluene in Methanol in the Selected tests list and then click Remove. **Resolution Toluene in Methanol** moves to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material	Toluene in Methanol	–	–
Serial number (from certificate)	–	–	–
Calibration date (from certificate)	–	–	–
Re-calibration date (from certificate)	–	–	–
Slit (0.5 and/or 1.0 and/or 2.0) (nm)	0.5 nm	0.5 nm	2.0 nm
Minimum Ratio 0.5 nm	0.0	0.0	10.0
Minimum Ratio 1.0 nm	0.0	0.0	10.0
Minimum Ratio 2.0 nm	0.0	0.0	10.0
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

NOTE: If a slit width is specified that is not compatible with those available for a particular spectrometer, an error message will be displayed when you try to run the test.

What are the fixed parameters for the test?

Start (nm)	290
End (nm)	230
Slit width (nm)	0.5, 1.0, 2.0
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	326

NOTE: Only a 1 nm slit is used when performing the tests on a Lambda 25.

NOTE: If a slit width is specified that is not compatible with those available for a particular spectrometer, an error message will be displayed when you try to run the test.

NOTE: An Autozero is performed between scans. This test uses a scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Stray Light

Stray radiation is any unwanted radiant energy measured by the detector that is outside the spectral region isolated by the bandpass of the instrument. Stray radiation is that portion of the light reaching the detector that is not of the desired wavelength, given the position of the monochromator.

$$\%T \text{ observed} = \frac{(\%T \text{ true} + \text{stray radiation}) \times 100}{(100 + \text{stray radiation})}$$

What is the effect of stray radiation?

A high amount of stray radiation can lead to severe limitations in linearity. In cases where stray radiation is present and not absorbed by the sample, the absorbance reading may be too low. This effect is much smaller at low absorbance values. With increasing absorbance, the stray radiation becomes more dominant and the readings become too low. In quantitative analysis at higher absorbance values, the result is strongly dependent on the influence of stray radiation.

The stray radiation specifications consequently have a direct influence on the calibration curve in quantitative analysis and on the precision of the results.

For example, assume that the instrument's wavelength is set to the peak of an absorption band, but that 0.1% of the radiation in the beam is another wavelength which is not absorbed by the sample. Consider 4 samples whose actual peak absorbances are 1, 2, 3, and 4. The observed %T will be 0.001 greater than the actual in each case due to the 0.1% stray radiation.

The table below illustrates the effect of 0.1% stray radiation on the transmission and absorbance readings of a sample with a real absorbance of 1, 2, 3, 4 A.

Actual peak absorbance	Actual %T at peak	Observed %T at peak	Observed peak absorbance
1	10.00	10.10	0.996
2	1.00	1.10	1.959
3	0.10	0.20	2.699
4	0.01	0.11	2.959

The following table illustrates the effect of different amounts of stray radiation.

Actual absorbance	Absorbance readings from instruments with different % stray radiation specifications				
	0.001 %	0.02%	0.03%	0.05%	0.1%
1.000	0.99996	0.9991	0.9994	0.9978	0.996
2.000	1.9995	1.991	1.987	1.979	1.959
3.000	2.996	2.921	2.886	2.824	2.699

How do I measure stray radiation?

The amount of stray radiation energy varies throughout the spectral range and between instruments, depending on the age of the spectrometer and on environmental conditions. It is therefore important that the stray radiant energy check is done at several points of the spectrum, particularly in regions where the best performance is required.

Stray radiation is measured with cutoff filters at a wavelength where they do not transmit light. To allow proper stray radiation measurement these filters should have a wide region where they transmit (stray) light. Filters with continuous high absorbance are not suited because they also block most of the stray radiation and give unrealistic low stray radiation values.

A general method to make sure an instrument is within the stray radiation tolerance for a specific application is the stray radiation test according to DAB 10/EP and other pharmacopoeias.

See also

Four stray light tests are available:

Stray light with potassium chloride solution at 200 nm

Stray light with potassium chloride solution

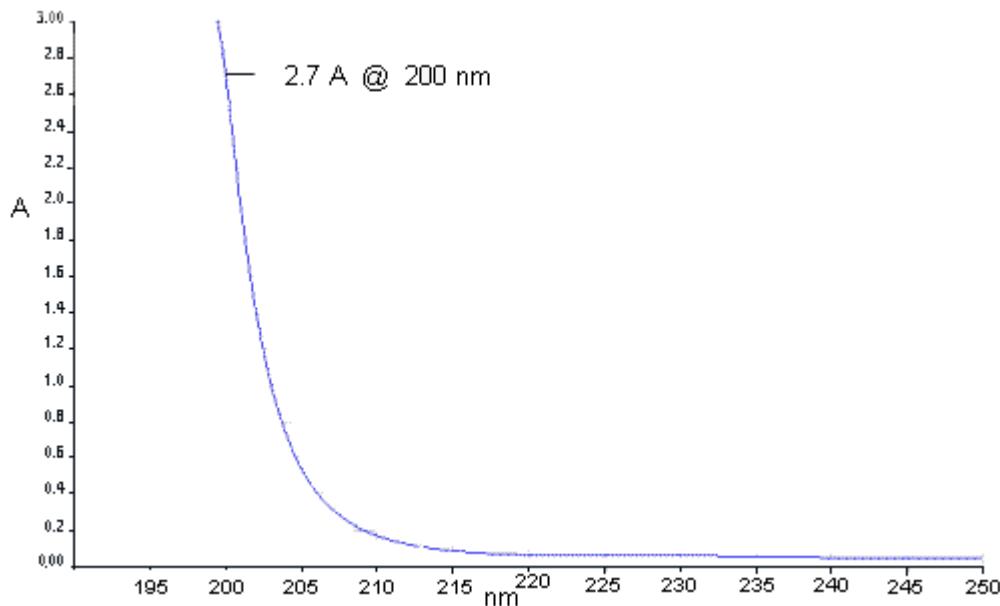
Stray light with sodium iodide solution

Stray light with sodium nitrite solution

Stray Light with Potassium Chloride Solution at 200 nm Test

This test tests the stray light at 200 nm using a potassium chloride solution (12g/l). The measured absorbance must be greater than or equal to 2 A. When the test is performed, if the result is equal to, or below the upper limit, the test passes.

The spectrum below is of a KCl solution (12 g/l). It shows the cut-off at 200 nm.



How do I define a Stray light with Potassium Chloride solution at 200 nm test?

1. Select **Stray Light KCl 200 nm** from the list of tests in the Folder List on the IPV Setup page:



The Stray Light with Potassium Chloride test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the Reference material.
3. Enter the **Serial number** of the Reference material and water.

This can be found on the certificate provided with the reference material.

4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. Enter the Calibrated absorbance (A).

The value can be found on the certificate provided with the reference material.

7. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

8. Enter the text to appear before the test is run.

9. Repeat steps 7 and 8 for the **Sample,Pass** and **Fail** prompts.

10. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.

11. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.

12. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

13. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

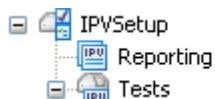
If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Stray light with Potassium Chloride Solution at 200 nm test to the list of tests I want to perform?

1. Select **Instruments** from the tree, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Tree within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Stray Light KCl 200 nm** is not currently selected, it will appear in the **Available tests** list.

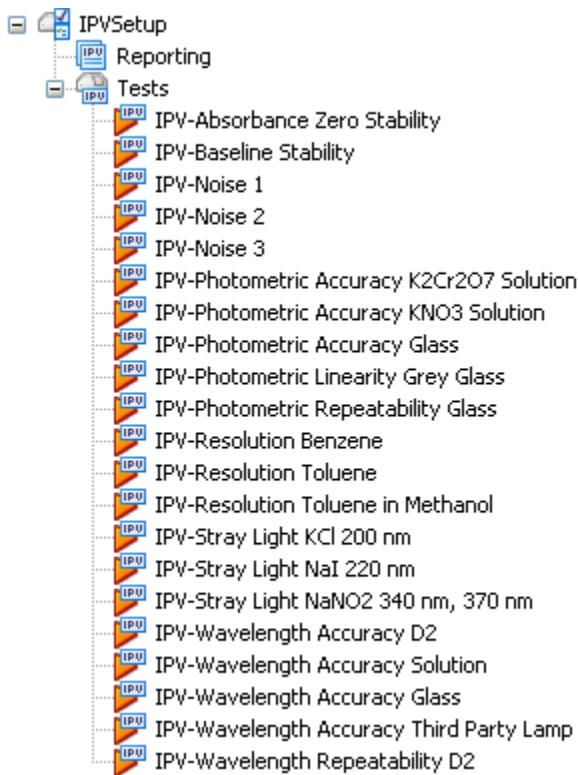
5. Select Stray Light KCl 200 nm and then click Add.

Stray Light KCl 200 nm moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Tree:



How do I remove Stray light with Potassium Chloride solution at 200 nm from the list of tests I want to perform?

1. Select the **Tests** folder on the Tree within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Stray Light KCl 200 nm in the Selected tests list and then click Remove.

Stray Light KCl 200 nm to the **Available tests** list. It is also removed from the Tests folder in the Tree.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer Stray Light Reference Material – Potassium Chloride)	PerkinElmer Stray Light Reference Material – Potassium Chloride	–	–
Serial number (KCl)	–	–	–
Serial number (H ₂ O)	–	–	–
Calibration date (from certificate)	(today)	–	–
Re-calibration date (from certificate)	(today)	–	–

Calibrated absorbance (from certificate)	-	2.0	5.0
Stray light limit (A)	> 2	2.0	5.0
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Start (nm)	202
End (nm)	198
Slit width (nm)	1.0
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

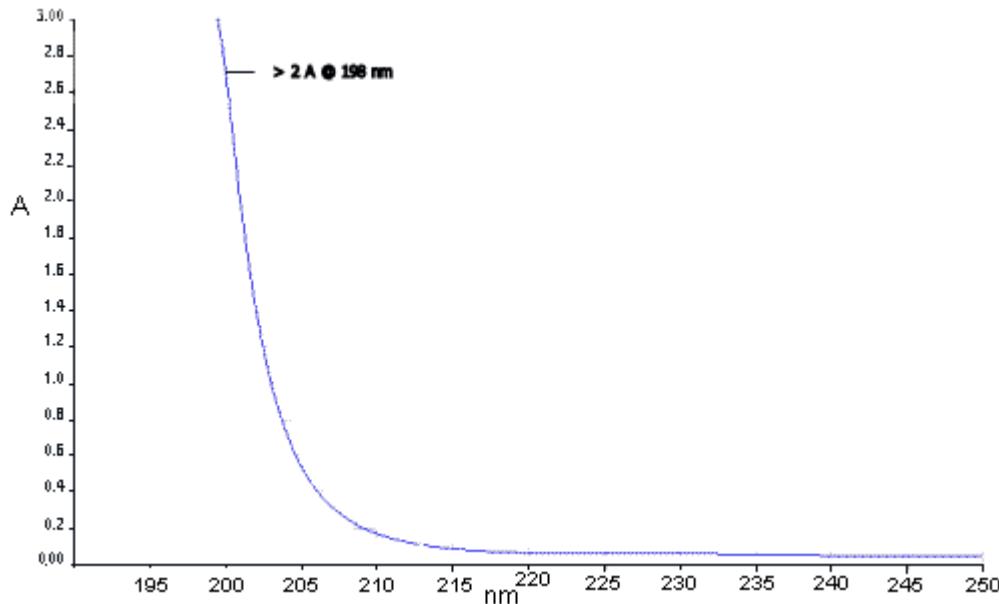
[IPV Setup](#)

[Running an IPV](#)

Stray Light with Potassium Chloride Solution Test

This test tests the stray light at a wavelength you define using a potassium chloride solution (12g/l). The measured absorbance must be greater than or equal to 2 A. When the test is performed, if the result is equal to, or above the limit, the test passes.

The spectrum below is of a KCl solution (12 g/l). It shows the absorbance is greater than 2.0 at 198 nm.



How do I define a Stray light with Potassium Chloride solution test?

1. Select **Stray Light KCl** from the list of tests in the Folder List on the IPV Setup page:



The Stray Light with Potassium Chloride test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the Reference material.

3. Enter the **Serial number** of the Reference material and water.

This can be found on the certificate provided with the reference material.

4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. In the **Measurement (nm)** field, enter the wavelength that you wish to perform the test.
7. Enter the Calibrated absorbance (A).
The value can be found on the certificate provided with the reference material.
8. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
9. Enter the text to appear before the test is run.
10. Repeat steps 8 and 9 for the **Sample**, **Fail** and **Pass** prompts.
11. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
12. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
13. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

14. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Stray light with Potassium Chloride Solution test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

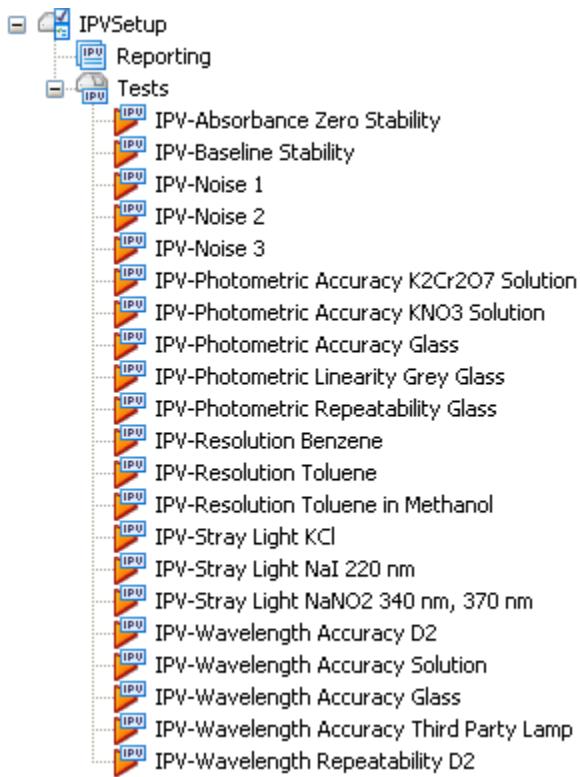
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Stray Light KCl** is not currently selected, it will appear in the **Available tests** list.

5. Select **Stray Light KCl** and then click **Add**.
Stray Light KCl moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Stray light with Potassium Chloride solution from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select **Stray Light KCl** in the **Selected tests** list and then click **Remove**.
Stray Light KCl moves to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer Stray Light Reference Material – Potassium Chloride)	PerkinElmer Stray Light Reference Material – Potassium Chloride	-	-
Serial number (KCl)	-	-	-
Serial number (H ₂ O)	-	-	-
Calibration date (from certificate)	(today)	-	-
Re-calibration date (from certificate)	(today)	-	-
Measurement (nm)	-		

Calibrated absorbance (from certificate)	-	2.0	5.0
Stray light limit (A)	2.000	2.000	5.000
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Start (nm)	202
End (nm)	194
Slit width (nm)	1.0
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

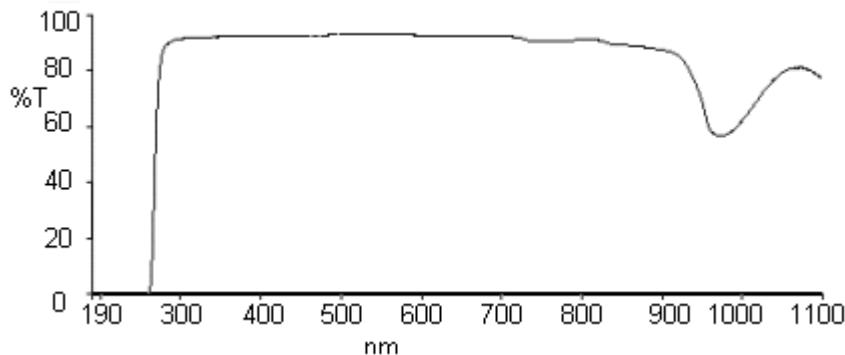
[Running an IPV](#)

Stray light with Sodium Iodide (NaI) Solution at 220 nm Test

Sodium Iodide (NaI) absorbs very strongly at wavelengths below 260 nm and is completely transparent through the rest of the spectrum up to 1100 nm. Consequently, an aqueous solution of 10 g/l NaI can be used to measure stray radiation between 210 nm and 259 nm.

This test tests the stray light at 220 nm using sodium iodide solution. The measured transmission difference between the blocked beam and the sodium iodide solution must be greater than or equal to the specified limit.

The spectrum below is of an aqueous solution of NaI 10 g/l. It illustrates the cut-off point at 260 nm.



How do I define a Stray light with Sodium Iodide solution test?

1. Select **Stray Light NaI 220 nm** from the list of tests in the Folder List on the IPV Setup page:



The Stray light with Sodium Iodide test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the Reference material.
3. Enter the Serial number.
This can be found on the certificate provided with the reference material.

4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. Enter the Calibrated transmission (%T).

The value can be found on the certificate provided with the reference material.

7. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

8. Enter the text to appear before the test is run.

9. Repeat steps 7 and 8 for the **Sample**, **Block**, **Pass** and **Fail** prompts.

10. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.

11. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.

12. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

13. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

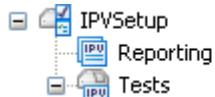
If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Stray light with Sodium Iodide Solution test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Stray Light NaI 220 nm** is not currently selected, it will appear in the **Available tests** list.

5. Select Stray Light NaI 220 nm and then click Add.
Stray Light NaI 220 nm moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Stray light with Sodium Iodide solution from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:
2. Select Stray Light NaI 220 nm in the Selected tests list and then click Remove.
Stray Light NaI 220 nm moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer Stray Light Reference Materials – Sodium Iodide)	PerkinElmer Stray Light Reference Material – Sodium Iodide	–	–
Serial number (from certificate)	–	–	–
Calibration date (from certificate)	–	–	–
Re-calibration date (from certificate)	–	–	–
Calibrated Transmission %T (from certificate)	–	2.0	5.0
Stray light limit <(%T) (reference only)	< 0.01 %T	0.000000 %T	1.000000 %T
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Wavelength (nm)	220
Slit width (nm)	1
Total Time (sec)	60
Time Interval (sec)	1
Ordinate Mode	%T
Response (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Timedrive method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

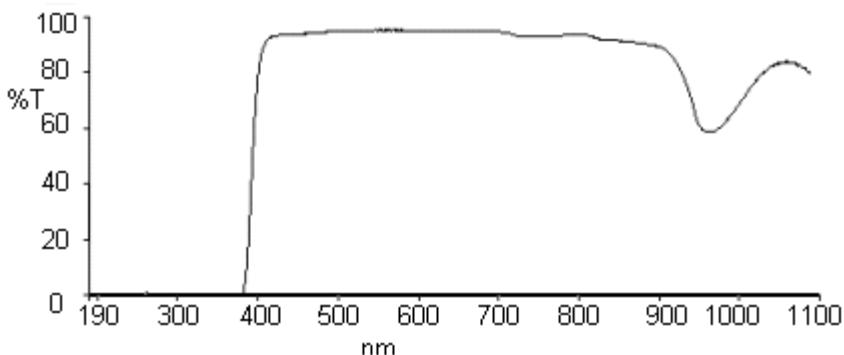
[Running an IPV](#)

Stray light with Sodium Nitrite (NaNO_2) at 340 nm and 370 nm Test

Sodium Nitrite solution absorbs very strongly in the UV range of the spectrum between 300 nm and 385 nm. It is completely transparent for all radiation of the remaining spectrum up to 1100 nm. Consequently, a 50 g/l NaNO_2 solution can be used to measure stray radiation between these wavelengths.

This test uses a Timedrive method to test the stray light of the instrument at both 340 nm and 370 nm using sodium nitrite solution. The measured transmission difference between the blocked beam and the solution must be greater than or equal to the specified limit. When the test is performed, if the result is equal to, or below the upper limit, the test passes.

The spectrum below is of a 50 g/l NaNO_2 solution. It illustrates the cut-off point at 385 nm.



How do I define a Stray light with Sodium Nitrite solution test?

1. Select **Stray Light NaNO_2 340 nm, 370 nm** from the list of tests in the Folder List on the IPV Setup page:



The Stray light with Sodium Nitrite test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the Reference material.
3. Enter the Serial number.

This can be found on the certificate provided with the reference material.

4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. Enter the Calibrated transmission (%T) at 340 nm and 370 nm.

The values can be found on the certificate provided with the reference material.

7. If you wish to change the default **Stray light limit**, enter a new value in the field.

8. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

9. Enter the text to appear before the test is run.

10. Repeat steps 8 and 9 for the **Sample**, **Block**, **Pass** and **Fail** prompts.

11. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.

12. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.

13. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

14. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Stray light with Sodium Nitrite Solution test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

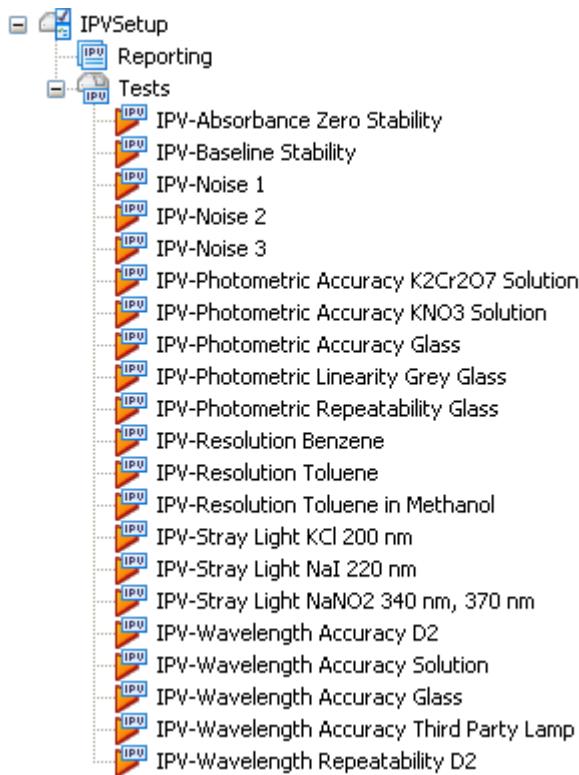
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Stray Light NaNO₂ 340 nm, 370 nm** is not currently selected, it will appear in the **Available tests** list.

5. Select Stray Light NaNO₂ 340 nm, 370 nm and then click Add.
Stray Light NaNO₂ 340 nm, 370 nm moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Stray light with Sodium Nitrite solution from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Stray Light NaNO2 340 nm, 370 nm in the Selected tests list and then click Remove.

Stray Light NaNO2 340 nm, 370 nm to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer Stray Light Reference Materials – Sodium Nitrite or user-defined)	PerkinElmer Stray Light Reference Material – Sodium Nitrite	-	-
Reference material serial number (from certificate)	-	-	-
Reference material calibration date (from certificate)	-	-	-
Reference material re-calibration date (from certificate)	-	-	-

Calibrated Transmission 340 nm (from certificate)	-	0.000000 %T	1.000000 %T
Calibrated Transmission 370 nm (from certificate)	-	0.000000 %T	1.000000 %T
Stray light limit (default)	< 0.01 %T	0.000000 %T	1.000000 %T
Perform every X days	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Wavelengths (nm)	340 and 370
Slit width (nm)	1.0
Total Time (sec)	60
Time Interval (sec)	1.0
Ordinate Mode	A
Response (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Timedrive method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Wavelength Accuracy

Wavelength accuracy is the closeness of the wavelength value reported by the instrument to the actual value, determined by comparing measured and known values of sharp lines. It defines how precisely a selected wavelength can be set by a spectrometer. If a wavelength calibration is specified as ± 0.5 nm at 656.1 nm, its wavelength accuracy is not as good as an instrument specified at ± 0.1 nm. If a spectrometer had a wavelength accuracy of ± 0.1 nm, the maximum transmittance might be found anywhere between 656.0 nm and 656.2 nm.

Wavelength accuracy is not as critical for spectrometers functioning in the region 400 nm to 800 nm, where wide-band chromophores are encountered. However, when working in the UV region where sharp absorption bands are often encountered, wavelength accuracy becomes much more critical.

The following Wavelength Accuracy tests are available within UV WinLab:

Wavelength Accuracy Check with Deuterium lamp

Wavelength Accuracy with Solution

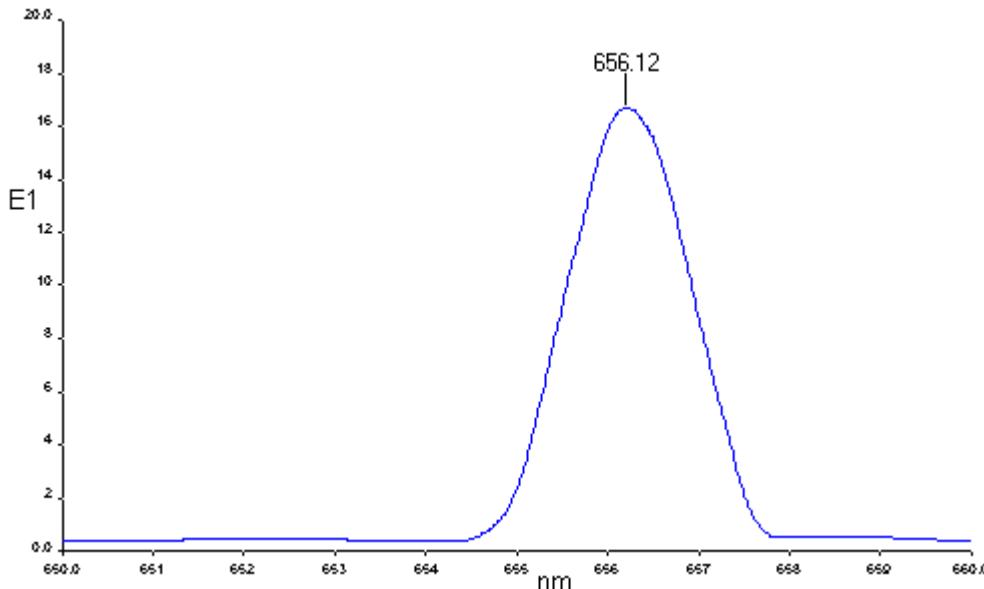
Wavelength Accuracy with Third-party lamp

Wavelength Accuracy with Glass Filters

Wavelength Accuracy with Deuterium Lamp Test

Wavelength accuracy specification is commonly measured with the Deuterium peak (D₂ peak) of the UV lamp. The D₂ peak has a very sharp maximum at 656.1 nm and is therefore well suited to testing the wavelength accuracy in that range.

The deuterium lamp wavelength calibration check tests the wavelength accuracy using the Deuterium (D₂) lamp emission line at 656.1 nm and/or 486 nm, against the limit specified. When the test is performed, if the result is below or equal to the upper limit, or higher than or equal to the lower limit, the test is passed.



The spectrum below is obtained from a Lambda 35 (2 nm slit setting). The D₂ peak is at 656.1 nm.

How do I define a Wavelength accuracy with Deuterium lamp test?

1. Select **Wavelength accuracy D2** from the list of tests in the Folder List on the IPV Setup page:



The Wavelength accuracy with Deuterium lamp test page is displayed. The page contains all the default settings.

2. Select the Number of wavelengths.

The default (and maximum) is 2. The minimum is 1. The measurement wavelengths are displayed for reference; they cannot be edited.

3. If you wish to change the default **Wavelength accuracy limit**, enter a new value in the field.
4. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

5. Enter the text to appear before the test is run.
6. Repeat steps 4 and 5 for the **Pass** and **Fail** prompts.
7. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
8. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
9. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

10. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

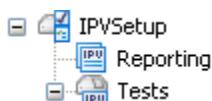
If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Wavelength accuracy with Deuterium lamp test to the list of tests I want to perform?

11. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
12. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
13. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
14. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Wavelength accuracy D2** is not currently selected, it will appear in the **Available tests** list.

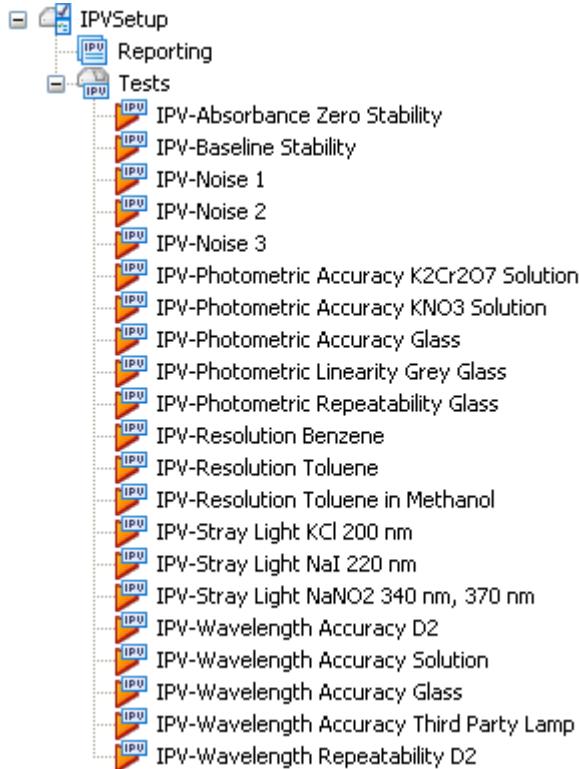
15. Select Wavelength accuracy D2 and then click Add.

Wavelength accuracy D2 moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Wavelength accuracy with Deuterium lamp from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Wavelength accuracy D2 in the Selected tests list and then click Remove.

Wavelength accuracy D2 moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Number of wavelengths	2	1	2
Measurement wavelength (nm)	656.1	–	–
Measurement wavelength (nm)	486.0	–	–

Wavelength Accuracy Limit (nm) +/-	+/- 0.3	0.0	10
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Start (nm)	658.1
End (nm)	654.1
Slit width (nm)	1
Data Interval (nm)	0.1
Ordinate Mode	E2
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	Off
Lamp change at (nm)	319.2

NOTE: There is no Autozero energy scan. There is no reference material for this test.

NOTE: This test uses a Scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Wavelength Accuracy Check with Solution Test

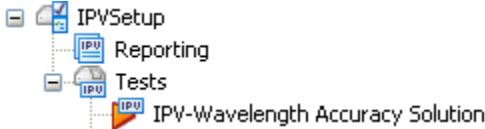
Solutions are available as a ready-to-use reference material from PerkinElmer, NIST, or other organization; for example Holmium perchlorate from PerkinElmer. The exact positions of the glass peaks vary slightly for each production batch. Therefore, the reference materials are calibrated individually.

The influence of temperature on the peak positions is negligible in the range 20–30 °C. The influence of the spectral bandpass (slit) on the peak positions is also negligible in the range 1–2 nm for the peaks selected for calibration.

This test tests the wavelength repeatability using the emission line of the Deuterium (D_2) lamp at 656.1 nm. The standard deviation of the 10 measurements must be less than or equal to the limit specified. When the test is performed, if the result is below or equal the upper limit, or higher than or equal the lower limit, the test is passed.

How do I define a Wavelength accuracy with solutions test?

1. Select **Wavelength accuracy solution** from the list of tests in the Folder List on the IPV Setup page:



The Wavelength accuracy with glass filters test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the reference material.
The Serial number is on the certificate provided with the Reference material.
3. Enter the **Serial number** of the Reference material.
The Serial number is on the certificate provided with the Reference material.
4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .
The Calibration date can be found on the certificate provided with the reference material.
Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.
5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .
The Re-calibration date can be found on the certificate provided with the reference material.
Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. Select the **Number of wavelengths** to be used in the test.

The number of wavelengths will be given on the certificate provided with the reference material.

The default is 4. The minimum is 1 and the maximum is 10. The default wavelength values are displayed.

7. Enter the **Calibrated wavelengths** for the reference material.

The values can be found on the certificate provided with the Reference material.

8. Enter the default **Wavelength accuracy limit** for each of the solutions.

The minimum is 0.00 and the maximum is 10.00 A.

9. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.

10. Enter the text to appear before the test is run.

11. Repeat steps 9 and 10 for the **Sample**, **Pass** and **Fail** prompts.

12. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.

13. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.

14. To define the next test, select the test from the list in the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

15. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Wavelength accuracy with solutions test to the list of tests I want to perform?

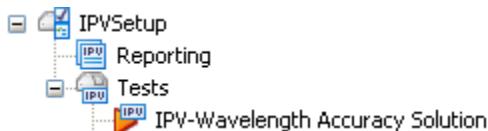
1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

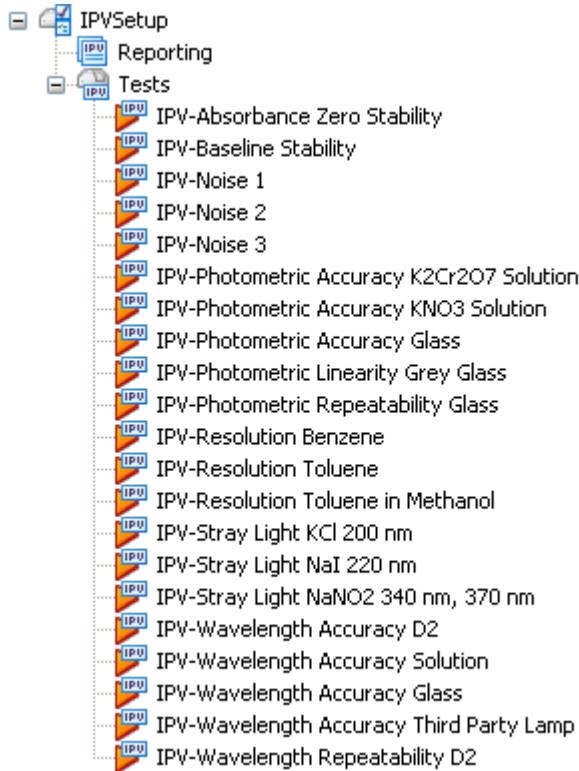
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Wavelength accuracy solution** is not currently selected, it will appear in the **Available tests** list.

5. Select Wavelength accuracy solution and then click Add.
Wavelength accuracy solution moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Wavelength accuracy with solutions test from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Wavelength accuracy solution in the Selected tests list and then click Remove. **Wavelength accuracy solution** to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer Holmium Perchlorate solution, NIST SRM 2034 Holmium Perchlorate solution or user-defined)	PerkinElmer Holmium Perchlorate solution	—	—
Serial number (from certificate)	—	—	—
Calibration date (from certificate)	(today)	—	—
Re-calibration date (from certificate)	(today)	—	—

Wavelength accuracy limit (nm) +/-	This field is left blank in the software for you to enter your value. You must add the limit of the instrument (0.25 nm) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0	10.0
Select how many lines need to be tested	4	1	10
Calibrated wavelength (nm) (from certificate) [number of these options depends on how many lines to be tested (4 by default)]	-	190 nm	1100 nm
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

NOTE: An Autozero is performed between scans.

What are the fixed parameters for the test?

Start (nm)	243.1
End (nm)	239.1
Slit width (nm)	1
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1
Cycle Time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.

This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Wavelength Accuracy with Glass Filters Test

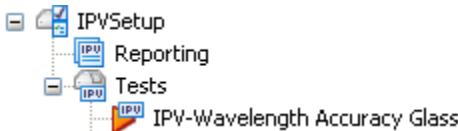
Glass filters are available from PerkinElmer, NIST, or other organizations as ready-to-use reference materials, for example Holmium Oxide glass. The exact positions of the glass peaks vary slightly for each production batch. Therefore, the glass filters are calibrated individually.

The influence of temperature on the peak positions is negligible in the range 20–30 °C. The influence of the spectral bandpass (slit) on the peak positions is also negligible in the range 1–2 nm for the peaks selected for calibration.

This test tests the wavelength accuracy using peaks of a selected glass filter (for example, Holmium Oxide Glass at 287.3, 360.9, 459.0 and 540.1 nm), against the limit specified. When the test is performed, if the result is below or equal the upper limit, or higher than or equal the lower limit, the test is passed.

How do I define a Wavelength accuracy with glass filters test?

1. Select **Wavelength Accuracy Glass** from the list of tests in the Folder List on the IPV Setup page:



The Wavelength accuracy with glass filters test page is displayed. The page contains all the default settings.

2. Select the **Reference material** from the drop-down list or enter the name of the reference material.

3. Enter the **Serial number** of the Reference material.

The Serial number is on the certificate provided with the Reference material.

4. Enter the **Calibration date** of the reference material using the calendar displayed by clicking .

The Calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

5. Enter the **Re-calibration date** of the reference material using the calendar displayed by clicking .

The Re-calibration date can be found on the certificate provided with the reference material.

Use the left and right arrow buttons on the calendar to navigate through the months and then click on the correct day. The calendar closes and the Calibration date is updated.

6. Select the **Number of wavelengths** to be used in the test.

The number of wavelengths will be given on the certificate provided with the reference material.

The default is 4. The minimum is 1 and the maximum is 10. The default wavelength values are displayed.

7. Enter the **Calibrated wavelengths** for the reference material.
The values can be found on the certificate provided with the Reference material.
8. If you wish, alter the default **Wavelength accuracy limit** for each of the solutions.
The minimum is 0.00 and the maximum is 10.00 A.
9. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
10. Enter the text to appear before the test is run.
11. Repeat steps 9 and 10 for the **Sample**, **Pass** and **Fail** prompts.
12. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
13. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
14. To define the next test, select the test from the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

15. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Wavelength accuracy with glass filters test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.

4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Wavelength Accuracy Glass** is not currently selected, it will appear in the **Available tests** list.

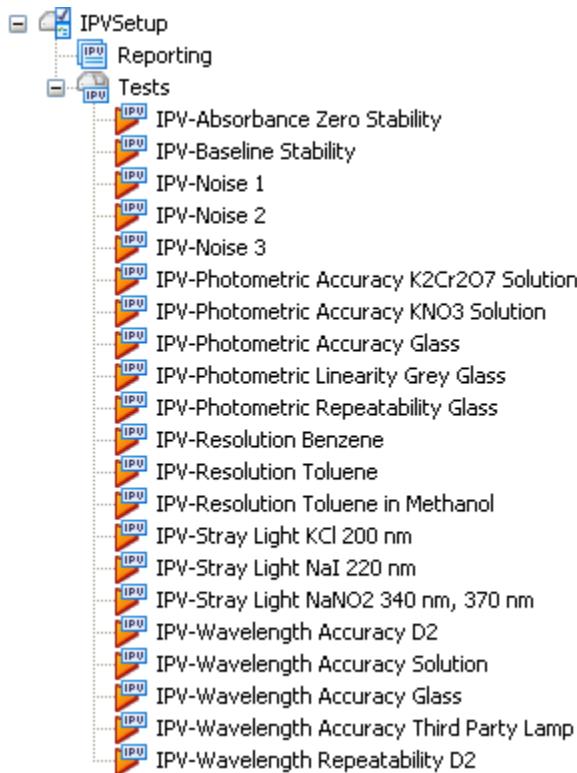
5. Select Wavelength Accuracy Glass and then click Add.

Wavelength Accuracy Glass moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List—Example



How do I remove Wavelength accuracy with glass filters from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Wavelength Accuracy Glass in the Selected tests list and then click Remove.

Wavelength Accuracy Glass moves to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference material (PerkinElmer Holmium Oxide Filter)	PerkinElmer Holmium Oxide Filter	–	–
Serial number (from certificate)	–	–	–
Calibration date (from certificate)	(today)	–	–
Re-calibration date (from certificate)	(today)	–	–
Number of wavelengths	4	1	10
Calibrated wavelengths (from certificate) [number of these options depends on how many lines to be tested (4 by default)]	–	190 nm 1100 nm	
Wavelength accuracy limit (nm) +/-	This field is left blank in the software for you to enter your value. You must add the limit of the instrument (0.25 nm) to the tolerance of the reference material (this value can be found on the certificate provided with the reference material). Enter the sum of these two values in this field.	0.0 nm	10.0 nm
Perform every (days)	7	1	365
Stop tests on failure	On	–	–

NOTE: An Autozero is performed between scans.

What are the fixed parameters for the test?

Start (nm)	281.3
End (nm)	277.3
Slit width (nm)	1
Data Interval (nm)	0.1
Ordinate Mode	A
Scan Speed (nm/min)	30
Number of cycles	1

Cycle time (sec)	1
UV Lamp	On
Visible Lamp	On
Lamp change at (nm)	319.2

NOTE: This test uses a Scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days**, **Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Wavelength Accuracy with Third-Party Lamp Test

Wavelength accuracy is commonly measured with the mercury lamp.

This test tests the wavelength accuracy using specified emission lines of a third party lamp (for example, a Mercury lamp at 253.70, 302.25, 313.16, 334.15, 365.48, 404.66, 435.83, 546.07, 576.96 and 579.07 nm), against the limit specified. When the test is performed, if the result is below or equal the upper limit, or higher than or equal the lower limit, the test is passed.

NOTE: This test is not available if a cell changer is installed. The test must always be run manually.

How do I define a Wavelength accuracy with third-party lamp test?

1. Select **Wavelength Accuracy Third Party Lamp** from the list of tests in the Folder List on the IPV Setup page:



The Wavelength accuracy with third-party lamp test page is displayed. The page contains all the default settings.

2. Select the **Reference lamp** from the drop-down list or enter the name of the reference lamp.
3. Enter the **Serial number** of the Reference lamp.
4. Select the **Number of wavelengths** to be used in the test.
The number of wavelengths will be given on the certificate provided with the reference material.
The default is 10. The minimum is 1 and the maximum is 10.
5. Enter the Calibrated wavelengths.
6. If you wish, alter the Wavelength accuracy limit.
The minimum is 0.00 and the maximum is 10.00 A.
7. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
8. Enter the text to appear before the test is run.
9. Repeat steps 7 and 8 for the **Pass**, **Fail** and **Post test** prompts.
10. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
11. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.

12. To define the next test, select the test from the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time. The Save or Save As dialog is displayed.

13. If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Wavelength accuracy with third-party lamp test to the list of tests I want to perform?

1. Select **Instruments** from the Folder List, and then click (to select) on the instrument whose performance you wish to verify.
2. Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog is displayed.
3. Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
4. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Wavelength Accuracy Third Party Lamp** is not currently selected, it will appear in the **Available tests** list.

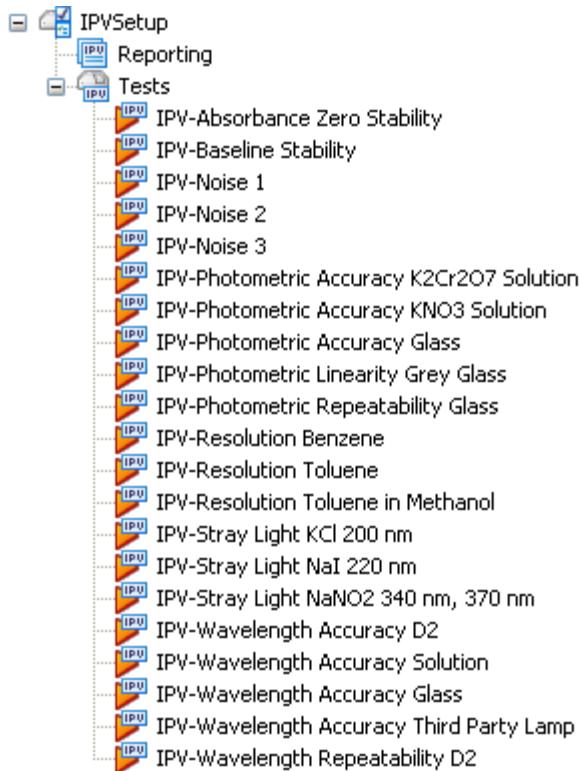
5. Select **Wavelength Accuracy Third Party Lamp** and then click **Add**.

Wavelength Accuracy Third Party Lamp moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Folder List:



How do I remove Wavelength accuracy with third-party lamp test from the list of tests I want to perform?

1. Select the **Tests** folder on the Folder List within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Wavelength Accuracy Third Party Lamp in the Selected tests list and then click Remove.

Wavelength Accuracy Third Party Lamp is added to the **Available tests** list. It is also removed from the Tests folder in the Folder List.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Reference Lamp	Mercury lamp	–	–
Serial number	–	–	–
Number of wavelengths	10	1	10
Calibrated Wavelength (nm)	253.70	190	1100
Calibrated Wavelength (nm)	302.25	190	1100
Calibrated Wavelength (nm)	313.16	190	1100
Calibrated Wavelength (nm)	334.15	190	1100
Calibrated Wavelength (nm)	365.48	190	1100
Calibrated Wavelength (nm)	404.66	190	1100
Calibrated Wavelength (nm)	435.83	190	1100
Calibrated Wavelength (nm)	546.07	190	1100
Calibrated Wavelength (nm)	576.96	190	1100
Calibrated Wavelength (nm)	579.07	190	1100
Wavelength Accuracy limit (nm) +/-	+/- 0.1	0.0	10.0
Perform every X days	7	1	365
Stop tests on failure	On	–	–

What are the fixed parameters for the test?

Start (nm)	255.7
End (nm)	251.7
Slit width (nm)	1
Data Interval (nm)	0.1
Ordinate Mode	E2
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1

UV Lamp	Off
Visible Lamp	Off
Lamp change at (nm)	319.2

NOTE: There is no Autozero energy scan.

NOTE: This test uses a scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks** or **Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

[IPV Setup](#)

[Running an IPV](#)

Wavelength Repeatability D2 Test

Wavelength repeatability is a measure of the reproducibility of the measured wavelength value. Together with wavelength accuracy, repeatability is an important factor in the accuracy of quantitative photometric analysis. The repeatability indicates the precision of the reading of a specific peak at the same wavelength on the same sample.

When you wish to make measurements at a given wavelength it is important that each time the instrument reproduces the same wavelength setting (\pm a small tolerance). Failure to do so could decrease the sensitivity of the test and possibly include an error, yielding false results.

This test tests the wavelength repeatability using the emission line of the deuterium lamp at 656.1 nm. The standard deviation of the 10 measurements must be less than or equal to the limit specified.

How do I define a Wavelength Repeatability D2 test?

1. Select **Wavelength Repeatability D2** from the list of tests in the Folder List on the IPV Setup page:



The Wavelength Repeatability test page is displayed. The page contains all the default settings.

NOTE: The calibrated wavelength is displayed for reference only and cannot be edited.

2. If you wish, enter a new value for the **Standard deviation**.
The default is < 0.05, the minimum is 0.00 and the maximum is 10.00.
3. If you wish to alter the default message, click in the **Prompt** field for the **Pre-Test** prompt.
4. Enter the text to appear before the test is run.
5. Repeat steps 3 and 4 for the **Pass** and **Fail** prompts.
6. Select how often you wish the test to be performed. Enter a value (the default is 7), and then select **days**, **weeks** or **months** from the drop-down list.
7. If you do not want the tests to stop when a test fails, clear **Stop tests on failure**.
8. To define the next test, select the test from the Folder List and enter the information as required.

OR

If you have defined all the tests for the IPV setup, select the IPV Setup page from the Folder List and click **Save** (if you have previously named and saved the setup), or **Save As** if you are saving the setup for the first time.

The Save or Save As dialog is displayed.

- If you selected **Save**, select a **Reason** from the drop-down list, and if required enter a **Comment**.

OR

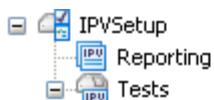
If you selected **Save As**, enter a name for the setup and click **OK**. On the next dialog, select a **Reason** from the drop-down list, and if required enter a **Comment**.

NOTE: If you are using the Standard version of UV WinLab, clicking **Save** will save the test without asking for a reason or comment. Selecting **Save As** will only display the Save As dialog. The additional dialog described above is for the Enhanced Security version only.

NOTE: If you Cancel the Save or Save As dialog, the Setup changes are not saved.

How do I add Wavelength Repeatability D2 test to the list of tests I want to perform?

- Select **Instrument Performance Verification** from the Tools menu within the Explorer, and from the submenu select **Create IPV Setup**.
The IPV Setup dialog shows all the tests currently included in the IPV Setup.
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Wavelength Repeatability D2** is not currently selected, it will appear in the **Available tests** list.
- Select the **Name** of a previously saved IPV setup that includes this test or create a new IPV setup to include this test.
- Select the **Tests** folder on the Tree within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup.

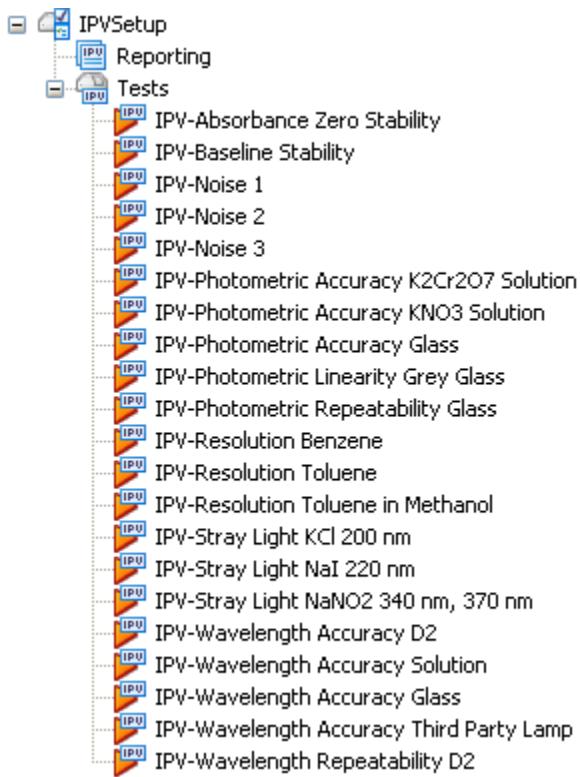
The IPV Setup dialog displays the list of **Available tests** and **Selected tests**. If **Wavelength Repeatability D2** is not currently selected, it will appear in the **Available tests** list.

- Select Wavelength Repeatability D2 and then click Add.
Wavelength Repeatability D2 moves from the **Available tests** list to the **Selected tests** list. It is also added to the Tests folder in the Tree:



How do I remove Wavelength Repeatability D2 from the list of tests I want to perform?

1. Select the **Tests** folder on the Tree within the IPV Setup dialog:



The folder expands to show all the tests currently included in the IPV Setup. The IPV Setup dialog displays the list of **Available tests** and **Selected tests**.

2. Select Wavelength Repeatability in the Selected tests list and then click Remove.

Wavelength Repeatability moves from the **Selected tests** list to the **Available tests** list. It is also removed from the Tests folder in the Tree.

What are the variable parameters and the default, minimum and maximum values?

Parameter	Default	Min	Max
Calibrated Wavelength (nm)	656.1	N/A	N/A
Standard deviation (nm)	< 0.05	0.000	10.000
Perform every (days)	7	1	365
Stop tests on failure	On	-	-

What are the fixed parameters for the test?

Start (nm)	658.1
End (nm)	654.1
Slit width (nm)	1
Data Interval (nm)	0.1
Ordinate Mode	E2
Scan Speed (nm/min)	30
Number of cycles	1
Cycle time (sec)	1
UV Lamp	On
Visible Lamp	Off
Lamp change at (nm)	319.2

Calculate standard deviation of 10 measurements.

NOTE: There is no Autozero energy scan. There is no reference material for this test.

NOTE: This test uses a Scan method.

How does 'Perform every' work?

Perform every allows you to select how often you want to perform the test.

- Select **Days, Weeks or Months** from the drop down list and then enter the appropriate value in the field to the left.
This test will then only be performed at the set interval.

It is possible to define different time intervals for each of the tests within an IPV setup. The software will check when each of the tests are due and will only prompt for the necessary tests to be run.

How does Stop tests on failure work?

If **Stop tests on failure** is selected, as soon as one of the tests fails the IPV is stopped. The Event log section of the Audit trail records that the IPV failed. You can run the entire IPV again.

If **Stop tests on failure** is not selected, a failure of the test is recorded but the software proceeds with any remaining tests.

See also

IPV Setup

Running an IPV

Running an IPV

An IPV can be setup to run at a particular time or on demand only. See IPV Setup.

NOTE: IPV is only available for Medium-performance instruments.

How do I run an IPV that has been setup to run at a particular time?

If the IPV has been setup to run at a particular time, the first time you enter the software after the selected time, a message will be displayed informing you that the IPV is due if at least one of the IPV tests in the setup is due to be run.

The Instrument Performance Verification dialog lists the tests to be run.

1. Click **Run**.

The tests are performed in the order listed.

2. Follow the prompts on the screen.

Immediately a test has been run the result of the test (PASS or FAIL) is displayed. If print report after each test was selected, the report for the test is printed using the default report template for the IPV test.

NOTE: If **Stop tests on failure** was selected and a test fails, the IPV stops and no further tests are run. If **Stop tests on failure** was not selected, all the tests are performed regardless of whether other tests in the IPV have passed or failed, and the results were recorded.

If print report at end of tests was selected in the setup, when the tests are complete the reports are printed/saved as previously defined. If **Print hardcopy of report** was selected, the Print/Reprint dialog is displayed.

3. Select a **Reason** and if required, enter a **Comment** to confirm printing of the report.

The report is printed using the default IPV report template for the test.

It is also possible to see the report from each test:

4. Select the test whose report you wish to view and then click **Display Report**.

The report is displayed in the Communiqué Print Preview window. If **Summary** was previously selected in the IPV Setup, only a summary of the report is displayed. For further information on printing the report from the Communiqué Print Preview window, see the CommuniquéReport Creator section of this Help  Welcome to Communiqué .

If you are using the Standard version of UV WinLab you do not have to save the results of the IPV tests. However, if you are using the Enhanced Security version, you must save the results before exiting.

5. Click **Save**.

The Save dialog is displayed.

6. Enter your Name, Password, Reason and Comment.

The fields displayed on this dialog depend on the settings previously defined by the UV WinLab Administrator.

7. Click **OK**.

The IPV results are saved.

NOTE: If the setup contains a series of tests that need to be run at different time intervals, the software will determine which of the tests need to be run at the time and only perform those tests.

How do I run an IPV setup on demand?

The IPV setup will be run on the default instrument. All tests in the setup will be run regardless of the frequency defined. The IPV setup assigned to the instrument remains unchanged.

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have permission to create IPV setups in order to be able to run an IPV on demand.

1. From the Folder List (Tree) select **Instruments**.
2. In the Main pane, select the Medium performance instrument whose performance you want to verify.
3. From the Tools menu select **Instrument Performance Verification**, and then from the sub-menu select **Perform now**.
The Select IPV Setup dialog is displayed.
4. Select the IPV setup that you wish to perform from the drop-down list of available setups.
5. Click **Perform**.
The Instrument Performance Verification dialog is displayed with the list of tests to be performed.

NOTE: If an SOP was defined in the IPV setup, the **Display SOP** button is available. Click **Display SOP** to display the SOP for running the IPV.

6. Click **Run**.
The tests are performed in the order they are listed.
7. Follow the prompts on the screen.
Immediately a test has been run the result of the test (PASS or FAIL) is displayed. If print report after each test was selected, the report for the test is printed using the default report template for the IPV test.

NOTE: If **Stop tests on failure** was selected and a test fails, the IPV stops and no further tests are run. If **Stop tests on failure** was not selected, all the tests are performed regardless of whether other tests in the IPV have passed or failed, and the results are recorded.

If print report at end of tests was selected in the setup, when the tests are complete the reports are printed/saved as previously defined. If **Print hardcopy of report** was selected, the Print/Reprint dialog is displayed.

8. Select a **Reason** and if required, enter a **Comment** to confirm printing of the report.
The report is printed using the default IPV report template for the test.

It is also possible to see the report from each test:

9. Click on the test whose report you wish to view (to select it) and then click **Display Report**.

The report is displayed in the Communiqué Print Preview window. If **Summary** was previously selected in the IPV Setup, only a summary of the report is displayed. For further information on printing the report from the Communiqué Print Preview window, see the Communiqué Report Creator section of this Help  Welcome to Communiqué.

If you are using the Standard version of UV WinLab you do not have to save the results of the IPV tests. However, if you are using the Enhanced Security version, you must save the results before exiting.

10. Click **Save**.

The Save dialog is displayed.

11. Enter your Name, Password, Reason and Comment.

The fields displayed on this dialog depend on the settings previously defined by the UV WinLab Administrator, and whether you are using the Standard or Enhanced Security version of UV WinLab..

12. Click **OK**.

The IPV results are saved.

What will happen if I try to run IPV tests with an accessory fitted?

If any accessory is fitted, a message will be displayed asking you to remove the accessory before the IPV is run.

What happens if an IPV test fails?

If Stop tests on failure was selected, the IPV stops and no further tests are run. If Stop tests on failure was not selected, all the tests are performed regardless of whether other tests in the IPV have passed or failed, and the results are recorded.

If an IPV test fails the entire IPV setup is deemed to have failed and the instrument IPV status is FAILED. The IPV status will remain FAILED until all tests in the IPV Setup subsequently pass.

To re-run you must exit the IPV and then select to perform the setup again.

Can I stop the tests once they have been started and then rerun them?

Yes.

1. Click **Stop** to stop the tests.

You must exit the IPV and then select to perform the setup again.

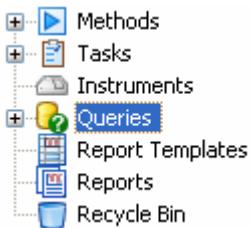
What are the test priorities if all of the tests are selected to run at same time?

1. Wavelength accuracy D2
2. Wavelength repeatability D2
3. Wavelength accuracy with third-party lamp
4. Wavelength accuracy with glass filters
5. Wavelength accuracy with solution
6. Baseline Stability
7. Noise at wavelength 1
8. Noise at wavelength 2
9. Noise at wavelength 3
10. Photometric accuracy with glass filters
11. Photometric repeatability with glass filters
12. Photometric accuracy with K2Cr2O7 solution
13. Photometric accuracy with Potassium Nitrate solution
14. Photometric linearity with glass filters
15. Resolution Toluene
16. Resolution Benzene
17. Stray light with Potassium Chloride solution at 200 nm
18. Stray light with Sodium Iodide solution at 220 nm
19. Stray light with Sodium Nitrite solution at 340 and 370 nm
20. Absorbance Zero Stability

How do I access the results of IPV tests at a later date?

All IPV results can be accessed using Queries. A search can be performed on all the results in the database and only the results that match your search criteria are returned. For further information see Queries.

Queries



Query Setup allows you to select the criteria to be used to search the results database. For example, you may wish to view only the results obtained using a particular method, or all the results generated by a particular Analyst over the previous month.

By selecting an option on each of the Sample Query Setup tabs it is possible to generate a very specific search of the results database.

There are three types of query – sample query , IPV query , and Calibration query .

A Sample query is used to view results. Opening a query lists sample results that fit the criteria specified in the setup. Selecting a result shows the details of that results, such as method details and instrument settings. You can also view the task that the sample came from. Multiple samples can be trended.

An IPV query is used to view the results of an instrument performance verification. Opening a query lists IPVs that fit the criteria specified in the setup. Selecting a verification gives the details for the individual tests while opening one of those tests displays the task that created it.

A Calibration query is used to view results of calibrations in Quant type methods. Opening a query lists calibrations that fit the criteria specified in the setup. Selecting a calibration gives details of the individual calibration.

What is displayed when Queries is selected in the Folder List?

When Queries is selected in the Folder List, the Main pane shows the list of queries that have already been created. Clicking on one of the queries displays information (Name, Description, Type, Created by, Created on, Modified by, Modified on, and Status) for the selected query in the Display pane.

How do I display the Task that created that Sample Result?

1. Select the sample from the Results Table.
2. From the Actions menu select **View Task**.

The Workspace opens with the Task that was used to create the sample result.

What menu commands are available when I right-click on a saved query in the Explorer?

The following menu commands are available:

Open	Displays the relevant Query Window. The user must have the correct permission for this command to be available.
Cut	Cuts the item to the clipboard.
Delete	Deletes the item.

NOTE: Paste is always grayed as it is not applicable here.

How do I save a query?

Queries can be performed on a one off basis or they can be saved.

1. From the File menu select **Save As**.
The Save Query dialog is displayed.
2. If folders have been created, select the folder to which the query should be saved.
3. Enter a name and click **OK**.
The query is saved.

How do I run a previously saved query?

All queries are saved in the Queries folder within the Explorer.

1. Select the Queries folder.
All the saved queries are listed in the Main pane.
2. Double-click on the required query.
The Query window opens and displays all the data currently in the database relating to the selected query.

How do I export the Results Table?

1. From the Actions menu select **Export Results**.
A Save As dialog is displayed.
2. Select the required directory and enter a name for the results.
The file extension must be .csv.
3. Click **Save**.
The dialog closes and the file is saved.

How do I export a spectrum from the query results?

1. From the list of results, select the sample whose spectrum you wish to export.
2. From the Results Tree, select **Sample Spectrum**.
The spectrum is displayed.

3. From the Actions menu select **Export Spectrum**.
A Save As dialog is displayed.
4. Enter the name of the spectrum and select the format for the file – either *.asc or *.sp.
5. Click **Save**.
The spectrum is exported to the selected directory.

How do I add a folder to store my query results in?

- To add a folder, select **New** from the File menu in the Explorer Window and then select **Folder** from the submenu.

OR

Right-click within the Query Main pane and from the menu select **New** and then select **Folder** from the submenu.

A new folder is added, and you can name the folder.

For further information about folders, including settings permissions on the folder, see Folders.

How do I delete a folder?

- To delete a folder, select the folder in the Explorer Window and then select **Delete** from the File menu.
You will be asked to confirm that you wish to delete the folder.

NOTE: The folder must be empty before it can be deleted.

For further information about folders, see Folders.

See also

Printing

Approving results

Adding comments

Sample Queries

How do I set up a Samples query?

1. In Explorer, from the File menu select **New** and then **Query**, and then from the submenu select **Samples**.

The Sample Query Setup dialog is displayed.

2. Set the options on the tabs as required:

NOTE: If you perform a Sample Query you will get the results for samples and IPVs. This is because a spectrum generated during an IPV test can only be viewed by selecting the IPV test from the Results Table of a Sample Query. To avoid displaying the IPV results, specify a **Selected Method** or **Method of type** on the Method tab or the Sample Query Setup.

The Method tab is displayed by default. However, once a query has been performed, the dialog will then default to the last used tab on the dialog.

Method tab

Select **All Methods**, **Selected method**, **Method of type**, or **Methods containing**.

All Methods	When All Methods is selected, the query searches all methods.
Selected method	Select Selected method to search for samples from the selected method only. Select the method from the drop-down list of available methods.
Method of type	Select Method of type to search for samples from only tasks of the correct type. Select the type of task from the drop-down list. Choose from Scan, Wavelength Program, Timedrive and WavelengthQuant, Scan Quant and Polarization.
Methods containing	Select Methods containing to search for samples in methods with the selected text. Enter the required text to search on.
In the Name	Select In the Name to search for the text in the method names. This option is only available when Methods containing is selected. NOTE: It is possible to select In the Name and In the Description. The first time Methods containing is selected, these check boxes only become active after text has been entered in the Methods containing field.
In the Description	Select In the Description to search for the text in the method descriptions. This option is only available when Methods containing is selected. NOTE: It is possible to select In the Name and In the Description. The first time Methods containing is selected, these check boxes only become active after text has been entered in the Methods containing field.

Sample tab

Select All Samples, Samples of Type, or Samples containing.

All Samples	When All Samples is selected, the query searches all samples.
Samples of type	Select Samples of type to search for samples of a particular type. Select the type from the drop-down list of available sample types: Sample, Control, Blank, Replicate Mean, Replicate and Measurement.
Samples containing	Select Samples containing to search for samples which contain the selected text. Enter the required text to search on.
In the Name	Select In the Name to search for the text in the names of the samples. This option is only available when Samples containing is selected. NOTE: It is possible to select In the Name, In the Description and In a custom column. The first time Samples containing is selected, these check boxes only become active after text has been entered in the Samples containing field.
In the Description	Select In the Description to search for the text in the descriptions of the samples. This option is only available when Samples containing is selected. NOTE: It is possible to select In the Name, In the Description and In a custom column. The first time Samples containing is selected, these check boxes only become active after text has been entered in the Samples containing field.
In a custom column	Select In a custom column to search for the text in any custom columns. This option is only available when Samples containing is selected. NOTE: It is possible to select In the Name, In the Description and In a custom column.

Person tab

Select All Analysts, Current Login, Analyst, or Approver.

NOTE: Approver is only available in the Enhanced Security version of UV WinLab.	
All Analysts	When All Analysts is selected, the query searches all analysts.
Current Login	Select Current Login to search for samples based on the name currently logged in.
Analyst	Select Analyst to search for samples run by a selected person. Select the name from the drop-down list of all users.
Approver	Select Approver to search for samples approved by a selected person. Select the name from the drop-down list of Approvers (users with permission to approve Methods and IPV Setups).

Date tab

Select All Dates, Today, Over the past X days, or Over selected period.

All Dates	When All Dates is selected, the query searches all dates.
Today	Select Today to search only for samples run today.
Over the past (days)	Select Over the past (days) to search for samples run over the past X days. Select the number of days from the drop-down list.
Over selected period	Select Over selected period to search for samples run between (and including) selected dates. Select the From (start of search period) and To (end of search period) dates. Click  to display a calendar. Select the month using the left and right arrow buttons next to the month and then click on a date to select it.

Result tab

Select All Results, Results with Status, or Results where a custom column reports.

All Results	When All Results is selected, the query searches all results.
Results with Status	Select Results with Status to search for samples where the result has the selected status. Select the status -Pending, Complete, Approved, Reviewed, Imported, Halted, Excluded, from the drop-down list.
Results where a custom column reports	Select Results where a custom column reports to search for samples where the entered text or number exists in a custom column.

Instrument tab

Select All Instruments, Instrument type or Selected Instrument.

All Instruments	When All Instruments is selected, the query searches all instruments.
Instrument type	Select Instrument type to search for samples run on a selected instrument type. Select the instrument type from the drop-down list.
Selected Instrument	Select Selected Instrument to search for samples run on a specific instrument. Select the instrument from the drop-down list.

Sorting tab

Sort by	Select Sort by to sort the query results by a selected column. Select the column from the drop-down list.
Then sort by	Select Then sort by to do a second level sort. Select the column from the drop-down list.

1. Click **OK**.

The database is searched based on the criteria specified and the results are displayed in a Query window.

IPV Queries

How do I set up an IPV query?

A **verifications** query is used to view the results of an instrument verification (IPV).

1. From the File menu select **New** and then **Query**, and then from the submenu select **Verifications**.

The IPV Query Setup dialog is displayed.

2. Set the options on the tabs as required:

The Instrument tab is displayed by default. However, once a query has been performed, the dialog will then default to the last used tab on the dialog.

Instrument tab

Select either All Instruments or Selected Instrument.

All Instruments	When All Instruments is selected, the query does not search based on the actual result.
Selected Instrument	Select Selected Instrument to search for IPV results run on a specific instrument. Select the instrument from the drop-down list.

Setup tab

Select either All Setups or This setup only.

All Setups	When All Setups is selected, the query does not search based on IPV setup.
This setup only	Select This setup only to search for IPV results from a particular setup. Select the setup from the drop-down list.

Test tab

Select either All Tests or IPVs that include this test.

All Tests	When All Tests is selected, the query does not search based on individual IPV tests.
IPVs that include this test	Select IPVs that include this test to search for IPV results from the selected test.

Person tab

Select All Analysts, Current Login, Analyst, or Approver.

All Analysts	When All Analysts is selected, the query does not search based on analysts.
Current Login	Select Current Login to search for IPV results based on the name currently logged in.

Analyst	Select Analyst to search for IPV results run by a selected person. Select the user from the drop-down list of all users.
Approver	Select Approver to search for IPV results approved by a selected person. Select the name from the drop-down list of Approvers (users with approval permission).

Date tab

Select All Dates, Today, Over the past X days, Over selected period.

All Dates	When All Dates is selected, the query does not search based on dates.
Today	Select Today to search only for IPV results run today.
Over the past (days)	Select Over the past(days) to search for IPV results run over the past X days. Select the number of days from the drop-down list.
Over selected period	Select Over selected period to search for IPV results run between (and including) selected dates. Select the From (start of search period) and To (end of search period) dates from the drop-down lists. Click  to display a calendar. Select the month using the left and right arrow buttons next to the month name and then click on a date to select it.

Result tab

Select All Results, All failures or All passes.

All Results	When All Results is selected, the query does not search based on the actual result.
All failures	Select All failures to search only for IPV tests that failed.
All passes	Select All passes to search only for IPV tests that passed.

Sorting tab

Sort by	Select Sort by to sort the query results by a selected column. Select the column from the drop-down list.
Then sort by	Select Then sort by to do a second level sort. Select the column from the drop-down list.

1. Click **OK**.

The database is searched based on the criteria specified, and the results are displayed in a UV WinLab Sample Query window.

Calibration Queries

How do I set up a Calibration query?

1. In the Explorer, from the File menu select **New** and then **Query**, and then from the submenu select **Calibrations**.
The Calibrations Query Setup dialog is displayed.
2. Set the options on the tabs as required.

NOTE: The Method tab is displayed by default. However, once a query has been performed, the dialog will then default to the last tab used on the dialog.

Method tab

Select All Methods, Selected, or Methods containing.

All Methods	When All Methods is selected, the query does not search based on a particular method. (All methods are included in the search.)
Selected	Choose Selected to search for calibrations from the selected quant method only. Select the method from the drop-down list of available methods.
Methods containing	Select Methods containing to search for calibrations in quant methods with the selected text. Enter the required text to search on.
In the Name	Select In the Name to search for the text in the method names. This option is only available when Methods containing is selected. NOTE: It is possible to select In the Name and In the Description.
In the Description	Select In the Description to search for the text in the method descriptions. This option is only available when Methods containing is selected. NOTE: It is possible to select In the Name and In the Description.

Person tab

Select All Analysts, Current login, or Analyst.

All Analysts	When All Analysts is selected, the query does not search based on analysts. (All analysts are included in the search.)
Current login	Select Current login to search for calibrations based on the name currently logged in.
Analyst	Select Analyst to search for calibrations run by a selected person. Select the name from the drop-down list of all users.

Date tab

Select All Dates, Today, Over the past X days, or Over selected period.

All Dates	When All Dates is selected, the query does not search based on dates. (All dates are included in the search.)
Today	Select Today to search only for calibrations run today.
Over the past (days)	Select Over the past (days) to search for calibrations run over the past X days. Select the number of days from the drop-down list.
Over selected period	Select Over selected period to search for calibrations run between (and including) selected dates. Select the From (start of search period) and To (end of search period) dates. Click <input type="button" value="..."/> to display a calendar. Select the month using the left and right arrow buttons next to the month and then click on a date to select it.

Result tab

Select either All Calibrations or All Calibrations with.

All Calibrations	When All Calibrations is selected, the query does not search based on a particular calibration.(All calibrations are included in the search)
All Calibrations with	Select All Calibrations with to search for calibrations with specific parameters. Select Correlation (R-squared Values) and/or Intercept Between.
Correlation (R-squared) Values	Select Correlation (R-squared) Values to search for calibrations with a correlation value within a specified range. Select Less Than (equal to) or Greater Than (equal to), and then a value between 0.00001 and 1.00000. This option is only available when All Calibrations with is selected. NOTE: It is possible to select Correlation (R-squared Values) and Intercept Between.
Intercept Between	Select Intercept Between to search for calibrations with an intercept between two specified values. Enter a Lower Value and an Upper Value. This option is only available when All Calibrations with is selected. NOTE: It is possible to select Correlation (R-squared Values) and Intercept Between.

Sorting tab

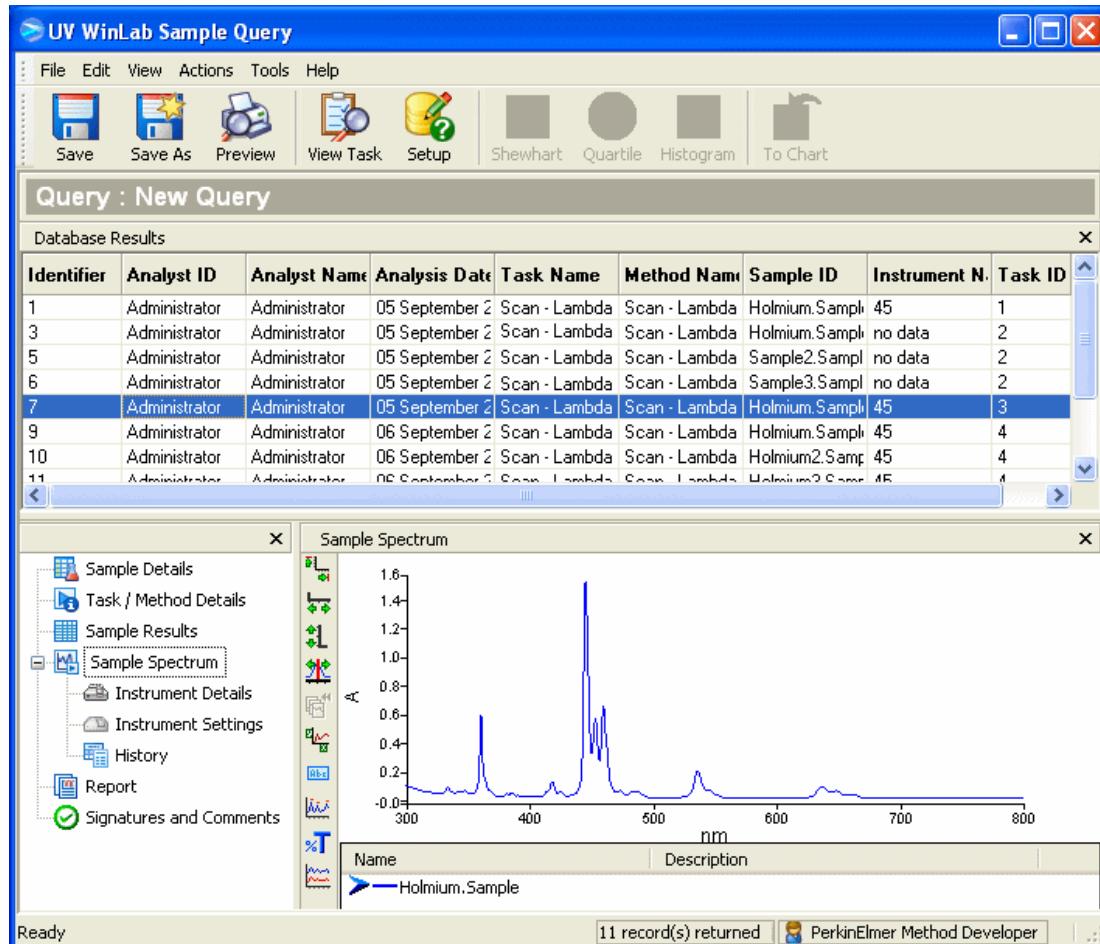
Sort by	Select Sort by to sort the query results by a selected column. Select the column from the drop-down list.
Then sort by	Select Then sort by to do a second level sort. Select the column from the drop-down list.

- Click **OK**.

The database is searched based on the criteria specified and the results are displayed in a Query window.

The Query Window

The Query window is used to view and further analyze the results of the sample and verification queries. The selected view is generated by using the Query Editor dialog to specify the requirements to use when searching the results database.



The Query window is divided into four main areas:

The Menus and Toolbar enable you to interact with the results.

The Results Table displays the results and by selecting a row or rows, it enables you view the information related to particular results.

The Results Tree enables you to navigate around the results shown in the Results Table.

The Results Display displays the information about the sample or samples selected in the Results Table, such as sample details, instrument settings, spectra and charts.

NOTE: Raw spectra (not processed spectra) are displayed in the Results Display.

What do the commands on the File menu do?

Save	Saves the current query details. See Saving a Query
Save As	Enables you to edit a previously saved query and then save it with a new name. NOTE: If the name is not changed the query is overwritten.
Print Preview	Displays a preview of the report in the Communiqué Print Preview window. NOTE: This is only possible if a report is associated with the method/task.
Exit	Exits the Query window. If it is a new query or an edited query you are asked if you wish to save the query.

What do the commands on the Edit menu do?

Copy table	Copies the current table to the clipboard.
Copy graph/spectrum/data	Copies the currently displayed graph/spectrum/or data. NOTE: A spectrum is only displayed if one row of the Results Table is selected and sample spectrum is selected in the Tree View. However, it is not applicable and therefore not available for Wavelength Program data. The option to Copy data is only available for Wavelength Program results.

What do the commands on the View menu do?

Table	Toggles the Table on and off. A check mark is displayed when the Table is switched on.
Tree View	Toggles the Results Tree View on and off. A check mark is displayed when the Tree View is switched on.
Display pane	Toggles the Display pane on and off. A check mark is displayed when the Display pane is switched on.
Toolbar	Toggles the Toolbar on and off. A check mark is displayed when the Toolbar is switched on.
Options	Displays an options dialog where you can set the number of decimal places to be used when displaying data, and long or short date format.

What do the commands on the Actions menu do?

Open task	Opens the Task window and displays the task used to create the result. NOTE: This is only available for a sample query and only when a single sample is selected.
Query Setup	Displays the relevant Query Setup dialog with details of the currently loaded query. Also use Query Setup to edit queries.
Select Columns	Displays a dialog for selecting the columns displayed on the query table.
Export Results	Allows the selected results to be exported as a .csv file.
Export Spectrum/Export Data	Export Spectrum allows selected spectra to be exported as .asc or .sp. If the results are from a Wavelength Program Task, the Actions menu option is Export Data.
Add Spectra	Displays a dialog enabling spectra to be added to the spectral graph.
Add chart	Displays a submenu of Shewhart, Quartile and Histogram. Selecting an option opens the Format Chart dialog enabling you to set up a new chart to be displayed. Add chart is only available if multiple (appropriate) results are selected.
Format chart	Opens the Shewhart Chart dialog, Quartile Chart dialog or Histogram dialog as appropriate, enabling the current chart to be re-formatted. See Shewhart Chart, Quartile Chart, Histogram.
Return to chart	Returns the view to the Quartile or Shewhart chart from the Sample Results Tree. See Shewhart Chart and Quartile Chart for more information.

What do the commands on the Tools menu do?

Approve	Displays the approval dialog enabling an authorized user to approve a sample. NOTE: This is only available in the Enhanced Security version of UV WinLab. See Approving Results.
Review	Displays the Review dialog enabling an authorized user to review a sample. NOTE: This is only available in the Enhanced Security version of UV WinLab. See Reviewing results.
Comments	Displays the Comments dialog enabling an authorized user to add a comment to a sample. See Adding Comments.

What do the commands on the Help menu do?

Contents and Index	Displays the on-screen HTML Help system at the default page.
--------------------	--

What menu commands are available as Toolbar buttons?

	Save
	Save As
	Print Preview
	View task
	Query setup
	Format Shewhart chart
	Format Quartile chart
	Format Histogram
	Return to chart

You can also:

Approve Results

Create and Print Reports

Results Table

Identifier	Analyst ID	Analyst Name	Analysis Date	Task Name	Method Name	Sample ID	Instrument Na
286	fmq	Farakh Developer	28/04/2004 14:40 GMT De	Scan 28 April 2004 1	Scan 28 April 2004	Sample1.Samp	L35
287	fmq	Farakh Developer	28/04/2004 14:40 GMT De	Scan 28 April 2004 1	Scan 28 April 2004	Sample2.Samp	L35
288	fmq	Farakh Developer	28/04/2004 14:42 GMT De	Scan 28 April 2004 1	Scan 28 April 2004	Sample3.Samp	L35
289	fmq	Farakh Developer	28/04/2004 14:42 GMT De	Scan 28 April 2004 1	Scan 28 April 2004	Sample4.Contrc	L35
309	PEService	PerkinElmer Service Persor	28/04/2004 18:12 GMT De	OQ Scan and Repo	OQ Scan	D2 Scan.Sampl	L35
312	PEService	PerkinElmer Service Persor	28/04/2004 18:35 GMT De	OQ Scan Revision 2	OQ Scan	D2 Scan.Sampl	L35
313	PEService	PerkinElmer Service Persor	28/04/2004 18:44 GMT De	OQ Data Overwrite	OQ Scan	D2 Scan.Sampl	L35
314	PEService	PerkinElmer Service Persor	28/04/2004 18:50 GMT De	OQ Data Overwrite	OQ Scan	D2 Scan-2.Sam	L35

The Results Table displays all the results of the query.

NOTE: All data values are reported to 2 decimal places.

How do I view the information associated with a particular result in the Results Table?

- To view the results associated with a particular sample, IPV or calibration result, click on the required row of the Results Table.

The Results Tree is displayed. It is the means of navigation around all the data associated with the results displayed in the Results Table for a query. Information related to the selected branch of the Results Tree is displayed in the Display Pane.

How do I view the results associated with more than one sample?

- To view the results associated with more than one sample in the Results Table, highlight the required rows using the mouse.

NOTE: When selecting more than one sample it is only possible to select all samples or a block of samples. It is not possible to select non-contiguous samples from the Results Table. If the samples you require are not in a block, edit the query using the Sort By tab to sort your data.

When more than one sample is selected it is possible to display the results using any of the following plots: Shewhart Chart, Quartile Chart, or Histogram.

NOTE: The Results Tree for multiple samples is not shown until a chart has been generated.

NOTE: Shewhart Charts and Quartile Charts are not available for IPV results.

How can I filter samples, standards and baselines?

A query may return standards and baselines as well as samples as they are all treated as 'samples' in a sample query. If you wish to sort these into separate groups, click on the Sample ID column title. The query results will then sort by Sample ID and you will be able to easily distinguish samples, standards and baselines.

How can I sort the results of the query?

If you wish to reorganize the results, you can click on a column title. This will then sort the results according to this column. This is the same as using the Sort By tab when setting up the query.

Display Pane

The Display Pane is used to display spectra, plots, or information, depending on the Results Tree item selected.

How do I display the spectrum of a sample in the Results Table?

NOTE: Spectra are only available for Timedrive and Scan samples.

1. Click in the required row of the Results Table to select the sample.
The row is highlighted blue to show that the sample is selected and the Results Tree is updated.
2. On the Results Tree click **Sample Spectrum** (for sample queries) or **Spectrum** (for IPV queries).
The spectrum of the selected sample/ IPV result is displayed in the Results Display. The spectrum is the raw spectrum.
It does not show any of the processing that may have been applied as part of the Task (for sample queries).

NOTE: If another sample is selected from the Results Table the Display Pane is updated accordingly.

NOTE: Calibration query results display graphs and not individual spectra. See Results Tree for further information on the available data.

Is the spectrum displayed the raw or processed spectrum?

The raw spectrum is displayed.

- To view the processed spectrum (of a Sample Query Result only), double-click on the sample row in the Results Table.
The Task opens and the processed spectrum can be viewed.

How do I manipulate the spectra displayed?

For information on manipulating the spectrum using the toolbar buttons, see Graphs.

How do I display a Shewhart Chart, Quartile Chart or Histogram?

Shewhart Charts, Quartile Charts and Histograms can be displayed within the Results Display when multiple samples are selected in the Results Table.

NOTE: When selecting more than one sample it is only possible to select all samples or a block of samples. It is not possible to select non-contiguous samples from the Results Table. If the samples you require are not in a block, edit the query using the Sort By tab to sort your data.

1. Select the samples in the Results Table to be used to generate the plot.
2. From the Actions menu select **Add Chart** and from the submenu select **Shewhart Chart, Quartile Chart or Histogram**.
The appropriate dialog is displayed.

3. Select the required parameters and click **OK**.

The plot is displayed in the Results Display.

NOTE: Shewhart Charts and Quartile Charts are not available for IPV Query results.

See also

[Displaying Multiple Spectra](#)

Adding Comments

You can add your own comment to a sample in the Results Table.

How do I add a comment to a sample in the Results Table?

1. Select the result(s).
2. From the Tools menu select **Comments**.
3. Enter a **Comment** and click **OK**.

The Comment can be viewed in the **Signatures and Comments** section of the Tree.

NOTE: A signature is not required when adding a comment.

How can I view the comment that was added to a sample?

- Select the result in the Results Table and then select **Signatures and Comments** from the Results Tree.

Any comments added to the sample are listed.

Shewhart Chart

A Shewhart chart is a control chart that can be used to determine when variation may no longer be due to random variation, but to an assignable cause. Shewhart charts involve plotting the actual observation on a chart with warning and action limits depicted.

How do I display a Shewhart Chart of the results?

NOTE: Shewhart Chart is only available if at least two rows of the Results Table are selected. It is only possible to select consecutive rows within the Results Table. If the rows that you require are not consecutive, use the Sorting tab within the Sample Query Setup dialog to rearrange the order in which the results are displayed.

NOTE: Shewhart Charts are not available for IPV Query results.

1. Select the samples from the Results Table that are to be included in the Shewhart Chart.
 2. From the Actions menu select **Add Chart**, and from the submenu select **Shewhart**  **Chart** or click .
- The Shewhart Chart dialog is displayed.
3. Enter the necessary text and select the required options.
 4. Click **OK**.
- The Shewhart Chart is displayed in the Results Display.

NOTE: It is the type of chart and the specified parameters that are saved rather than a chart that relates to currently selected samples in the Results Table. For example, you may select all the samples in the Results Table and then display a Shewhart Chart. If you then select a subset of the samples in the Results Table the Shewhart Chart will update to reflect the samples now selected. The next time you highlight more than one sample in the Results Table, the **Multiple Sample Results** Results Tree will be displayed. The Results Tree will list the branch **Shewhart Charts** and the **Name** of the chart under this branch, and the chart will reflect the currently highlighted samples in the Results Table.

It is possible to generate more than one plot for each type of chart. Each plot is listed under the appropriate sub-branch of the **Multiple Sample Results** Results Tree.

The following parameters are set within the Shewhart Chart dialog:

Name	A default Name is provided for the chart. The default name changes depending upon the Y-Axis selected. To edit the Name , select the Name field and enter the required text. The Name of the chart is displayed in the Results Tree.
Y-Axis	Select the required Y-Axis from the drop-down list.
X-Axis	The X-Axis is Sample Index-Date Order.
Y-Axis label	A default Y-Axis label is provided. To edit the label, select the field and enter the required text. The Y-Axis label is displayed on the chart.

Title	A default Plot Title is provided. To edit the label, select the field and enter the required text. The Plot Title is displayed on the chart.
No limits	Select this to show the chart without limit lines.
Show statistical limits 2 and 3 standard deviations	Select this if you wish to display limit lines at 2 and 3 standard deviations on the chart.

How can I modify the settings of a Shewhart Chart?

1. Select the required chart from the folder list.

The chart is displayed in the Display Pane.



2. From the Actions menu select **Format Chart** or click . The Shewhart Chart dialog is displayed.
3. Edit the required options.
All the options can be edited except the Name of the chart.
4. Click **OK** to save the modified settings.
The chart is updated to reflect the new settings.

NOTE: It is also possible to modify the Shewhart chart via the Results Tree. Right-click on the selected chart name and from the menu select **Format Chart**. The Shewhart Chart dialog is displayed.

NOTE: Changing the Axes changes the default details in the label boxes, so the Axes should be selected first.

How do I delete a Shewhart Chart?

It is possible to delete a Shewhart chart if you do not wish it to be saved as part of the query.

1. Select the name of the Shewhart chart from the Results Tree.
The name is highlighted blue and the chart is displayed in the Display Pane.
2. Right-click on the name of the Shewhart chart in the Results Tree, and from the menu select **Delete**.
The selected chart is deleted.

How do I identify a particular sample within the Shewhart Chart?

Each of the points on the Shewhart Chart corresponds to one of the selected samples from the Results Table.

- To view the spectrum associated with a particular point, position the mouse over the point and then click.
When the mouse is over a point it changes to a cross to show that the point can be clicked.
When the point is clicked, the Results Tree for that sample is displayed, allowing you to view the analysis spectrum in the Display pane, and the result to which the spectrum belongs is highlighted in the Results Table.
- To return to the Shewhart Chart after viewing the spectrum, select **Return To Chart** from the Actions menu.
The Shewhart Chart is redisplayed.

NOTE: **Return to Chart** will be grayed out until a point on the chart has been selected and the individual sample is shown in the Display pane.

Quartile Chart

A quartile chart shows results as a function of time using polar coordinates and therefore giving a circular plot. The chart can include limits.

How do I display a Quartile Chart of the results?

NOTE: **Quartile Chart** is only available if at least two rows of the Results Table are selected. It is only possible to select consecutive rows within the Results Table. If the rows that you require are not consecutive, use the Sorting tab within the Sample Query Setup dialog to rearrange the order in which the results are displayed.

NOTE: Quartile Charts are not available for IPV Query results.

1. Select the results to be displayed in the chart from the Results Table.
2. From the Actions menu select **Add Chart**, and from the submenu select **Quartile Chart** or click  .
The Quartile Chart dialog is displayed.
3. Enter the necessary text and select the required options.
4. Click **OK**.
The Quartile Chart is displayed in the Results Display.

NOTE: It is the type of chart and the specified parameters that are saved rather than a chart that relates to currently selected samples in the Results Table. For example, you may select all the samples in the Results Table and then display a Quartile Chart. If you then select a subset of the samples in the Results Table the Quartile Chart will update to reflect the samples now selected. The next time you highlight more than one sample in the Results Table, the **Multiple Sample Results** Results Tree will be displayed. The Results Tree will list the branch **Quartile Chart** and the **Name** of the plot under this branch, and the plot will reflect the currently highlighted samples in the Results Table.

Is it possible to generate more than one plot for each type of chart?

It is possible to generate more than one plot for each type of chart. Each plot is listed under the appropriate sub-branch of the **Multiple Sample Results** Results Tree.

For each **Chart Name**, the Results Display lists the **Y Axis**, **X Axis**, and **Chart Title**.

The following parameters are set within the Quartile Chart dialog:

Name	Enter the required Name of the Quartile Chart. The Name appears in the Results Tree.
Radial	Select the required Radial from the drop-down list.
Angular	Select either Sample Date or Sample Index-Date Order.
Y-Axis label	A default Y-Axis label is provided. To edit the label, select the field and enter the required text. The Y-Axis label is displayed on the Quartile Chart

Title Enter the required Title. The Title is displayed on the Quartile Chart.

Show statistical limits 2 and 3 standard deviations Select this if you wish to display limit lines at 2 and 3 standard deviations on the chart.

NOTE: Changing the Axes changes the default details in the label boxes, so the Axes should be selected first.

How can I modify the settings of a Quartile Chart?

1. Select the required Quartile Chart from the folder list.

The chart is displayed in the Display Pane.



2. From the Actions menu select **Format Chart** or click .

The Quartile Chart dialog is displayed.

3. Edit the required options.

All the options can be edited except the Name of the Quartile Chart.

4. Click **OK** to save the modified settings.

The Quartile Chart is updated to reflect the new settings.

NOTE: It is also possible to modify the Quartile chart via the Results Tree. Right-click on the selected chart name and from the menu select **FormatChart**. The Quartile dialog is displayed.

How do I delete a Quartile Chart?

It is possible to delete a Quartile chart if you do not wish it to be saved as part of the query.

1. Select the name of the Quartile chart from the Results Tree.

The name is highlighted blue and the chart is displayed in the Display Pane.

2. Right-click on the name of the Quartile chart in the Results Tree, and from the menu select **Delete**.

The selected chart is deleted.

How do I identify a particular sample within the Quartile Chart?

Each of the points on the Quartile Chart corresponds to one of the selected samples from the Results Table.

- To view the spectrum associated with a particular point, position the mouse over the point and then click.

When the mouse is over a point it changes to a cross to show that the point can be clicked.

When the point is clicked, the Results Tree for that sample is displayed, allowing you to view the analysis spectrum in the Display pane, and the result to which the spectrum belongs is highlighted in the Results Table.

- To return to the Quartile Chart after viewing the spectrum, select **Return To Chart** from the Actions menu.
The Quartile Chart is redisplayed.

NOTE:**Return to Chart** will be grayed out until a point on the chart has been selected and the individual sample is shown in the Display pane.

Histogram

A histogram is a graphical representation of class frequencies as rectangles against class interval, the value of frequency being proportional to the area of the corresponding rectangle.

How do I display a Histogram of the results?

NOTE: Histogram is only available if at least two rows of the Results Table are selected. It is only possible to select consecutive rows within the Results Table. If the rows that you require are not consecutive, use the Sorting tab within the Sample Query Setup dialog to rearrange the order in which the results are displayed.

1. Select the results to be displayed in the chart from the Results Table.
2. From the Actions menu select **Add Chart**, and from the submenu select **Histogram** or click . The Histogram dialog is displayed.
3. Enter the necessary text and select the required options.
4. Click **OK**. The Histogram Chart is displayed in the Results Display.

NOTE: It is the type of chart and the specified parameters that are saved rather than a chart that relates to currently selected samples in the Results Table. For example, you may select all the samples in the Results Table and then display a Histogram. If you then select a subset of the samples in the Results Table the Histogram will update to reflect the samples now selected. The next time you highlight more than one sample in the Results Table, the **Multiple Sample Results** Results Tree will be displayed. The Results Tree will list the branch **Histograms** and the **Name** of the plot under this branch, and the plot will reflect the currently highlighted samples in the Results Table.

Is it possible to generate more than one plot for each type of chart?

It is possible to generate more than one plot for each type of chart. Each plot is listed under the appropriate subbranch of the **Multiple Sample Results** Results Tree.

For each **Chart Name**, the Results Display lists the **Y Axis**, **X Axis**, and **Chart Title**.

The following parameters are set within the Histogram dialog:

Name	Enter the required Name of the Histogram. The Name appears in the Results Tree.
Y-Axis	Select the required Y-Axis from the drop-down list. The options are determined from the columns displayed in the Results Table.
X-Axis	Select the required X-Axis from the drop-down list. The options are determined from the columns displayed in the Results Table.
Y-Axis label	A default Y-Axis label is provided. To edit the label, select the field and enter the required text. The Y-Axis label is displayed on the histogram.

Title Enter the required **Title**. The **Title** is displayed on the Histogram.

NOTE: Changing the Axes changes the default details in the label boxes, so the Axes should be selected first.

How can I modify the settings of a Histogram?

1. Select the required histogram from the folder list.
The histogram is displayed in the Display Pane.
2. From the Actions menu select **Format Chart** or click 
The Histogram dialog is displayed.
3. Edit the required options.
All the options can be edited except the Name of the histogram.
4. Click **OK** to save the modified settings.
The histogram is updated to reflect the new settings.

NOTE: It is also possible to modify the histogram via the Results Tree. Right-click on the selected chart name and from the menu select **Format Chart**. The Histogram dialog is displayed.

How do I delete a Histogram?

It is possible to delete a histogram if you do not wish it to be saved as part of the query.

1. Select the name of the histogram from the Results Tree.
The name is highlighted blue and the histogram is displayed in the Display Pane.
2. Right-click on the name of the histogram in the Results Tree, and from the menu select **Delete**.
The selected histogram is deleted.

Copying a Spectrum, Chart or Table

Results, Sample spectra, Shewhart Charts, Quartile Charts and Histograms can be copied to the clipboard and from there pasted into other applications such as WordPad.

How do I copy a spectrum to the clipboard?

1. Select the required sample from the Results Table.
2. From the Results Tree select **Sample Spectrum**.
The Display Pane displays the selected spectrum.
3. From the Edit menu select **Copy Graph**.
The selected chart is copied to the clipboard.

How do I copy a Shewhart Chart, Quartile Chart or Histogram to the clipboard?

The detail of the plot displayed in the Results Display is dependent upon the samples selected within the Results Table. For example, a different Histogram will be displayed depending upon whether all or only some of the results are selected. The plot updates to reflect the currently selected samples. It is therefore important to ensure the required samples are selected in the Results Table before generating and subsequently copying the plot to the clipboard.

1. Select the required samples from the Results Table.
2. From the Results Tree select the previously saved plot type.
The Display Pane displays the selected plot.
3. From the Edit menu select **Copy Graph**.
The selected graph is copied to the clipboard.

How do I copy a Results Table to the clipboard?

NOTE: Only the rows selected are copied to the clipboard.

1. Select the rows of the table to be copied.
2. From the Edit menu select **Copy Table**.
The selected rows are copied to the clipboard.

Displaying Multiple Spectra

How do I overlay spectra in the Results Display?

NOTE: Overlay is available only if at least two rows of the Results Table are selected. It is only possible to select consecutive rows within the Results Table. If the rows that you require are not consecutive, use the Sorting tab within the Sample Query Setup dialog to rearrange the order in which the results are displayed.

1. Select the samples from the Results Table whose spectra you want to overlay.

An Overlay branch is added to the Results Tree:



2. Select **Overlay** in the Results Tree.

The selected spectra are displayed overlaid in the Results Display.

What is displayed if I select different types of results?

The information displayed in the Results Display depends on the first result that is highlighted in your selection. For example, if you select some rows that are scan results and some that are wavelength program results and the first result is from a scan task, then the spectra are displayed and the wavelength program data not included. If the first result is from a wavelength program task, all the wavelength program data in the selected results are displayed but the spectra from the scan tasks are not displayed.

How do I remove one of the overlaid spectra from the graph?

1. Click the right mouse button on the name of the spectrum you wish to remove in the legend below the graph.
2. Select Remove curve.

The selected spectrum is removed from the graph.

Printing a Query

A variety of reports can be printed from the Query results. These can be for a single result or for selected multiple results in the Query Table.

How do I print a result from the Query Table?

A query result is printed via the CommuniquéReport Creator.

1. Select the result from the Table.

2. From the File menu select **Print Preview**.

The report is displayed in the Communiqué print preview window.

NOTE: If a report template has been specified as part of the method it is this template that is used to display the result. If a template has not been specified, a default template is used.

The report can also be viewed in the Display Pane by selecting **Report** from the Results Tree.

How do I print multiple reports from the Query Table?

Multiple results can be selected from the table and printed.

1. Select the query results that you want to print.

2. From the File menu select **Print Preview**.

The Print Query dialog is displayed.

3. Select Single Report using template, Single Report or Multiple reports using template.

If you select **Single Report**, select to print one or more from **Query Summary**, **Table**, and **Chart**. **Single report** prints a single report for the multiple rows if selected. **Query Summary** prints a summary of the options selected in the Query Setup window. **Table** prints the selected results in the Summary page, and **Chart** prints the currently displayed chart. Chart is only available if a chart is currently displayed.

If you select **Single Report using template**, or **Multiple reports using template**, select the template from the appropriate drop-down list of available templates.

The report can also be viewed in the Display Pane by selecting **Report** from the Results Tree.

For further information on report templates and printing reports, see the Communiqué Report Creator section of this Help.

Queries – Options dialog

The Options dialog defines the number of significant figures to be displayed and the date format in the Results Table.

How do I define the number of decimal places to be displayed when displaying numerical values within the Query window?

The number of **Decimal places** to be used when displaying numerical values within the Results Browser is defined within the Options dialog.

1. From the Tools menu select **Options**.
The Options dialog is displayed.
2. Use the up and down arrows (under the decimal places heading) to select the required number of **Decimal places**.

Can I change the date format used within the Query window?

The dates displayed within the Results Browser can be displayed in **Short Date** or **Long Date** format. The **Short Date** is displayed as 05/06/2002, while the **Long Date** is displayed as 05 June 2002.

1. From the Tools menu select **Options**.
The Options dialog is displayed.
2. Select **Long Date** or **Short Date** from the drop-down list.

Columns

How do I select the columns to appear in the Results Table?

The Columns dialog enables you to select the columns that will appear in the Query Results Table, and also the order in which these columns will appear. In addition, any custom fields are also listed in the columns dialog. These are selected (and so appear in the Results Table) by default.

1. From the Actions menu select **SelectColumns**.

The Columns dialog is displayed.

The selected columns (indicated by a check mark) will appear in the Results Table in the order they appear on the list.

NOTE: When a Results Table is displayed, the Columns dialog can also be displayed by clicking the right mouse button on any of the column headings.

2. To select a column to appear in the table, click the mouse in the box next to the required column name.
A check mark appears in the box to show that the column is selected.
3. To select a column to be removed from the table, click the mouse in the box next to the required column name.
A check mark is removed from the box to show that the column is no longer selected.
4. To move a column, select the name of the column you wish to move by clicking on the name and then click **Move Up** or **Move Down** to move the column up or down the list as required.
5. Click **OK**.

The columns dialog closes and the table is updated.

NOTE: If the content of a column is not applicable to a particular sample, then the field is left blank. If the first sample in the table does not have an entry for that column, the column is not displayed.

Results Tree

The Results Tree is the means of navigation around all the data associated with the results displayed in the Results Table for a query. There are basically four types of Results Tree – one for Sample Details (when only one row of the Results Table is selected in a Sample Analysis view), one for IPV (when only one row of the Results Table is selected in an IPV view), one for calibrations (when only one row of the Results Table is selected in a Calibration view), and one for Multiple Sample Results (when more than one row of the Results Table is selected within a Sample Query or Calibration Query).

NOTE: The Results Tree for multiple samples is not shown until a chart has been generated.

How do I view the information associated with a particular sample in the Sample Query Results Table?

- To view the results associated with a particular sample, click on the required row of the Results Table.

The following 'branches' of the **Sample Details** Results Tree are available when the sample is from a scan task, a scan Quant task, or a Timedrive task:

Sample Details – lists the Sample ID, Sample Description, Sample Type, Analyst ID, Analyst Name, Analysis Date, and any custom fields that have been defined.

Task / Method Details – lists the Task Name, Task Description, Method Name, Method Description, Method Revision, Method Last Modified, and Method Status.

Sample Results – lists the analysis results.

Sample Spectrum has the sub-branches: Instrument Details, Instrument Settings and History.

Sample Spectrum – displays the spectrum.

Instrument Details – lists details of the instrument such as the model and Serial Number and any accessories.

Instrument Settings – lists details of the instrument settings such as Resolution and Scan Number.

History – displays the history of the sample, that is the Analyst who recorded the result, the date of the analysis and any comments entered.

Report – shows a preview of the report.

Signatures and Comments – lists any comments added to the file and shows any reviews and approvals. Signatures Points must be enabled in the software for Signatures to be displayed.

For a result from a Wavelength Program task or a Wavelength Quant task, **Sample Spectrum** is replaced by **Data**. **Data** has the sub-branches **Instrument Details**, **Instrument Settings**, and **History**.

How do I view the information associated with a particular sample in the IPV Query Results Table?

- To view the information associated with a particular validation, click on the required row of the Results Table.

The following 'branches' of the **Verifications** (IPV)Results Tree are available:

IPV Setup – lists the Setup Name, Description, Revision and Status.

Instrument – lists the Instrument Type, Serial No. and Firmware.

IPV Results – lists the User Name, User ID, Date, Result.

Spectrum – Displays the spectrum obtained from the selected IPV test. This branch also has the sub-branches **Instrument Details**, **Instrument Settings** and **History**. These sub-branches display the instrument information associated with the IPV test.

Report – Displays a preview of the IPV report.

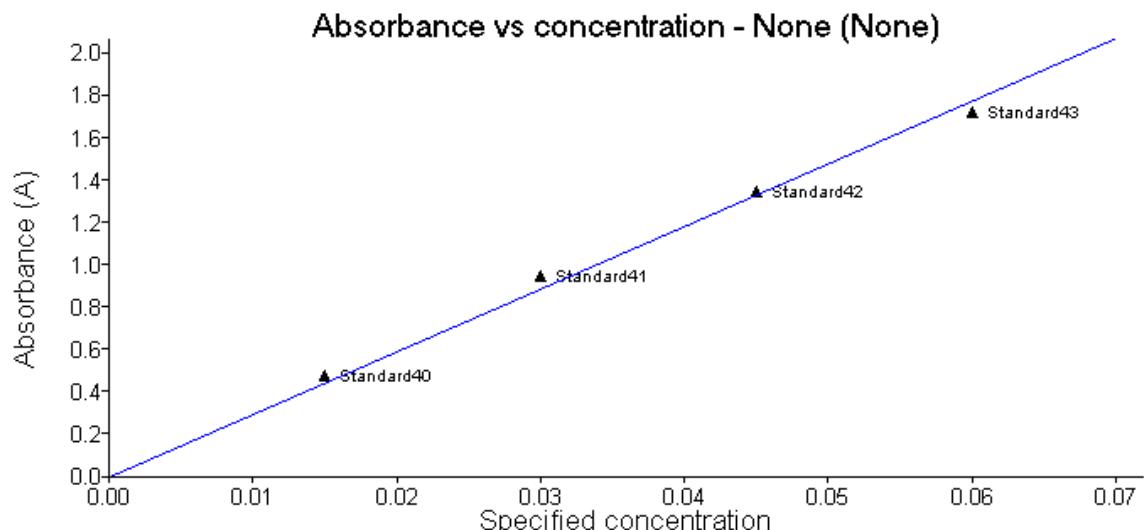
Signatures and Comments – lists any comments added to the file and shows any reviews and approvals.

How do I view the information associated with a particular sample in the Calibration Results Table?

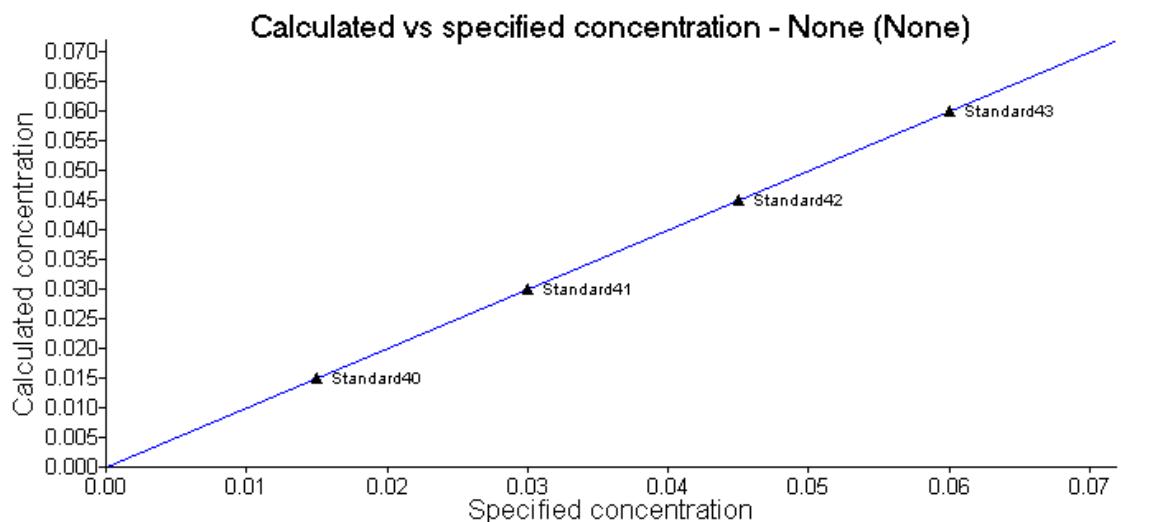
- To view the results associated with a particular sample, click on the required row of the Results Table.

The following 'branches' of the **Sample Details** Results Tree are available:

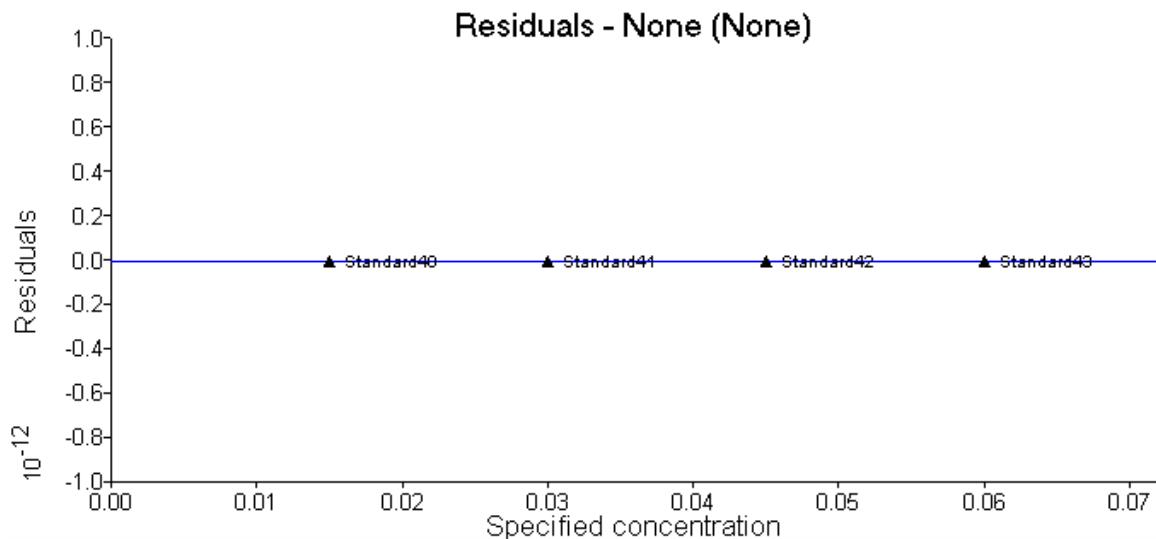
Absorbance vs Concentration graph:



Calculated vs Specified graph:



Residuals graph:



Standards Table:

Standards Table		
Standard ID	Concentration	Excluded
Standard40	0.015000	No
Standard41	0.030000	No
Standard42	0.045000	No
Standard43	0.060000	No

For each standard, wavelength(s) and value(s):

Standard	Standard40
Wavelength (nm)	Value (A)
600	0.403589
500	0.223738
400	0.482870

Instrument Details – lists details of the instrument such as the type and Serial Number and any accessories.

Instrument Settings – lists details of the instrument settings such as Slit Width, Lamp and Lamp change point.

History – displays the history of the sample, that is the Analyst who recorded the result, the date of the analysis and any comments entered.

Tasks – The tasks that use the selected calibration.

Report – shows a preview of the report.

Signatures and Comments – lists any comments added to the file and shows any reviews and approvals. Signatures Points must be enabled in the software for Signatures to be displayed.

How do I view the results associated with more than one sample?

- To view the results associated with more than one sample in the Results Table, highlight the required rows using the mouse.

NOTE: When selecting more than one sample it is only possible to select all samples or a block of samples. It is not possible to select non-contiguous samples from the Results Table. If the samples you require are non-contiguous, edit the query and use the Sort by tab to filter your data.

When more than one sample is selected it is possible to display the results using any of the following plots: Shewhart Chart, Quartile Chart, or Histogram.

NOTE: The Results Tree for multiple samples is not shown until a chart has been generated.

NOTE: Shewhart Charts and Quartile Charts are not available for IPV results.

How do I define a chart?

1. From the Actions menu select **Add Chart**, and then from the submenu select the required chart.
The dialog for the selected chart is displayed. For example, if you select **Shewhart Chart**, the Shewhart Chart dialog is displayed.
2. Select the required options and click **OK**.
The plot is displayed and is added to the Results Tree.
3. From the File menu select **Save** to save the chart as part of the Multiple Sample Results within the saved view.

NOTE: It is the type of chart and the specified parameters that are saved rather than a chart that relates to currently selected samples in the Results Table. For example, you may select all the samples in the Results Table and then display and save a Shewhart Chart as part of a particular view. If you then select a subset of the samples in the Results Table the Shewhart Chart will update to reflect the samples now selected. The next time you select more than one sample in the Results Table, the **Multiple Sample Results** Results Tree will be displayed. The Results Tree will list the branch **Shewhart Charts** and the **Name** of the plot under this branch, and the plot will reflect the currently highlighted samples in the Results Table.

It is possible to generate more than one plot for each type of chart. Each plot is listed under the appropriate sub-branch of the **Multiple Sample Results** Results Tree.

When you click on the sub-branch name, **Shewhart Chart**, **Quartile Chart**, or **Histogram**, all the Chart Names saved for the selected type of plot are listed under the selected sub-branch within the Results Tree. For each **Chart Name**, the Results Display lists the **Y Axis**, **X Axis**, and **Chart Title**.

How do I delete a chart?

1. To delete a chart from the **Multiple Sample Results** Results Tree, select the name of the chart and then right-click.
2. From the menu select **Delete**.
The selected chart is deleted.

Reviewing and Approving Results

What is meant by reviewing and approving?

The ability to formally review and approve methods, tasks, IPV setups and sample results is a function of the Enhanced Security version of UV WinLab.

It is up to the UV WinLab Administrator to set the correct privileges to ensure that only the appropriate people can 'sign off' data.

We use the term 'Review' to mean that the person has looked at the data and has agreed that it is correct. This is along the lines of a peer review and any number of people can review data as determined by your internal procedures.

We use the term 'Approve' to mean that a person with the 'authority' has signed off the data as fit-for-purpose and again details of who is allowed to do this should be documented in your internal procedures.

How do I review or approve a result?

Reviewing and Approving results is a function of the Enhanced Security version of UV WinLab and can only be performed by someone with the appropriate permission.

Review and Approve Signature Points must be enabled for these actions to be recorded in the Signatures and Comments dialog.

1. Select one or more result from the Results Table that you wish to review or approve.
2. From the Tools menu select **Review** or **Approve**.
3. Enter your **User name** and **Password**.
4. Select the **Reason** for the review or approval.
5. Enter any **Comment** required.
6. Click **OK**.

NOTE: It is possible for more than one person to approve a result.

How can I see if a result has been reviewed or approved?

Review and Approve Signature Points must be enabled for these actions to be recorded in the Signatures and Comments dialog.

- To view the list of reviews/approval, select the required sample and then select **Signatures and Comments** from the Results Tree view.
Each Review or Approval is listed.

How do I add a Comment to a result?

Adding Comments can be done in the Enhanced Security and Standard versions of UV WinLab.

1. Select the result(s).
2. From the Tools menu select **Comments**.
3. Enter a **Comment** and click **OK**.

The Comment can be viewed in the **Signatures and Comments** section of the Tree.

NOTE: A signature is not required when adding a comment.

Add Spectra

Spectra can be added to the graph in the Results Display. The spectra are for comparison only and are not saved as part of the task.

How do I add a spectrum to the graph?

1. Select **Add Spectra** from the Actions menu.

The Add Spectra dialog is displayed.

2. Select the spectrum (or spectra) to import.

3. Click **Open**.

The spectrum is displayed on the graph.

NOTE: The spectrum is for comparison only and is not saved as part of the task.

Report Templates



A default report template is provided with each example method. Also, when a new method is created using the New Method Wizard, a report template is automatically attached. These templates can be edited as required or new templates can be created. For further information on creating report templates see Communiqué Report Creator section within this Help.

NOTE: If you are using the Enhanced Security version of UV WinLab, a report template must be approved before a report can be saved to the database. For further information see Communiqué Report Creator section within this Help.

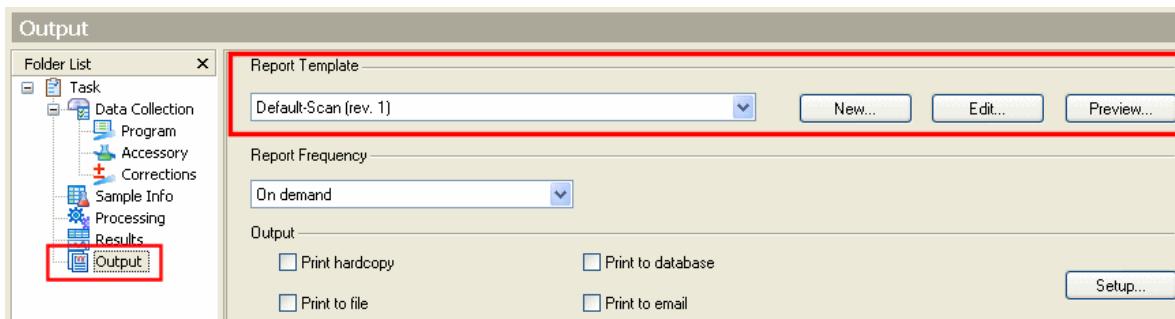
Where can I view a list of all the available report templates?

All the available report templates are listed under **Report Templates** within the Explorer.

The Explorer lists the Report Template Name, Type, Created on, Created by, and Status.

When you highlight a Report Template, the Display Pane lists Name, Description, Type, Created by, Created on, Revision and Status.

A list of all Report Templates is also available from the Output page within the Workspace, by selecting the **Report Template** drop-down list.



How can I open a report template?

- Double-clicking on a report template in Explorer opens the report template in Communiqué Report Creator.

OR

Right-click on a report template in Explorer and select **Open** from the context menu.

OR

Select the report template and press Enter.

The template can then be edited as required (if you have the necessary permission). See the Communiqué Report Creator section of the Help for further information.

How do I delete a report template?

- In the Explorer, highlight the name of the template and from the File menu select **Delete**.

OR

Right-click on the template and select **Delete** from the context menu.

You are asked to confirm the deletion.

How do I review or approve a report template?

Reviewing and approving can only be performed from the Print Preview dialog within Communiqué.

1. From Explorer, double-click on the report template to open it.
The report template opens in Communiqué.
2. From the File menu select **Print Preview**.
The report template is shown in the Print Preview window.
3. Click .
The Approve dialog is displayed.
4. Select the type of approval (**Approve Template** or **Review Template**) from the drop-down list and click **OK**.
The Approve Template or Review Template dialog is displayed.
5. Enter your **User name** and **Password**, select a **Reason** from the drop-down list and enter a **Comment** (if required).
The fields displayed on this dialog depend on whether you are using the Standard or Enhanced Security version of UV WinLab.
6. Click **OK**.
The dialog closes and the Review or Approval is recorded.

How can I see who has reviewed or approved a report template?

1. From Explorer, double-click on the report template to open it.
The report template opens in Communiqué .
2. From the File menu select **Print Preview**.
The report template is shown in the Print Preview window.
3. Click ✓.
The Approve dialog lists all the reviews and approval of the template. For each, the following information is recorded – Name, Time Stamp, Type (Review/Approve), Reason, E-Sig (Yes/No), Comments.

What menu items are available when I right-click on a report template?

The following menu items are available :

Open	Opens Communiqué and displays the template.
Delete	Deletes the item to the recycle bin.

What are the statuses of report templates?

A report template can have the status **Approved**, **Unapproved** or **Hold**.

Unapproved is the initial status of a report template, **Approve** locks it, **Hold** (reports only) rejects it.

Reports

Reports lists all the reports that have been saved to the database. The Main pane shows the reports available, while the Display pane displays a preview of the currently selected report.

NOTE: In the Enhanced Security version of UV WinLab, reports can only be saved to the database if they use a report template that has been approved. For further information on approving report templates see the Communiqué Report Creator section of the Help.

How do I view a previously created report?

1. In the UV WinLab Explorer select **Reports** from the Folder List.
The Main pane lists all the reports currently saved to the database.
2. Double-click on a report to open it in the Communiqué print preview window.

NOTE: It is not possible to edit a report. For information on printing a saved report see the Communiqué Report Creator section of this Help.

Recycle Bin



Selecting **Recycle Bin** on the Folder List displays a list of everything in the Recycle Bin.

How do I empty the recycle bin?

- From the File menu select **Empty Recycle Bin**.

OR

Right-click in the Main pane and select **Empty Recycle Bin**.

The recycle bin is emptied.

NOTE: You must have permission to Manage the database in order to be able to empty the recycle bin. Emptying the Recycle Bin empties all the contents.

How do I restore a file from the recycle bin?

- Select the file and then from the File menu select **Restore**.

OR

Right-click on the item in the Main pane and select **Restore** from the menu.

The file is restored to its original location.

Database Tools

Database Tools are used by the Database Manager to:

- set the active database
- compact the database after deletion of methods to free up disk space
- create a new empty repository database
- register an existing database on the system
- perform an integrity check on a database
- perform a partial archive.

How do I select and use Database Tools?

1. From the Start menu select PerkinElmer Applications\UV WinLab\Database Tools.
The Database Tools application starts.
2. Select the required type of database by clicking on the icon in the left-hand panel.
The list of available databases of that type is displayed with a tick in a green circle showing the current database.
The database tools available depend on the type of database selected.
3. Click the button for the database tool required:
Set Active Database – Sets the selected database to be the active one.

NOTE: You will need to close down UV WinLab and start it again to activate the database.

Compact Database – Compacts the files to free up disk space.

Create Database – Creates a new database.

Register Database – Enables you to connect the PC to a database already in place on a network.

Check Database – Performs an integrity check on the database to ensure it has not been tampered with or corrupted. A log file will be generated.

Partial Archive – Allows you to archive the database but leave the methods available for future use.

Archive Utilities – Loads the Communiqué archiving utility.

The following table lists the tools that are available for each type of database. X indicates that the option is available.

	Set active database	Compact database	Create database	Register database	Check database	Partial archive	Archive Utilities
UV WinLab	X	X	X	X	X	X	
Security		X		X			
Communiqué	X	X	X	X			X

NOTE:Check database is available in the Enhanced Security version of UV WinLab only.

How do I create a new Database?

We recommend that you do not create a new Communiqué database, but instead archive this database at regular intervals. If you create a new database it will not contain the default templates. If, for example, you create a new Communiqué database, then perform an IPV and set reporting to database, you will get a report template error message because the report templates are not available. If you really do need to create a new Communiqué database, the required templates can be imported from C:\Program Files\PerkinElmer\UVWinLab\6.0\Comumniqe Templates.

1. From the Database Tools dialog, select the required type of database by clicking on the icon in the left-hand panel and then click **CreateDatabase**.
The Create Database dialog is displayed.

NOTE:The database tools available depend on the type of database selected. Certain types of database may not allow you to create new ones.

2. In **Database name**, enter the name for the database.
This is the name that will appear in the Database Tools dialog.
3. In **Path**, enter the full path for the new database including the file name required.

OR

Click **Browse**, use the file selector displayed to select where the database will be stored, enter the **File name** required, and click **Save**.

4. Click **OK**.
The new database is created and is added to the list in the Database Tools dialog.
You can now make this the default database by selecting it and then clicking **Set Active Database**.

NOTE: You will need to close down UV WinLab and start it again to activate the new database. You do not need to register the database.

What is a partial archive?

Partial archiving allows you to archive your database but leave your methods for future use.

1. Click Partial Archive.
2. In **Path**, enter the full path for the archive database, including the file name required.
OR
Click **Browse** and then select the path and enter a file name.
3. Click **OK**.
The archive is performed. The archived database appears in the database list.

How do I register a Database created previously?

Databases created using a different copy of UV WinLab software and either copied for your use or held on a network, must be registered with your copy of UV WinLab:

1. From the Database Tools dialog, select the required type of database by clicking on the icon in the left-hand panel and then click **RegisterDatabase**.

The Register Database dialog is displayed.

NOTE: The database tools available depend on the type of database selected. Certain types of database may not allow you to register new ones.

2. In **Database name**, enter a name for the database.

This is the name that will appear in the Database Tools dialog.

3. In **Path**, enter the full path of the database including the file name.

OR

Click **Browse**, use the file selector displayed to select the required database and click **Open**.

4. Click **OK**.

The database is now registered by the software and is added to the list in the Database Tools dialog.

You can now make this the default database by selecting it and then clicking **Set Active Database**, but you will then need to close down UV WinLab and start it again to activate the database.

What databases are used in UV WinLab?

It is essential that backups are made regularly of key files and databases in order to secure the data in case of computer failure or accidental loss or damage, or even intentional damage.

The following files/directories must be backed up regularly:

- ...\\PerkinElmer\\Security System\\Users.mdb
This is the database of users.
- ...\\PerkinElmer\\Security System\\Backup\\Users.bak
This is a backup of users.mdb)
- ...\\PerkinElmer\\UVWinLab\\Communique.mdb
This is the database of reports and report templates.
- ...\\PerkinElmer\\UVWinLab\\UVWinLab.mdb
This is the database of methods and tasks.

Where "..." represents:

For Windows XP, C:\Documents and Settings\All Users\Application Data

For Windows 7, C:\Program Data

How do I recover from Checksum failures?

UVWinLab uses a variety of security techniques to ensure that files cannot be tampered with either accidentally or deliberately – one of these is to use checksums to ensure the data has not been tampered with. Under normal operation, checksums are used in the application to validate the data security. However, a checksum failure can occur after a number situations:

- Hard disk failure
- Power failure
- Software crash, either the application or Windows or another application
- Deliberate attempt to falsify data
- Accidental data falsification.

The only remedy to an error message stating there is a checksum failure is to restore from a backup database.

UVWinLab database

If the UVWinLab Local Repository gets a checksum failure, the software will continue operating but the data in error will not be visible. You will receive an error message if you try to open an invalid task or method.

It is essential that backups are regularly made of the Repository in order to recover from a checksum failure. The Windows Administrator can restore from this backup if the Repository becomes corrupt as follows:

1. Log on as Windows Administrator.
2. Rename or move UVWinLab.mdb from ...\\PerkinElmer\\UVWinLab

Where "..." represents:

For Windows XP, C:\Documents and Settings\All Users\Application Data

For Windows 7, C:\Program Data\\PerkinElmer\\UVWinLab

3. Copy your backup file as UVWinLab.mdb to replace the old one.

It should then be possible to view all the data again. Some data may be lost if there were any changes to the database that were not backed up.

Security database

The security system automatically backs up the users.mdb database at the end of a session, and on exit from the administration dialog, in a subdirectory called \Backup:
...\\PerkinElmer\\Security System\\backup\\Users.bak

Where "..." represents:

For Windows XP, C:\\Documents and Settings\\All Users\\Application Data

For Windows 7, C:\\Program Data

The Windows Administrator can restore from this database if the active database becomes corrupt and gives a checksum failure as follows:

1. Log on as Windows Administrator.
2. Rename or move Users.mdb
3. Copy ...\\Backup\\Users.bak as Users.mdb to replace the old one.

It should then be possible to log on again. Some data may be lost if there were any changes to the database that were not backed up.

NOTE: Backups of Users.mdb and UV WinLab.mdb must be made at the same time to ensure they are synchronized. If this does not happen, and a problem occurs that requires an older version of one of the databases to be restored, the methods folder in the UV WinLab Explorer may not be visible.

Legacy File Converter

21 CFR Part 11 technical compliance mandates very high levels of data integrity and security. To ensure that UV WinLab only accesses and uses data acquired on a 21 CFR Part 11 compliant system, a data security checksum has been added to the spectrum data file.

Spectra without this 21 CFR Part 11 checksum will not be read into the system and cannot be processed. This feature stops data from older data systems from being automatically used in new compliant systems.

To allow users access to their legacy data a Conversion Utility has been included as part of the Administration tools. The Legacy File Converter allows a person with the **Manage the database** permission to add a data security checksum to a legacy spectrum.

NOTE: Use of the utility should be highly controlled and spectra that are processed should have full supporting GxP provenance as part of their audit trail.

NOTE: You must have permission to **Manage the database** to be able to access the Legacy File Converter.

What type of data can be converted?

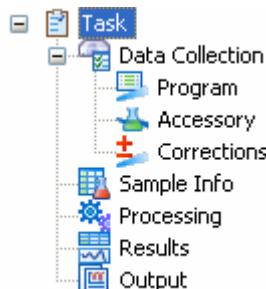
The legacy file converter will convert spectral (.sp) and timedrive (.td) files.

How do I convert legacy data?

1. Select **Legacy File Converter** from the Administration menu within the Explorer.
The Legacy File Converter is displayed.
2. For the Source Path, click **Browse** and on the file selector displayed, select the **Source Path** for the directory containing the legacy data.
3. For the Destination Path, click **Browse** and on the file selector displayed, select the **Destination Path** for the directory containing the converted data.
4. Click **Next**.
The data is copied and a checksum is added to each file, then the new files are written to the destination directory leaving the original data untouched. The default text _cs is appended to each filename.
5. To view information about the conversion, click **View Log**.
A log file is displayed.

The Workspace

The Workspace



The Workspace is shown below. The Workspace is where the analysis of samples takes place. A task is triggered from the Explorer – double-clicking on a Method or a Task opens the Workspace.

UV WinLab - Run - Scan - Lambda 45 20 November 2007 09:46 GMT Standard Time

File Edit View Data Collection Tools Help

Open Cut Copy Paste Report Send To DPV Start Stop Set λ Autozero Align

Idle	700.00 nm	0.022 A	Slit width 1.00 nm
------	-----------	---------	-----------------------

Data Collection

Folder List

- Task
 - +
 - Data Collection
 - Sample Info
 - Processing
 - Results
 - Output

Scan Settings

Start (nm) 700	End (nm) 200	Ordinate mode A	Slit width (nm) 1
-------------------	-----------------	--------------------	----------------------

Advanced Scan Settings

Scan speed (nm/min) 480	Data interval (nm) 1
----------------------------	-------------------------

Cycles

Number of cycles 1	Cycle as fast as possible
	<input checked="" type="radio"/> Cycle time
	1 seconds

Lamps

<input checked="" type="checkbox"/> UV lamp on	Lamp change at (nm) 326
<input checked="" type="checkbox"/> Visible lamp on	

45 | PerkinElmer Method Developer

NOTE: If you cannot see any of the menu items listed below it is because you do not have the necessary permission. Permissions are assigned by the UV WinLab Administrator. Please contact your UV WinLab Administrator for further details about your permissions.

What do the commands on the File menu do?

Open	Enables you to import samples. Open is only available when Sample Info is selected in the Folder List.
Import Table	Enables you to import a sample table. Import Table is only available when Sample Info is selected in the Folder List.
Save Results	Submenu of To Task and As New Task . Saves changes to the existing Task or to a new Task.
Save Settings	Submenu of To Method and As New Method . Saves changes to the existing Method or to a new Method.
Save Spectra	Displays the Save Spectra dialog, which enables you to save selected samples in *.sp/*.td format.
Export	Displays the Export Data dialog. Allows you to export the data from a Task at any time, rather than at the end of run. For more information on the Export Data dialog and the file formats available, see Data Export.
Export Table	Enables you to export a sample table. Export Table is only available when Sample Info is selected in the Folder List.
Send To	Submenu of UV WinLab DPV . Sends the raw data, processed data and the results of any equation to UV WinLab Data Processor and Viewer (DPV) and opens the application.
Report Preview	Displays the Communiqué Print Preview window.
Report	Generates a report with the selected Communiqué template and prints or saves the report depending on how defined in the setup.
Exit	Closes the Task Workspace.

What do the commands on the Edit menu do?

Cut	Cuts the selected item(s). Only available when one or more items are selected.
Copy	Copies the selected item(s). Only available when one or more items are selected.
Paste	Pastes the items from the clipboard. Only available when there is something on the clipboard.

What do the commands on the View menu do?

Go To	Submenu of all the current Folder List nodes.
Toolbars	Submenu of Standard and Data Collection . A check mark is displayed when a toolbar is switched on.
Folder List	Switches the Folder List on and off. A check mark is displayed when the Folder List is switched on.
Instrument Status Bar	Switches the Instrument Status Bar on and off. A check mark is displayed when the Instrument Status Bar is switched on.
Status Bar	Switches the Status Bar on and off. A check mark is displayed when the Status Bar is switched on.
Overlay Samples	Switches the Graph mode from one sample per graph to showing all samples and vice versa.

What do the commands on the Data Collection menu do?

Start	Starts collecting data from the instrument.
Stop	Stops data collection.
Set Wavelength	Opens the Set Wavelength dialog.
Autozero	Zeros the instrument.
Alignment Mode	Activates Alignment Mode.
Insert Event Marker	Inserts an event marker into the scan time to allow you to record an event as it happens. This is only available when a Timedrive Method is open.

What do the commands on the Tools menu do?

Display	Submenu of SOP, Task Summary and Audit Trail. SOP displays the SOP that has been attached to the method. Task Summary displays a summary of the Task in the Communiqué Print Preview window using a default template. Audit trail displays the audit trail for a Method.
Lock	Locks the method. Menu items toggles between Lock and Unlock .
Review	Submenu of Method and Task. Allows you to review the selected Method or Task.
Approve	Submenu of Method and Task. Allows you to approve the selected Method or Task.
Event log	Submenu of Method and Task. Displays the event log for a Method or Task.
Options	Displays the Options dialog.

What do the commands on the Help menu do?

Contents and Index	Displays the on-screen HTML Help system.
Tutorials	Displays the on-screen HTML tutorials.
PerkinElmer on the Web	Goes to the PerkinElmer home page.
About	Displays the About box with version information.

What menu commands are available as Toolbar buttons?

	Open
	Cut
	Copy
	Paste
	Report
	Send to DPV
	Start
	Stop
	Set Wavelength
	Autozero
	Align (Alignment Mode)
	Event Marker

What commands are available when I run a method, edit a method, view a method, view a task, reprocess a task, or continue a task?

Below is a table summarizing the commands available depending on whether you run a method, edit a method, view a method, view a task, reprocess a task, or continue a task.

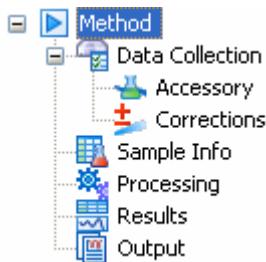
Action	Run Method	Edit Method	View Method	View Task	Reprocess a Task	Continue a Task
Open	yes	yes	no	no	no	yes
Import Table	yes	yes	yes	no	yes	yes
Save Results to Task menu command	yes if data collected	yes if data collected	no	no	yes (if using the standard version of UV WinLab)	yes (if using the standard version of UV WinLab)
Save Settings to Method menu command	yes if you have permission to edit methods	yes	no	no	no	no
Save Results As New Task menu command	yes if data collected	yes if data collected	no	no	yes	yes
Save Settings As New Method menu command	yes if you have permission to edit methods	yes	yes	yes if you have permission to edit methods	yes	yes
Save Spectra menu command	yes if data collected	yes if data collected	no	yes	yes	yes
Export	yes if data collected	yes if data collected	no	yes	yes	yes
Export Table	yes	yes	yes	yes	yes	yes
Cut (menu and toolbar)	yes	yes	no	no	yes	yes

Copy (menu and toolbar)	yes	yes	no	no	yes	yes
Paste (menu and toolbar)	yes	yes	no	no	yes	yes
Start (menu and toolbar)	yes	yes	no	no	no	yes
Stop (menu and toolbar)	yes	yes	no	no	no	yes
Set Wavelength (menu and toolbar)	yes	yes	no	no	no	yes
Autozero (menu and toolbar)	yes	yes	no	no	no	yes
Alignment Mode (menu and toolbar)	yes	yes	yes	no	no	yes
Insert event marker (menu and toolbar)	yes	yes	no	no	no	yes
Folder List	yes	yes	yes	yes	yes	yes
Instrument Status Bar	yes	yes	yes	yes	yes	yes
Status bar	yes	yes	yes	yes	yes	yes
Toolbar Standard	yes	yes	yes	yes	yes	yes
Toolbar Data Collection	yes	yes	yes	yes	yes	yes
SOP	yes	yes	yes	yes	yes	yes
Audit Trail	yes	yes	yes	yes	yes	yes

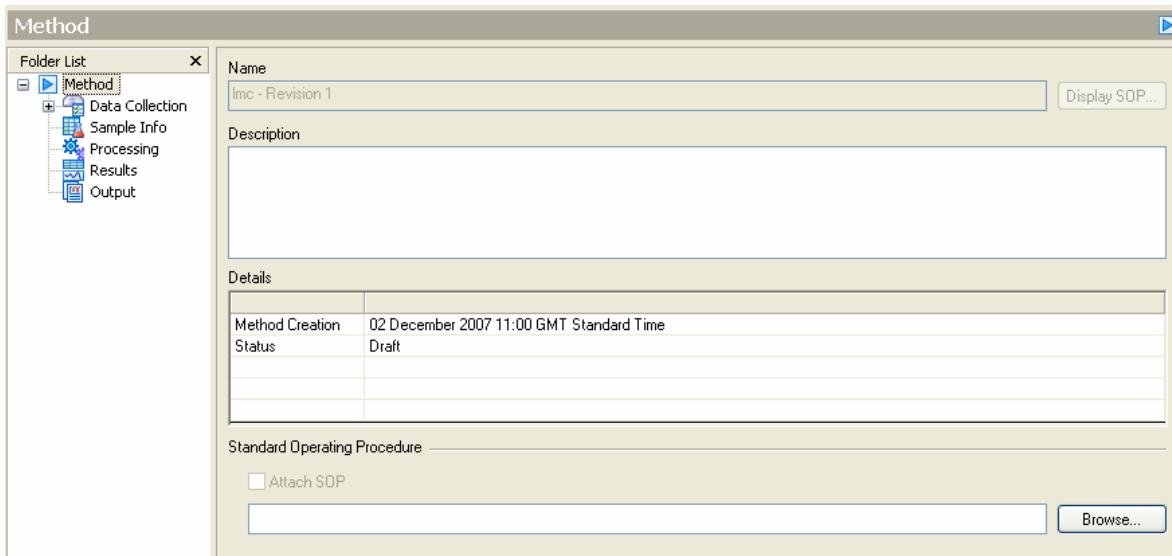
Event Log Method	yes	yes	yes	yes	yes	yes
Event Log Task	yes	yes if data collected	yes	yes	yes	yes
Options	yes	yes	yes	yes	yes	yes
Lock	no	yes if permission	no	no	no	no
Review Method	no	yes if permission	yes if permission	no	no	no
Review Task	yes if permission	yes if permission	no	yes if permission	yes if permission	yes if permission
Approve Method	no	yes if permission	yes if permission	no	no	no
Approve Task	yes if permission	yes if permission	no	yes if permission	yes if permission	yes if permission
Task Summary	yes	yes	yes	yes	yes	yes

NOTE: It is not possible to reprocess a task if the method used to create the task has been locked, reviewed or approved.

Method



This is the top level page of the Method in the Workspace when a method is opened in Edit or View mode. The Name, Description and Details of the Method are displayed, along with details of any Standard Operating Procedure that may be associated with the method.



Can I change the Method Name?

No, you cannot rename Methods. To change the name of a Method, you must save the results as a new Method.

- Select **Save Settings** from the File menu and then **As New Method** from the submenu.

How do I add a Description of the Method?

Enter the required text in the Description field. When the Method is selected in the Explorer, the Description is displayed (along with other information) in the lower pane.

How do I attach a Standard Operating Procedure?

1. Click **Browse**.
The Open dialog is displayed.
2. Locate the file and click **Open**.
The document path is displayed.
3. Select **Attach SOP** to attach the document to the method.

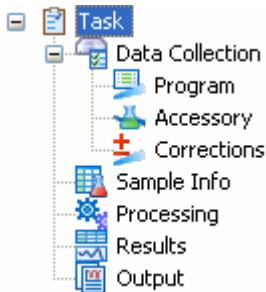
NOTE: The **Attach SOP** check box is not available unless the document path has been specified.

How do I view the Standard Operating Procedure that is attached to the method?

- Click **Display SOP**.
The document (specified at the bottom of the dialog) is opened.

NOTE: The **Display SOP** button is grayed unless an SOP document has been attached.

Task



This is the top level page of the task when it is opened in the Workspace. The Name, Description and Details of the task are displayed.

Can I change the Task Name?

No, you cannot rename Tasks. To change the name of a Task, you must save the results as a new Task.

- Select **Save Results** from the File menu and then **As New Task** from the submenu.

How do I add a Description of the Task

Enter the required text in the Description field. When the Task is selected in the Explorer, the Description is displayed (along with other information) in the lower pane.

How do I view the Standard Operating Procedure that is attached?

- Click Display SOP.
- The document (specified at the bottom of the dialog) is opened.

NOTE: The **Display SOP** button is grayed unless a SOP document has been attached.

The Instrument Status Bar

What is the Instrument Status Bar?

The Task Workspace has a special status bar that provides the status of the system and the instrument readings.

Move the mouse over an area of the picture below to find out about each region of the Instrument Status Bar.

Idle	550.00 nm 10.000 A	Slit width 1.00 nm	Cell 2	Temp. 27.7	Ext. Temp. 20.0
------	-------------------------	-----------------------	-----------	---------------	--------------------

NOTE: If the detector is overloaded during a scan, an exclamation mark appears next to the ordinate value, together with a 'Invalid data: overflow reference' tooltip.

What types of messages are displayed?

The following messages may be displayed depending upon the current operation:

Initializing – the instrument is initializing.

Idle – the instrument is switched on but idle.

Scanning/Performing Autozero – when any type of data is being collected.

Setting Up – when settings are applied to the instrument and/or accessories.

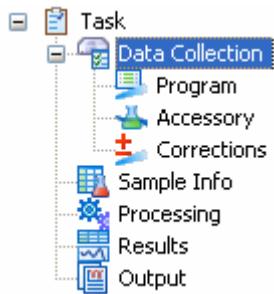
Other messages may also be displayed, depending on the accessory installed.

When the Universal Reflectance Accessory (URA) is installed, the slit width and angle are displayed.

During **Initializing** or **Setting Up** operations, a bar is displayed that indicates the operation is in progress:



Data Collection



The information on the Data Collection page depends on the type of task you are creating. Basic method settings such as wavelength, data interval (depending on the method type) can be set on the Data Collection page as well as more advanced settings.

Medium Performance Instruments Data Collection

The information on the Data Collection page depends on the type of task you are creating. The basic method settings (for example, wavelength, data interval, depending on the method type) can be set on this page.

NOTE: Scanning Quant and Wavelength Quant are derivatives of Scan and Wavelength Program, respectively, and so are not covered separately here.

See the following for information on the Data Collection page relating to your chosen task:

- Scan Task – Data Collection
- Timedrive Task – Data Collection
- Wavelength Program – Task Data Collection
- Manual Control – Data Collection

Scan Task – Data Collection

The Data Collection page allows you to define settings such as the scan range, **Ordinate mode** and **Slit width**.

What settings are available on the Data Collection page?

Start	Enter the Start of the scan range (nm).
End	Enter the End of the scan range (nm).
Ordinate mode	Select the Ordinate mode from the drop-down list.
Slit width	Select the Slit width from the drop-down list.
Scan speed	Enter the Scan speed for the scan (nm/min).
Data interval	Enter the Data interval for the scan (nm).
Number of cycles	Enter the Number of cycles (repeats) of the scan.
Cycle as fast as possible	Enables you to select the fastest cycle time. The time is displayed.
Cycle time	Enables you to set your own cycle time. Select Cycle time and enter a value in the field below. Select the units from the drop-down list. The time must be longer than the minimum cycle time.
UV lamp on	Select whether the UV lamp is on or off.
Visible lamp on	Select whether the Visible lamp is on or off.
Lamp change at (nm)	Enter the wavelength at which the lamps are to be switched.

How do I set the scan range?

- Enter the values for the Start and End of the range in the **Start** and **End** fields respectively.
The Start value must be greater than the End value.

How do I select the Ordinate mode?

- Select **A** (absorbance), **%T** (transmittance), **E1** (energy from sample beam), **E2** (energy from reference beam), or **%R** (reflectance) from the drop-down list.

How do I select the Slit width?

- Select **0.5, 1, 2, or 4 nm** from the drop-down list.

NOTE: The slit width is fixed on a Lambda 20 and Lambda 25. The drop-down list is grayed.

What Slit width should I use?

The slit width for the UV/Vis range is in nm.

- Set a slit width that is between one-fifth and one-tenth of the bandwidth of the spectral feature of interest.
- Set a wide slit to increase the energy throughput and the signal-to-noise ratio. This decreases the resolution and the photometric accuracy and can cause band broadening.
- Set a narrower slit to increase the resolution and the photometric accuracy. This decreases the signal-to-noise ratio.

How do I select the Scan speed?

- Select the required **Scan speed** from the drop-down list of available scan speeds. Use slow scan speeds for narrow bands and to improve the signal-to-noise ratio. Use faster scan speeds for broad bands.

How do I set the Data interval?

- Enter the required **Data interval** (in nm) in the field. The Data interval is set automatically if fast scan speeds are selected.

How do I define the Cycles?

It is possible to enter the Number of cycles, and then select either **Cycle as fast as possible**, or enter a **Cycle Time**.

- **Number of cycles** – Enter the number of cycles to be used. As the results are generated, each result includes .cycleX at the end of the name, where X is the number of the cycle.
- **Cycle as fast as possible** – As soon as a cycle finishes the next one begins.
- **Cycle Time** – Select the time unit, in **seconds**, **minutes** or **hours**, and enter the number in the field to the left of the unit. The time you enter must be more than the minimum, and in effect you are introducing a delay before the next cycle begins.

NOTE: Cycles are not used in Quant.

How do I select the lamps and the lamp change wavelength?

1. Select **UV lamp on** and **Visible lamp on** as required. A check mark indicates that the lamp is on.
2. If you want to change the lamps at another wavelength edit the default value. Edit the default value to shift the wavelength if a feature of special spectral interest is located at the default change wavelength.

NOTE: After switching the UV lamp off, always allow it to cool for at least 5 minutes before switching it on again. This prolongs the life of the UV lamp. In addition, where possible, only switch the lamp off at the end of the working day.

What happens if I am scanning 1100–190 nm with the UV lamp on and the Visible lamp off?

Only the UV lamp is used for the scan. Noise will be observed in the region where the Visible lamp is required.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

- Deselect Alignment mode.

See also

- Graphs

Wavelength Task – Data Collection

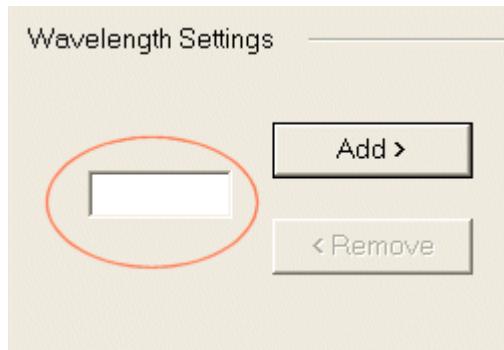
The Data Collection page allows you to define the Wavelengths, the **Ordinate mode**, **Slit width**, **Response** and **Cycles**. The table in the lower half of the Sample Info page contains the samples as set up on the Sample Info page, and the wavelengths defined for the task. The results are entered in this table.

What settings are available on the Data Collection page?

Wavelength Settings	Enter the required wavelengths (nm).
Ordinate mode	Select the Ordinate mode from the drop-down list.
Slit width (nm)	Select the Slit width from the drop-down list.
Response (s)	Enter the Response time in seconds.
Number of cycles	Enter the Number of cycles (repeats) of the scan.
Cycle as fast as possible	Enables you to select the fastest cycle time. The time is displayed.
Cycle time	Enables you to set your own cycle time. Select Cycle time and enter a value in the field below. Select the units from the drop-down list. The time must be longer than the minimum cycle time.
UV lamp on	Select whether the UV lamp is on or off.
Visible lamp on	Select whether the Visible lamp is on or off.
Lamp change at (nm)	Enter the wavelength at which the lamps are to be switched.

How do I add a wavelength?

1. Enter the wavelength in the field adjacent to the **Add** and **Remove** buttons:



2. Click **Add**.

The wavelength is added to the bottom of the list.

NOTE: The order in which the wavelengths are listed will be the order in which they are scanned.

NOTE: The default wavelengths listed in the table when the method is opened are dependant on the method being used to create the task.

How do I delete a wavelength?

1. From the list of wavelengths, click on the wavelength that you wish to delete.
The wavelength is highlighted to show that it is selected.
2. Click **Remove**.
The wavelength is removed from the list.

How do I re-order the list of wavelength settings?

The order of the wavelengths in the table will determine the order in which they are scanned.

To re-order to wavelengths:

1. From the list of wavelengths, click on the wavelength that you wish to move.
The wavelength is highlighted to show that it is selected.
2. Click **Move Up** to move the selected wavelength up one position in the list of wavelengths.

OR

Click **Move Down** to move the selected wavelength down one position in the list of wavelengths.

NOTE: **Move Up** and **Move Down** are only available when a wavelength is selected from the list.

How do I select the Ordinate mode?

- Select **A** (absorbance), **%T** (transmittance), **E1** (energy from sample beam), **E2** (energy from reference beam), or **%R** (reflectance) from the drop-down list.

How do I select the Slit width?

- Select **0.5, 1, 2, or 4 nm** from the drop-down list.

NOTE: The slit width is fixed on a Lambda 20 and Lambda 25. The drop-down list is grayed.

What Slit width should I use?

The slit width for the UV/Vis range is in nm.

- Set a slit width that is between one fifth and one tenth of the bandwidth of the spectral feature of interest.
- Set a wide slit to increase the energy throughput and the signal-to -noise ratio.
This decreases the resolution and the photometric accuracy and can cause band broadening.
- Set a narrower slit to increase the resolution and the photometric accuracy.
This decreases the signal-to-noise ratio.

What is Response and what values are available?

The **Response** time (in seconds) is the time that is spent scanning at the selected wavelength. For example, if the Response is set to 10 seconds, the spectrometer will scan for 10 seconds at a selected wavelength before moving onto the next wavelength.

- Select **0, 0.1, 0.2, 0.5, 1, 2, 5 or 10** from the drop-down list.

Zero means that the wavelength is scanned as fast as possible.

How do I define the Cycles?

It is possible to enter the Number of cycles, and then select **Cycle as fast as possible**, or enter a **Cycle Time**.

- **Number of cycles** – Enter the number of cycles to be used. As the results are generated, each result includes .cycleX at the end of the name, where X is the number of the cycle.
- **Cycle as fast as possible** – as soon as a cycle finishes the next one begins.
- **Cycle Time** – select the time unit in **seconds, minutes or hours** and enter the number in the field to the left of the unit. The time you enter must be more than the minimum, and in effect you are introducing a delay before the next cycle begins.

NOTE: Cycles are not used in Quant.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

- Deselect Alignment mode.

How do I export the information displayed in the table at the bottom of the Sample Info page when I am working with a Wavelength Task?

The data in the table can be copied to the clipboard.

- Right-click anywhere in the table and select **Copy To Clipboard**.

All the data in the table is copied to the clipboard, It can then be directly pasted into another application e.g. Excel.

Timedrive Task – Data Collection

The Data Collection page allows you to define settings such as the **Wavelength**, **Ordinate mode**, **Slit width**, **Time interval** and **Total time**.

What settings are available on the Data Collection page?

Wavelength (nm)	Enter the Wavelength at which readings will be taken (nm).
Ordinate mode	Select the Ordinate mode from the drop-down list
Slit width (nm)	Select the Slit width from the drop-down list
Time interval (s)	Enter the Time interval between readings (s).
Total time	Enter the duration of the experiment. Choose the units from the drop-down list – seconds , minutes or hours .
Response (s)	Enter the Response time (s), over which data will be collected and averaged.
UV lamp on	Select whether the UV lamp is on or off.
Visible lamp on	Select whether the Visible lamp is on or off.
Lamp change at (nm)	Enter the wavelength at which the lamps are to be switched.

How do I define the Wavelength for the task?

The **Wavelength** is the wavelength at which the readings will be taken.

- Enter the required wavelength (nm) in the **Wavelength** field

How do I select the Ordinate mode?

- Select **A** (absorbance), **%T** (transmittance), **E1** (energy from sample beam), **E2** (energy from reference beam), or **%R** (reflectance) from the drop-down list.

How do I select the Slit width?

- Select **0.5**, **1**, **2**, or **4 nm** from the drop-down list.

NOTE: The Slit width is fixed for a Lambda 25, Lambda 20, and Lambda 20 Bio.

What Slit width should I use?

The slit width for the UV/Visible range is in nm.

- Set a slit width that is between one fifth and one tenth of the bandwidth of the spectral feature of interest.
- Set a wide slit to increase the energy throughput and the signal-to -noise ratio.
This decreases the resolution and the photometric accuracy and can cause band broadening.
- Set a narrower slit to increase the resolution and the photometric accuracy.
This decreases the signal-to-noise ratio.

How do I define the Time interval?

The Time interval is the interval (in seconds) between readings (data points) during the data collection.

- Enter a value in the **Time interval** field.

How do I define the Total time?

The **Total time** is the duration of the experiment. The **Total time** must be a multiple of the **Time interval**.

- Enter a value in the **Total time** field and select the time units from the drop-down list (**seconds, minutes or hours**).

Can I extend the Total time when I am in the middle of collecting data?

Yes, the **Total time** can be extended while data is still being collected for a particular sample.

This new time will also be applied to samples that have not yet been run.

What is Response and what values are available?

The **Response** time in a Timedrive task is the amount of smoothing. (It is the length of time over which the data is collected and averaged.)

- Select **0, 0.1, 0.2, 0.5, 1, 2, 5 or 10** from the drop-down list.

A response time of zero means that there is no smoothing. The data is true raw data. It is collected as fast as possible.

NOTE: When a Cell Changer is installed, the **Response time** is the time for which the data is collected. For example, there are two samples in the Cell Changer; the **Time interval** is set to 60 seconds and the **Response time** is set to 10 seconds. The instrument will take reading of a cell for 10 seconds. It will then move to the second cell and take the reading for 10 seconds. It will then wait the remainder of the time specified in the Time interval (40 of the 60 seconds in this example), before returning to the first cell to begin the process again.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

- Deselect Alignment mode.

See also

- Graphs

Manual Control – Data Collection

What settings are available on the Data Collection page?

Response (s)	Select the Response time.
UV lamp on	Select whether the UV lamp is on or off.
Visible lamp on	Select whether the Visible lamp is on or off.
Lamp change at (nm)	Enter the wavelength at which the lamps are to be switched.

Wavelength (nm)	Enter the required wavelengths.
Ordinate mode	Select the Ordinate mode from the drop-down list.
Response (s)	Enter the response time (in seconds).
Slit width (nm)	Select the Slit width from the drop-down list.
Alignment mode	Select whether Alignment mode is on or off.
UV lamp on	Select whether the UV lamp is on or off.
Visible lamp on	Select whether the Visible lamp is on or off.
Lamp change at (nm)	Enter the wavelength at which the lamps are to be switched.

Ordinate mode	Select the Ordinate mode from the drop-down list.
Slit width	Select the Slit width from the drop-down list.
UV lamp on	Select whether the UV lamp is on or off.
Visible lamp on	Select whether the Visible lamp is on or off.
Lamp change at (nm)	Enter the wavelength at which the lamps are to be switched.

How do I go to a particular wavelength?

- Enter the **Wavelength** to go to and click **Apply**.

The instrument goes to the selected wavelength and the Instrument Status Bar updates to show that the instrument is at the selected wavelength.

How do I select the Ordinate mode?

- Select **A** (absorbance), **%T** (transmittance), **E1** (energy from sample beam), **E2** (energy from reference beam), or **%R** (reflectance) from the drop-down list.

How do I select the Slit width?

- Select **0.5, 1, 2, or 4 nm** from the drop-down list.

NOTE: The slit width is fixed on a Lambda 20 and Lambda 25. The drop-down list is grayed.

What Slit width should I use?

The slit width for the UV/Visible range is in nm.

- Set a slit width that is between one fifth and one tenth of the bandwidth of the spectral feature of interest.
- Set a wide slit to increase the energy throughput and the signal-to-noise ratio.
This decreases the resolution and the photometric accuracy and can cause band broadening.
- Set a narrower slit to increase the resolution and the photometric accuracy.
This decreases the signal-to-noise ratio.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

- Deselect Alignment mode.

NOTE: When **Alignment mode** is selected, **Wavelength** is disabled.

How do I perform an Autozero?

1. Enter the Wavelength at which you want to perform the Autozero and then click **Apply**.
The instrument goes to the selected wavelength.
2. From the Actions menu select **Autozero**.
The following message is displayed – 'Remove sample(s) and then press OK to perform a 100%T / OA.
Autozero sets the ordinate to read zero absorbance.

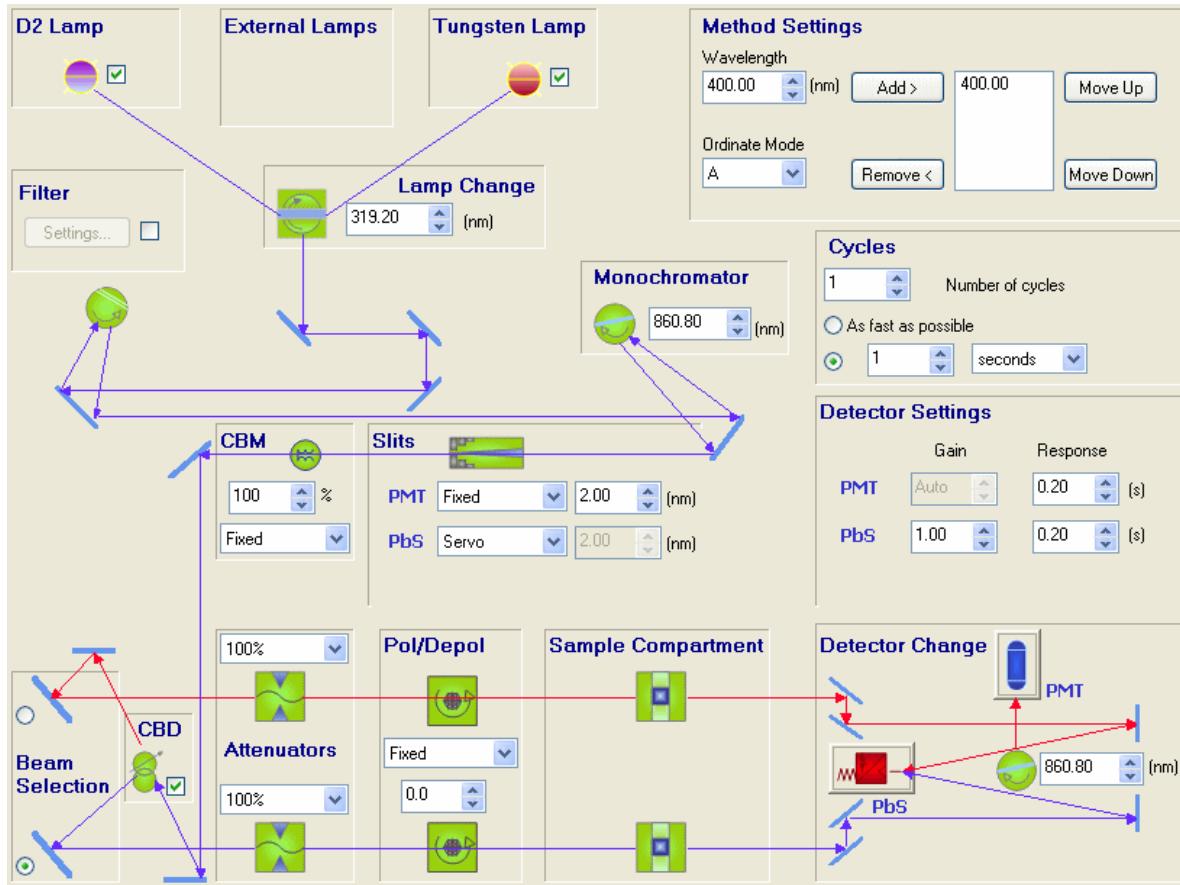
See also

- Manual Control
- Autosampler manual control
- Sipper manual control
- Cell changer manual control
- Peltier manual control.

High Performance Instruments – Data Collection

The picture below shows the Data Collection page for High performance instruments. Only the Method Settings vary between Methods, and depend on the type of task you are creating. Please see Scan Task Data Collection, Timedrive Task Data Collection, Wavelength Task Data Collection, Polarization Task Data Collection or Manual Control Data Collection for information about the Method Settings for the selected Method.

NOTE: Scanning Quant and Wavelength Quant are derivatives of Scan and Wavelength Program, respectively, and so are not covered separately here.



NOTE: If you enter a value outside the defined limits in a field, the field turns red until an acceptable value is entered. You are still able to access other nodes of the Folder List leaving invalid values on this page.

NOTE: The settings on the Data Collection page MUST NOT be altered when the Sample Table is being run.

D2 Lamp



The **D2 Lamp** selection is available for all High performance instruments. The check box allows the lamp to be selected or deselected for methods, allowing methods to be run without the lamp being switched on.

- Select the check box to enable the D2 Lamp.

NOTE: If you wish the lamps to be turned off when the instrument is not in use, this can be set within Manual Control.

If the **D2Lamp** is turned off (deselected), the **Lamp Change** is automatically grayed.

External Lamps



The External Lamp selection is available for all High performance instruments. The check box allows the lamp to be selected or deselected for methods, allowing methods to be run without the External Lamp switched on.

NOTE: If you wish the lamps to be turned off when the instrument is not in use, this can be set within Manual Control by deselecting the checkbox.

1. Select the check box to enable the External Lamp.
2. Select the type of lamp from the drop-down list.

NOTE: The drop-down list is empty unless a lamp has been added in the Instrument properties dialog.

Tungsten Lamp



The check box allows the lamp to be selected or deselected for methods, allowing methods to be run without the lamp switched on.

NOTE: If you wish the lamp to be turned off when the instrument is not in use, this can be set within Manual Control.

- Select the check box to enable the Tungsten Lamp.

If the **Tungsten lamp** is turned off (deselected), the **Lamp Change** is automatically grayed.

Filter



This Filter setting only appears if you have Service permission. Selecting Filter and clicking settings displays the Filter Change Settings dialog.

The Filter Change Settings dialog displays the current filter table settings that can be modified, and is mainly for use by PerkinElmer Service Engineers. You should not alter anything unless you fully understand the implications of doing so.

Filters are held on a disc within the instrument, which moves to the correct position when the appropriate wavelength is reached. The disc can rotate in one direction only. The table below shows how the filter is related to the position (defaults). Positions 8, 9, and 10 are NIR filters, and are only available on the Lambda 750, 900, 950 and 1050 spectrometers.

Position	Wavelength (nm)	Filter
1	150	Glass Filter
2	319.2	T=100%
3	379.2	UG11
4	562.4	BG38
5	690.4	OG550
6	810.4	RG665
7	1190.4	RG780
8	1670.4	T-LPG-1.0
9	2620.8	T-LPG-1.5
10	3350	T-LPG-2.5

A maximum of 20 rows can be specified in the Filter table.

1. To add a row, click **Add**.

A row is added to the table.

2. Click in the **Wavelength** field, and enter the required wavelength.

It is not possible to enter the same wavelength twice.

3. Click in the **Position** field and select the position from the drop-down list. The **Filter Type** is automatically updated.

OR

Click in the **Filter Type** field and select the filter from the drop-down list. The **Position** field is automatically updated.

The same filter position and filter type can be specified in the table more than once.

NOTE: The wavelengths can be added in any order. If you wish to re-order the wavelengths, you can click on the wavelength column header. When the dialog is closed the software will automatically re-order the table into wavelength order if they are not already.

- To delete a row, click anywhere in the row and click **Remove**.
- To return the table to the default values, click **Defaults**.

Lamp Change



This is used to set the point at which beams are switched between deuterium and tungsten lamps.

- Enter or select the lamp change point.

The default wavelength is 319.2 nm.

NOTE: If you press or the wavelength increases or decreases by 0.1 nm respectively. If you keep the button depressed, the wavelength increases or decreases by 10 nm respectively.

NOTE: We recommend that you do not set the value outside 300–350 nm as this may damage the instrument.

Monochromator



NOTE: This is only applicable for Lambda 750, 900, 950 and 1050.

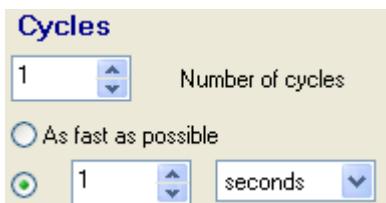
- Enter the monochromator change point.

The default is 860.8 nm. The minimum is 175 and the maximum is 900 nm. Values below 700 nm or greater than 900 nm may damage your instrument. Please refer to the manual provided with the instrument.

NOTE: If you press or the value increases or decreases by 0.1, respectively. If you keep the button depressed, the value increases or decreases by 10, respectively.

NOTE: When the monochromator change point is modified, the detector change point is updated to the same value automatically.

Cycles



Cycles are available for Wavelength Program, Polarization and Scan methods.

Cycles enable you to perform multiple scans per sample. It is possible to select the **Number of cycles**, and then select either **As fast as possible**, or enter a cycle time.

1. Enter or select the **Number of cycles**.

The default is 1. The minimum is 1 and the maximum is 999. As the results are generated, each result includes .cycleX at the end of the name, where X is the number of the cycle.

NOTE: If you press or the number increases or decreases by 1, respectively. If you keep the button depressed, the number increases or decreases by 10, respectively.

2. Select **As fast as possible** or a cycle time.

If As fast as possible is selected, the cycle time will be shown in brackets. As soon as a cycle finishes the next one begins.

3. If a cycle time is selected, enter the time and select the time units (**seconds**, **minutes** or **hours**) from the drop-down list. The time you enter must be more than the minimum, and, in effect, you are introducing a delay before the next cycle begins.

The limits are 1–999 seconds, 0.01–999.00 minutes and hours.

Common Beam Mask (CBM)



The Common Beam Mask applies a mask in the common beam to reduce the beam height. 100% means that the mask is fully open and 100% of the beam passes through. 0% means that the mask is closed and no light passes through.

The CBM is not available on the Data Collection page when a Universal Reflectance Accessory (URA) is installed, as the value is defined on the Universal Reflectance Accessory (URA) page. This applies when Absolute Reflectance Mode or Relative Reflectance Mode is selected. If Transmission (Sample Compartment) Mode is selected on the accessory page, the CBM is available on the Data Collection page, but the allowable values are limited due to the optics of the URA.

1. Select the source of the Common Beam Mask data.

Fixed will use the value shown above this drop-down list.

If you select Sample Table, the value will be defined in the Sample Table.

Selecting Sample Table means that different values can be specified for each sample if required.

2. Enter or select the mask percentage.

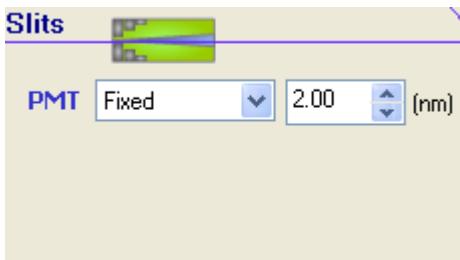
The default wavelength is 100%.

NOTE: If you press or the percentage increases or decreases by 1%, respectively. If you keep the button depressed, the percentage increases or decreases by 10%, respectively.

Slits

Lambda 650, 800 and 850 spectrometers

Lambda 750, 900 and 950 spectrometers



Lambda 1050 WB and 1050 NB spectrometers and Three Detector Module



NOTE: The Lambda 1050 has the Three Detector Module installed as standard. The Three Detector Module check box must be selected on the Accessory page for the Three Detector Module settings to appear.

NOTE: The PbS slit is only available for the Lambda 750, 900, 950 and 1050 spectrometers. The InGaAs slit is only available for the Lambda 1050 or a High performance instrument with the Three Detector Module installed.

NOTE: Slits are not available on the Data Collection page when a Universal Reflectance Accessory (URA) is installed as the slit width is defined on the Universal Reflectance Accessory (URA) page.

Select the slit mode – **Fixed**, **Servo**, **Programmed** or **Sample Table**.

- If **Fixed** is selected, enter the slit width in the adjacent field. For the PMT (UV/Vis) slit, the default is 2.00. The minimum is 0.05 and the maximum is 5.00. Click or to increment the slit width by 0.05 nm. If you keep the button depressed, the width increases or decreases by 0.5 nm.
Set a slit width that is between one-fifth and one-tenth of the bandwidth of the spectral feature of interest.
Set a wide slit width to increase the energy throughput and the signal-to-noise ratio. This decreases the resolution and the photometric accuracy and can cause band broadening.
Set a narrower slit width to increase the resolution and the photometric accuracy. This decreases the signal-to-noise ratio.
For the PbS and InGaAs (NIR) slits the default is 2.00. The minimum is 0.2 and the maximum is 20.0.
Click or and keep the button depressed to increase or decrease the width by 1 nm, respectively.
- If **Programmed** is selected, the slit width field is grayed, and values must be defined on the Program page.
- If **Servo** is selected, the slit width field is grayed, and the system will monitor the reference beam energy and adjust the slits to avoid over saturation of the detectors. Servo is only available for the PbS and InGaAs slits.
- If you select **Sample Table**, the value will be defined in the Sample Table. Selecting Sample Table means that different NIR nodes and slit widths can be specified for each sample if required.

Detector Settings

Lambda 650, 650R, 650S, 800 and 850 spectrometers

Detector Settings		
	Gain	Response
PMT	40	5.00 (s)

Lambda 750, 900 and 950 spectrometers

Detector Settings		
	Gain	Response
PMT	Auto	0.20 (s)
PbS	1.00	0.20 (s)

Lambda 1050 WB and 1050 NB spectrometers and Three Detector Module

Detector Settings		
	Gain	Response
PMT	Auto	0.20 (s)
InGaAs	0.00	0.20 (s)
PbS	0.00	0.20 (s)

NOTE: The Lambda 1050 has the Three Detector Module installed as standard. The Three Detector Module check box must be selected on the Accessory page for the settings for the Three Detector Module to appear.

Photomultiplier Tube (PMT) Gain Setting

Defines the photomultiplier tube (PMT) gain factor for the UV/Vis range. It is used for energy measurement in single beam mode. A gain must be set on the PMT to produce a useable spectrum. If the gain is too low, no spectrum will be observed. If the gain is too high, the detector will saturate and the spectrum will be truncated. Setting the gain is a matter of 'trial and error' as the correct gain will depend on other factors such as slit width.

- Enter or select the required gain.
The available options are Auto or 0–255.
If you press or the gain increases or decreases by 1 respectively. If you keep the button depressed, the gain increases or decreases by 10 respectively.

Photomultiplier Tube (PMT) Response Setting

Defines the signal average time. One chopper cycle is 0.04 sec. This is the minimal time for the measurement of one transmission value.

- Enter or select the required response (0.04–10.00).

If **Programmed** is selected for the slits, the PMT Response Setting is grayed. The value must instead be entered on the Program page.

InGaAs Gain Setting

Defines the gain factor for the InGaAs detector. It is used for energy measurement in single beam mode (E1 or E2 Ordinate Mode). A gain must be set on the InGaAs detector to produce a useable spectrum. If the gain is too low, no spectrum will be observed. If the gain is too high, the detector will saturate and the spectrum will be truncated. Setting the gain is a matter of 'trial and error' as the correct gain will depend on other factors such as slit width.

- Enter or select the required gain (0–20).

If you press  or  the gain increases or decreases by 0.1, respectively. If you keep the button depressed, the gain increases or decreases by 1, respectively.

If **Programmed** is selected for the slits, the InGaAs Gain Setting is grayed. The value must instead be entered on the Program page.

InGaAs Response Setting

Defines the InGaAs response. One chopper cycle is 0.04 sec. This is the minimal time for the measurement of one transmission value.

- Enter or select the required response (0.04–10.00).

Increasing the response setting decreases the scan speed.

If **Programmed** is selected for the slits, the InGaAs Response Setting is grayed. The value must instead be entered on the Program page.

PbS Gain Setting

Defines the gain factor for the lead sulfide detector. It is used for energy measurement in single beam mode (E1 or E2 Ordinate Mode). A gain must be set on the PbS detector to produce a useable spectrum. If the gain is too low, no spectrum will be observed. If the gain is too high, the detector will saturate and the spectrum will be truncated. Setting the gain is a matter of 'trial and error' as the correct gain will depend on other factors such as slit width.

- Enter or select the required gain (0–10).

If you press  or  the gain increases or decreases by 0.1, respectively. If you keep the button depressed, the gain increases or decreases by 1, respectively.

If **Programmed** is selected for the slits, the PbS Gain Setting is grayed. The value must instead be entered on the Program page.

PbS Response Setting

Defines the PbS response. One chopper cycle is 0.04 sec. This is the minimal time for the measurement of one transmission value.

- Enter or select the required response (0.04–10.00).

Increasing the response setting decreases the scan speed.

If **Programmed** is selected for the slits, the PbS Response Setting is grayed. The value must instead be entered on the Program page.

Beam Selection



The sample beam is purple. The reference beam is red.

NOTE: When using the Universal Reflectance Accessory (URA), the front beam must be the sample beam.

- Select the front or rear beam as the sample beam.
The front beam is selected by default.

Common Beam Depolarizer (CBD)

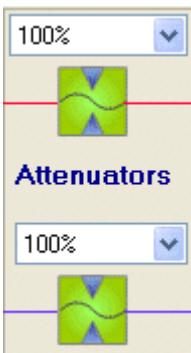


The Common Beam Depolarizer is an optional accessory. It is used to depolarize the radiation that comes from the monochromator. The Common Beam Depolarizer is mounted in the common beam within the spectrometer and does not affect the available space within the sample compartment. This is only available if the Common Beam Depolarizer is enabled from the Instrument Properties.

If you wish to perform Polarization scans, we recommend that you have the Common Beam Depolarizer and Depolarizer/Polarizer installed. If using a Universal Reflectance Accessory, we recommend that you have the Common Beam Depolarizer installed. If it is not installed, you may observe polarization effects.

- Select the check box to enable the Common Beam Depolarizer.

Rear and Front Beam Attenuators



The Front and Rear beam attenuators are used to select the attenuation in the sample and reference beam. Reference beam attenuation is used to improve noise levels at high absorbance/low transmittance. The instrument uses a single detector (photomultiplier for UV/Vis and PbS or InGaAs for NIR). A ratio is measured between the sample and reference beam. With highly absorbing samples this means there is a ratio between a very small signal (highly absorbing sample) and a big signal (the unblocked reference beam). The instrument cycles between the two readings every 40 milliseconds and the rapidly changing light levels can cause the detector to become noisy. In addition, there is the mathematical problem of performing a ratio with a large difference between the numerator and the denominator.

Adding a 10% (1 A), 1% (2 A) or 0.1% (3 A, Lambda 1050 only) attenuator in the reference beam reduces the reference signal and improves the situations described above and so a better spectrum results.

Select **0%, 0.1% (Lambda 1050 only), 1%, 10%, 100%, Automatic** or **Sample Table** from the drop-down lists for the front and rear beam attenuators.

NOTE: **Automatic** is only available for the sample beam.

0%, 0.1% (Lambda 1050 only), 1%, 10%, 100% correspond to the amount of energy passing through the attenuator. 100% means that the beam is open. 0% means that the beam is closed.

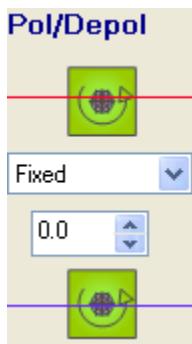
If you select **Sample Table**, the values (**0%, 0.1%, 1%, 10%, 100%, Automatic**) can be defined on a per sample basis in the Sample Table. Also, if **Programmed** is chosen as the slit mode, the attenuator drop-down lists will be grayed out and the attenuator can be defined on the Program page.

For samples that have an absorbance >3 A, you should consider using the attenuators. For samples that have an absorbance >4 A, we recommend that you use the attenuators.

When the sample beam attenuator is set to automatic and you click , the software will collect three spectra:

1. 100%T baseline – Sample beam and reference beam set to 100%
2. 0%T baseline – Sample beam 0% and reference beam 100%
3. Attenuator spectrum – Sample beam set to the value defined for the reference beam attenuator on the Data Collection page (for example 1% or 10%) and reference beam set to 100%.

Depolarizer/Polarizer



Along with frequency and amplitude, polarization is a key parameter that is used to describe the character of radiation. Radiation is said to be polarized when there is a preference direction of oscillation of the electromagnetic waves. Natural radiation (for example, the sun) consists of electromagnetic waves, with the oscillation of the electric and magnetic vectors being distributed in all planes perpendicular to the direction of propagation. When natural radiation interacts with optical elements, it becomes polarized to a certain extent. Therefore, the radiation within a spectrometer will inevitably be polarized.

The optional depolarizer/polarizer mounts within the sample compartment. It is used to polarize or depolarize the beam of light entering the sample. A switch on the accessory sets it to depolarizer or polarizer.

It can be configured with separate depolarizers or polarizers for the sample and reference beams. The accessory is fully automated and is recognised by the UV WinLab software.

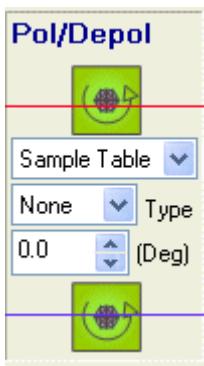
For certain types of samples, the spectroscopic results are dependent on the orientation of the sample to the existing plane of polarization of the sample beam. Placing depolarizer optics into the beam can eliminate this influence. The depolarizer unit comprises two quartz wedges: one of natural quartz and the other of Suprasil quartz. A depolarizer is recommended for all samples exhibiting anisotropic behavior of absorbance, and for all types of transmission and reflectance measurements involving a non near angle of incidence for the sample. The applicable wavelength range is 190 to 2600 nm. Since the radiation is depolarized directly in front of the sample, the depolarizing efficiency is better than 98%.

NOTE: The picture above shows the sample beam at the front. The actual position (front or rear) depends on the beam selection (see beam selection above). The Depolarizer/Polarizer is not shown on the Data Collection page if it is not installed in the instrument.

1. Select the Source (**Fixed** or **Sample Table**) from the drop-down list.
If Sample Table is selected, the value below becomes grayed and the values must be entered in the Sample Table.
2. If **Fixed** is selected, enter the angle in the field below the drop-down list.
The minimum is 0° and the maximum is 340°.

NOTE: The Depolarizer/Polarizer is grayed for a Polarization method. The settings are defined in the Method Settings area at the top of the Data Collection page.

When a Universal Reflectance Accessory (URA) is installed, an extra drop-down list is available:



The **Type** drop-down list is used to alter the effective pathlength to allow for the polarizer/depolarizer installed.

- Select the type from the drop-down list – **None, 1 mm, 2mm, 5 mm, 10 mm, 15 mm, 20 mm, or 30 mm**.

A Polarizer type column is added to the sample table. The value selected on the instrument page is used in the sample table by default but it can be altered for each sample or measurement.

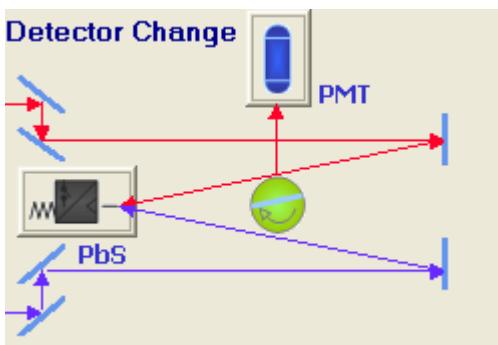
Sample Compartment

Sample Compartment No settings are required here.

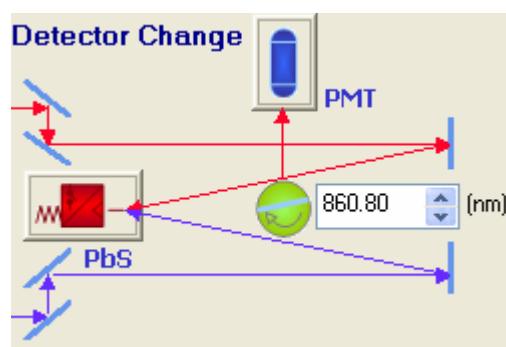


Detector Change

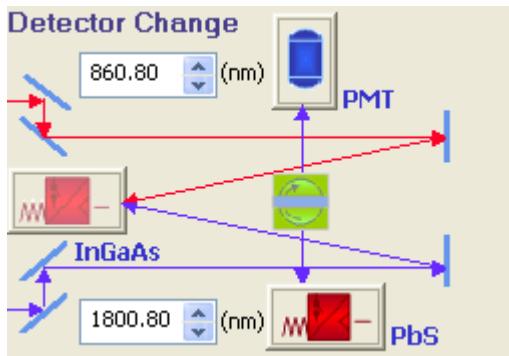
Lambda 650, 650R, 650S, 800 and 850 spectrometers



Lambda 750, 900 and 950 spectrometers



Lambda 1050 WB and 1050 NB spectrometers and Three Detector Module



NOTE: The Lambda 1050 spectrometer has the Three Detector Module installed as standard. The Three Detector Module check box must be selected on the Accessory page for the settings for the Three Detector Module settings to appear.

Defines the Detector Change point(s) (nm).

- For a Lambda 750, 900, 950 or 1050, enter or select the detector change point(s).
For the PMT/PbS change point the default is 860.80 nm.
For the PMT/InGaAs change point (Lambda 1050 and Three Detector Module only) the default is 860.80 nm.
For the InGaAs/PbS change point (Lambda 1050 and Three Detector Module only) the default is 1800.80 nm.
If you press or the value increases or decreases by 0.1 respectively. If you keep the button depressed, the value increases or decreases by 10 respectively.

NOTE: When the monochromator change point is modified, the detector change point is updated to the same value automatically. However, the detector change point can then be modified independently.

NOTE: This detector compartment changes if a Universal Reflectance Accessory (URA) or a Three Detector Module is installed. See What Detector Settings are displayed if a Universal Reflectance Accessory (URA) is installed? and What Detector Settings are displayed if a Three Detector Module is installed? below.

What elements of the user interface are available for the different method types?

Below is a quick look-up table that shows the different parts of the user interface that are available for the different methods (for example, Start polarization angle is only available for Polarization methods).

NOTE: Scanning Quant and Wavelength Quant are derivatives of Scan and Wavelength Program respectively and so are not listed separately in the tables below.

The area number refers to the area as numbered below on a map of the Data Collection page.

Lambda 650/650S/650R

Control	Wavelength Programming	Scan	Timedrive	Polarization Scan	Manual Control
D2 Lamp	<input type="checkbox"/>				
External Lamp	dependant on accessory				
Tungsten Lamp	<input type="checkbox"/>				
Wavelength	<input type="checkbox"/>				
Start Wavelength	<input type="checkbox"/>				
End Wavelength	<input type="checkbox"/>				
Wavelength Program	<input type="checkbox"/>				
Add	<input type="checkbox"/>				
Remove	<input type="checkbox"/>				
Move Up	<input type="checkbox"/>				
Move Down	<input type="checkbox"/>				
Data Interval	<input type="checkbox"/>				
Total Time	<input type="checkbox"/>				
Start Pol angle	<input type="checkbox"/>				
End Pol angle	<input type="checkbox"/>				
Ordinate Mode	<input type="checkbox"/>				
Number of cycles	<input type="checkbox"/>				
As fast as possible	<input type="checkbox"/>				
Defined interval	<input type="checkbox"/>				
Lamp Change	<input type="checkbox"/>				

Common Beam Mask	<input type="checkbox"/>				
Monochromator	<input type="checkbox"/>				
Photomultiplier Gain	<input type="checkbox"/>				
Photomultiplier Response	<input type="checkbox"/>				
NIR Gain	<input type="checkbox"/>				
NIR Response	<input type="checkbox"/>				
Front Beam	<input type="checkbox"/>				
Rear Beam	<input type="checkbox"/>				
Common Beam Depolarizer	dependant on accessory				
Front Beam Attenuator	<input type="checkbox"/>				
Rear Beam Attenuator	<input type="checkbox"/>				
Double Polarizer / Depolarizer	dependant on accessory				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Sample compartment	dependant on accessory				
Ordinate mode	<input type="checkbox"/>				
Scan speed (read only)	<input type="checkbox"/>				

Lambda 800/850

Control	Wavelength Programming	Scan	Timedrive	Polarization Scan	Manual Control
D2 Lamp	<input type="checkbox"/>				
External Lamp	dependant on accessory				
Tungsten Lamp	<input type="checkbox"/>				
Wavelength	<input type="checkbox"/>				
Start Wavelength	<input type="checkbox"/>				
End Wavelength	<input type="checkbox"/>				
Wavelength Program	<input type="checkbox"/>				
Add	<input type="checkbox"/>				
Remove	<input type="checkbox"/>				
Up	<input type="checkbox"/>				
Down	<input type="checkbox"/>				
Data Interval	<input type="checkbox"/>				
Total Time	<input type="checkbox"/>				
Start Pol angle	<input type="checkbox"/>				
End Pol angle	<input type="checkbox"/>				
Ordinate Mode	<input type="checkbox"/>				
Number of cycles	<input type="checkbox"/>				
As fast as possible	<input type="checkbox"/>				
Defined interval	<input type="checkbox"/>				
Lamp Change	<input type="checkbox"/>				

Common Beam Mask	<input type="checkbox"/>				
Monochromator	<input type="checkbox"/>				
Photomultiplier Gain	<input type="checkbox"/>				
Photomultiplier Response	<input type="checkbox"/>				
NIR Gain	<input type="checkbox"/>				
NIR Response	<input type="checkbox"/>				
Front Beam	<input type="checkbox"/>				
Rear Beam	<input type="checkbox"/>				
Common Beam Depolarizer	dependant on accessory				
Front Beam Attenuator	<input type="checkbox"/>				
Rear Beam Attenuator	<input type="checkbox"/>				
Double Polarizer / Depolarizer	dependant on accessory				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Sample compartment	dependant on accessory				
Detector change point	<input type="checkbox"/>				
Scan speed (read only)	<input type="checkbox"/>				

Lambda 750/900/950

Control	Wavelength Programming	Scan	Timedrive	Polarization Scan	Manual Control
D2 Lamp	<input type="checkbox"/>				
External Lamp	dependant on accessory				
Tungsten Lamp	<input type="checkbox"/>				
Wavelength	<input type="checkbox"/>				
Start Wavelength	<input type="checkbox"/>				
End Wavelength	<input type="checkbox"/>				
Wavelength Program	<input type="checkbox"/>				
Add	<input type="checkbox"/>				
Remove	<input type="checkbox"/>				
Up	<input type="checkbox"/>				
Down	<input type="checkbox"/>				
Data Interval	<input type="checkbox"/>				
Total Time	<input type="checkbox"/>				
Start Pol angle	<input type="checkbox"/>				
End Pol angle	<input type="checkbox"/>				
Ordinate Mode	<input type="checkbox"/>				
Number of cycles	<input type="checkbox"/>				
As fast as possible	<input type="checkbox"/>				
Defined interval	<input type="checkbox"/>				
Lamp Change	<input type="checkbox"/>				

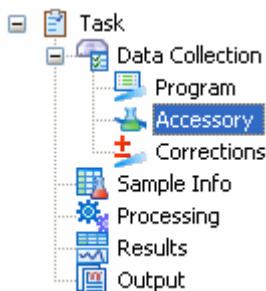
Common Beam Mask	<input type="checkbox"/>				
Monochromator	<input type="checkbox"/>				
Photomultiplier Gain	<input type="checkbox"/>				
Photomultiplier Response	<input type="checkbox"/>				
NIR Gain	<input type="checkbox"/>				
NIR Response	<input type="checkbox"/>				
Front Beam	<input type="checkbox"/>				
Rear Beam	<input type="checkbox"/>				
Common Beam Depolarizer	dependant on accessory				
Front Beam Attenuator	<input type="checkbox"/>				
Rear Beam Attenuator	<input type="checkbox"/>				
Double Polarizer / Depolarizer	dependant on accessory				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Sample compartment	dependant on accessory				
Detector change point	<input type="checkbox"/>				
Scan speed (read only)	<input type="checkbox"/>				

Lambda 1050/Three Detector Module

Control	Wavelength Programming	Scan	Timedrive	Polarization Scan	Manual Control
D2 Lamp	<input type="checkbox"/>				
External Lamp	dependant on accessory				
Tungsten Lamp	<input type="checkbox"/>				
Wavelength	<input type="checkbox"/>				
Start Wavelength	<input type="checkbox"/>				
End Wavelength	<input type="checkbox"/>				
Wavelength Program	<input type="checkbox"/>				
Add	<input type="checkbox"/>				
Remove	<input type="checkbox"/>				
Up	<input type="checkbox"/>				
Down	<input type="checkbox"/>				
Data Interval	<input type="checkbox"/>				
Total Time	<input type="checkbox"/>				
Start Pol angle	<input type="checkbox"/>				
End Pol angle	<input type="checkbox"/>				
Ordinate Mode	<input type="checkbox"/>				
Number of cycles	<input type="checkbox"/>				
As fast as possible	<input type="checkbox"/>				
Defined interval	<input type="checkbox"/>				
Lamp Change	<input type="checkbox"/>				
Common Beam Mask	<input type="checkbox"/>				

Monochromator	<input type="checkbox"/>				
Photomultiplier Gain	<input type="checkbox"/>				
Photomultiplier Response	<input type="checkbox"/>				
InGaAs Gain	<input type="checkbox"/>				
InGaAs Response	<input type="checkbox"/>				
PbS Gain	<input type="checkbox"/>				
PbS Response	<input type="checkbox"/>				
Front Beam	<input type="checkbox"/>				
Rear Beam	<input type="checkbox"/>				
Common Beam Depolarizer	dependant on accessory				
Front Beam Attenuator	<input type="checkbox"/>				
Rear Beam Attenuator	<input type="checkbox"/>				
Double Polarizer / Depolarizer	dependant on accessory				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Slit mode	<input type="checkbox"/>				
Slit width	<input type="checkbox"/>				
Sample compartment	dependant on accessory				
Detector change point(s)	<input type="checkbox"/>				
Scan speed (read only)	<input type="checkbox"/>				

Accessories



What Detector Settings are displayed if a Universal Reflectance Accessory (URA) is installed?



The detector settings available for the URA depend upon the ordinate mode selected in the Method Settings Section of the Data Collection page.

Silicon (Si) Gain Setting

Defines the Silicon (Si) detector gain factor for the UV/Vis range. It is used for energy measurement in single beam mode (E1 or E2 Ordinate Mode). A gain must be set on the Si detector to produce a useable spectrum. If the gain is too low, no spectrum will be observed. If the gain is too high, the detector will saturate and the spectrum will be truncated. Setting the gain is a matter of 'trial and error' as the correct gain will depend on other factors such as slit width.

- Enter or select the required gain.
If you press **+** or **-** the gain increases or decreases by 1 respectively. If you keep the button depressed, the gain increases or decreases by 10 respectively.

Silicon (Si) Detector Response Setting

Defines the signal average time. One chopper cycle is 0.04 sec. This is the minimal time for the measurement of one transmission value.

- Enter or select the required response (0.04–10.00).

If **Programmed** is selected for the slits, the Si Response Setting is grayed. The value must instead be entered on the Program page.

PbS Gain Setting

Defines the gain factor for the lead sulfide detector. It is used for energy measurement in single beam mode (E1 or E2 Ordinate Mode). A gain must be set on the PbS detector to produce a useable spectrum. If the gain is too low, no spectrum will be observed. If the gain is too high, the detector will saturate and the spectrum will be truncated. Setting the gain is a matter of 'trial and error' as the correct gain will depend on other factors such as slit width.

- Enter or select the required gain (0–9).

If you press or the gain increases or decreases by 0.1 respectively. If you keep the button depressed, the gain increases or decreases by 1 respectively.

If **Programmed** is selected for the slits, the PbS Gain Setting is grayed. The value must instead be entered on the Program page.

PbS Response Setting

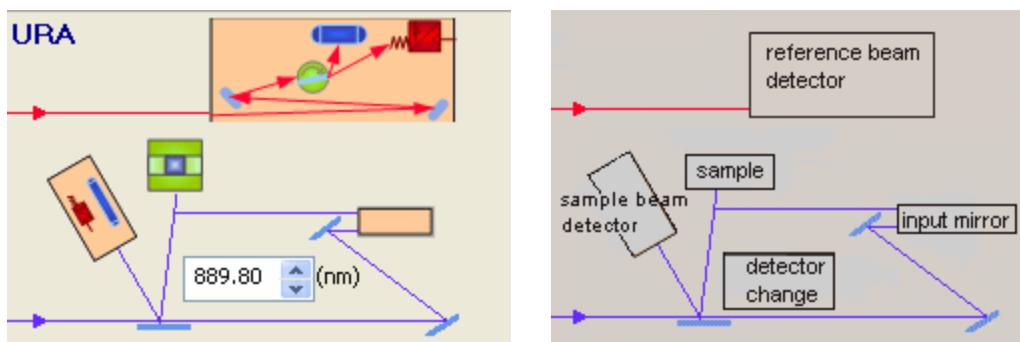
Defines the PbS response. One chopper cycle is 0.04 sec. This is the minimal time for the measurement of one transmission value.

- Enter or select the required response (0.04–10.00).

Increasing the response setting decreases the scan speed.

If **Programmed** is selected for the slits, the PbS Response Setting is grayed. The value must instead be entered on the Program page.

What Detector Change settings are displayed if a Universal Reflectance Accessory (URA) is installed?



1. For Lambda 750, 900, 950 or 1050 spectrometers, enter or select the Detector Change point.

You should only alter this value if you are fully aware of the implications of doing so.

NOTE: If you press or the value increases or decreases by 0.1 respectively. If you keep the button depressed, the value increases or decreases by 10 respectively.

What Detector Settings are displayed if a Three Detector Module is installed?

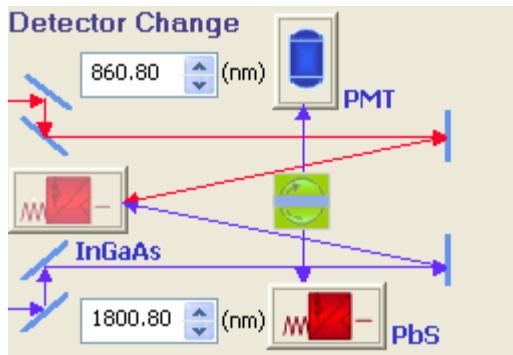
If a Three Detector Module is installed in the High performance spectrometer and selected on the Accessory page, the **Detector Settings** section will look like this:



See Detector Settings for more information.

What Detector Change settings are displayed if a Three Detector Module is installed?

If a Three Detector Module is installed in the High performance spectrometer and selected on the accessory page, the **Detector Change** settings will look like this:



Defines the Detector Change point(s) (nm)

- For a Lambda 750, 900, 950 or 1050 spectrometer, enter or select the Detector Change point(s).
 - For the PMT/PbS change point the default is 860.8 nm.
 - For the PMT/InGaAs change point the default is 860.8 nm.
 - For the InGaAs/PbS change point the default is 1800.8 nm.

NOTE: If you press or the value increases or decreases by 0.1 respectively. If you keep the button depressed, the value increases or decreases by 10 respectively.

What Detector Settings are displayed if a Sphere Accessory is installed?

PbS Sphere (NIR Spectrometer)

Detector Settings		
	Gain	Response
PMT	Auto	0.20 [s]
PbS	0.00	0.20 [s]

InGaAs Sphere (NIR Spectrometer)

Detector Settings		
	Gain	Response
PMT	Auto	0.20 [s]
InGaAs	1.00	0.20 [s]

The detector settings displayed will depend on the type of sphere (InGaAs or PbS) installed and the type of instrument (UV/Vis or UV/Vis/NIR).

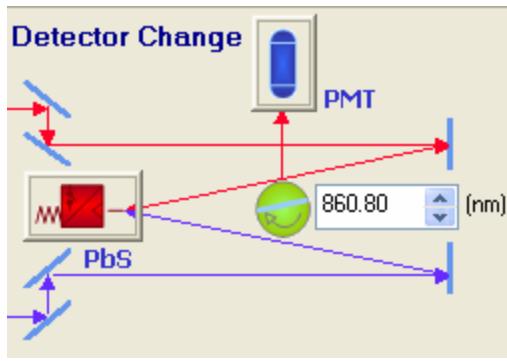
The spheres have two detectors. A PMT for the UV/Vis region and, depending on the variant of sphere accessory, an InGaAs or a PbS detector for the NIR region.

However, if a sphere is installed in a Lambda 650, 800 or 850 spectrometer, only the UV/Vis (PMT) detector will be available.

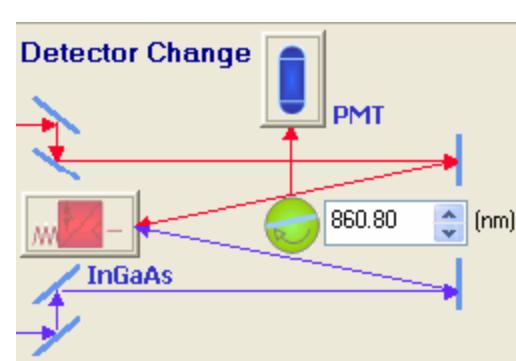
See Detector Settings for more information on the settings available for the different types of detector.

What Detector Change settings are displayed if a Sphere Accessory is installed?

PbS Sphere (NIR Spectrometer)



InGaAs Sphere (NIR Spectrometer)



The detector settings displayed will depend on the type of sphere installed (InGaAs or PbS) and the type of instrument (UV/Vis or UV/Vis/NIR).

The spheres have two detectors. A PMT for the UV/Vis region and, depending on the variant of sphere accessory, an InGaAs or a PbS detector for the NIR region.

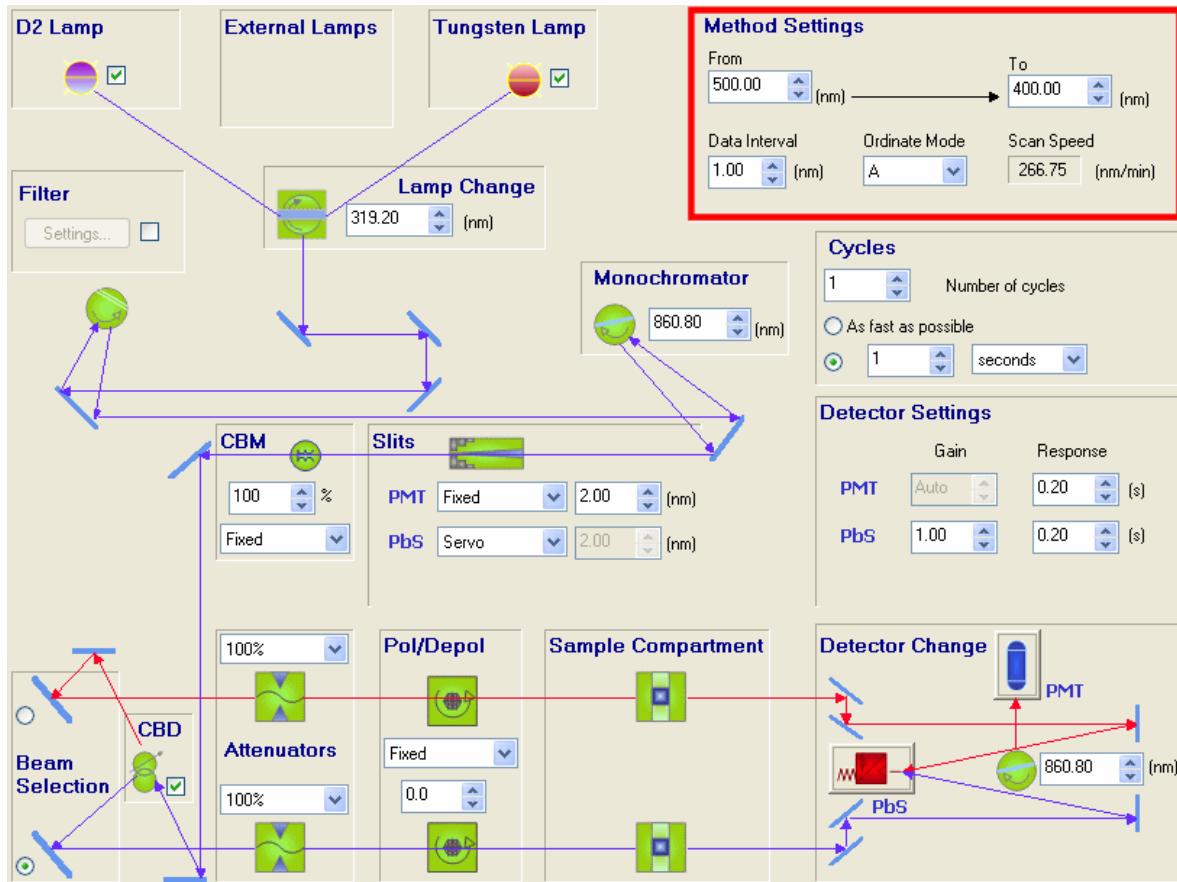
However, if a sphere is installed in a Lambda 650, 800 or 850 spectrometer, only the UV/Vis (PMT) detector will be available.

See also

- Scan Task Data Collection
- Wavelength Task Data Collection
- Polarization Task Data Collection
- Timedrive Task Data Collection
- Manual Control Data Collection.

Scan Task – Data Collection

The Data Collection page for the Scan Task can be used to set method settings such as the scan range and data interval. For information about the generic settings on the Data Collection page, see the Data Collection page for High performance instruments. Only the method settings that are unique to the Scan Task are discussed below.



NOTE: The scan speed is for reference only and cannot be edited.

How do I set the scan range?

1. Enter the upper wavelength in the **From** field.
2. Enter the lower wavelength in the **To** field.

NOTE: If a value outside the allowed range is entered, the field turns red until an acceptable value is entered. The **From** value must be higher than the **To** value.

What are the available wavelengths?

Lambda 650/650R/650S	190.0–900.0 nm
Lambda 750/750S	190.0–3300.0 nm
Lambda 800/850	175.0–900.0 nm
Lambda 900/950/1050	175.0–3300.0 nm

NOTE: It is possible to input a wavelength value down to -20 nm. These are to allow calibration only, rather than being real, experimental wavelengths.

How do I set the Data Interval?

- Enter the required value in the Data Interval field.

Lambda 650, 650R, 650S	0.01–10 nm
Lambda 800, 850	0.01–10 nm
Lambda 750, 900, 950, 1050	0.01–10 nm (UV/Vis) , 0.04–10 nm (NIR)

Pressing or increases / decreases the data interval by 0.01 nm.

How do I set the Ordinate Mode?

- Select the Ordinate Mode from the drop-down list.

The available options are A (absorbance), %T (transmittance), E1 (energy from sample beam), E2 (energy from reference beam), or %R (reflectance).

If E1 is selected as the Ordinate Mode, the sample beam is set as the front beam and the beam selection is disabled. If E2 is selected as the Ordinate Mode, the sample beam is set as the rear beam and the beam selection is disabled.

NOTE: Selecting E1 or E2 does not automatically set the slits to Fixed. However, the Slit mode and Ordinate Mode fields will turn red if E1 or E2 is selected with the slits not set to fixed.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

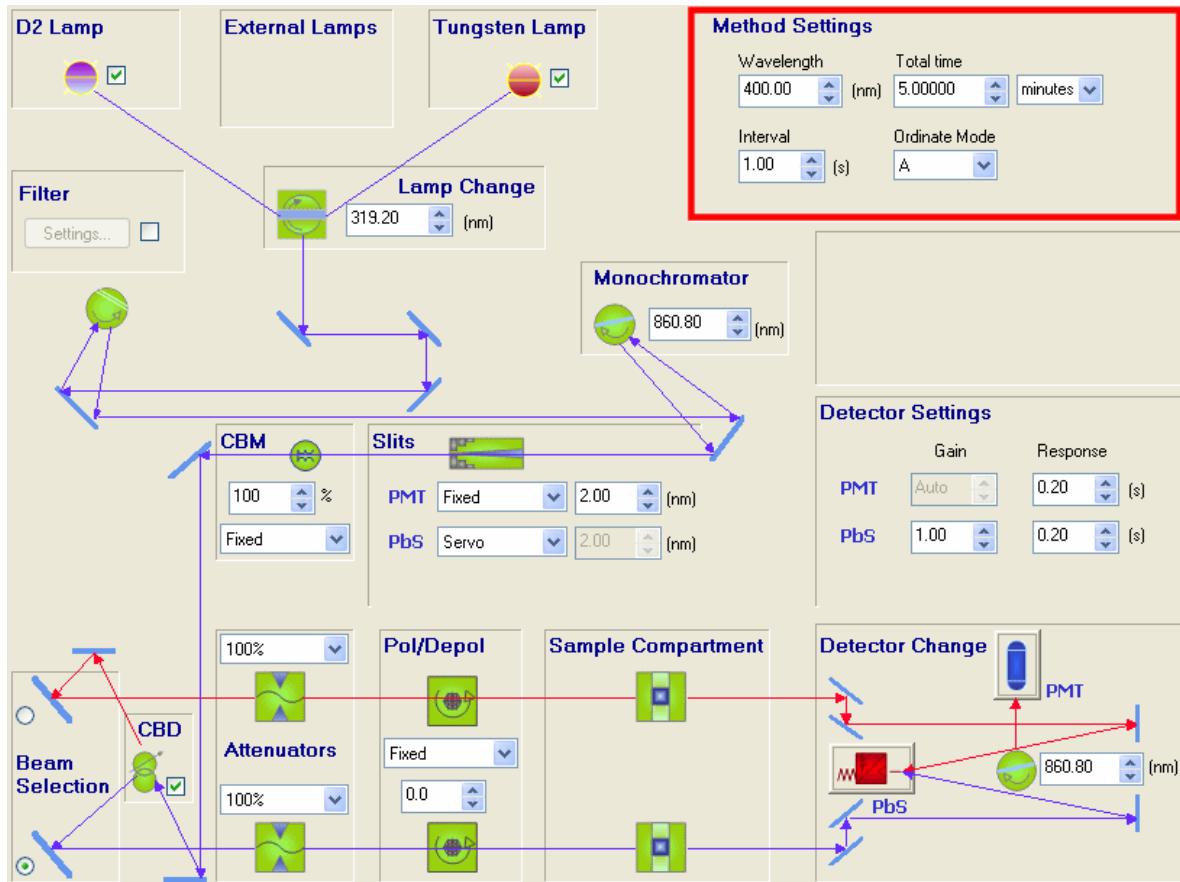
- Deselect Alignment mode.

See also

[Sample Info](#)

Timedrive Task – Data Collection

The Data Collection page for the Timedrive Task can be used to alter the method settings such as the wavelength and total time. For information about the generic settings on the Data Collection page, see the Data Collection page for High performance instruments. Only the method settings that are unique to the Timedrive Task are discussed below.



How do I define the Wavelength for the Timedrive method?

- Enter the required value in the **Wavelength** field.

NOTE: If a value outside the allowed range is entered, the field turns red until an acceptable value is entered.

What are the available wavelengths?

Lambda 650/650R/650S	190.0–900.0 nm
Lambda 750/750S	190.0–3300.0 nm
Lambda 800/850	175.0–900.0 nm
Lambda 900/950/1050	175.0–3300.0 nm

How do I define the Total time?

- Enter the required time and then select the units from the drop-down list.
The maximum allowed time is 360 000 seconds (equivalent to 6000 minutes, or 100 hours).

How do I set the Data Interval?

- Enter the required data **Interval**.
The value must be between 0.04 and 60 secs.

Pressing or increases / decreases the data interval by 0.04 secs.

NOTE: If the timedrive cannot be achieved due to sampling time constraints (such as constraints due to a cell changer being installed), the software will increase the data interval. The final data interval will not be reported until the timedrive is complete.

Can I extend the total time when I am collecting data?

Yes, the **Total time** can be extended while data is still being collected for a particular sample.

This new time will also be applied to samples that have not yet been run.

How do I set the Ordinate Mode?

- Select the Ordinate Mode from the drop-down list.
The available options are A (absorbance), %T (transmittance), E1 (energy from sample beam), E2 (energy from reference beam), or %R (reflectance).
If E1 is selected as the Ordinate Mode, the sample beam is set as the front beam and the beam selection is disabled. If E2 is selected as the Ordinate Mode, the sample beam is set as the rear beam and the beam selection is disabled.

NOTE: Selecting E1 or E2 does not automatically set the slits to Fixed. However, the Slit mode and Ordinate Mode fields will turn red if E1 or E2 is selected with the slits not set to fixed.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

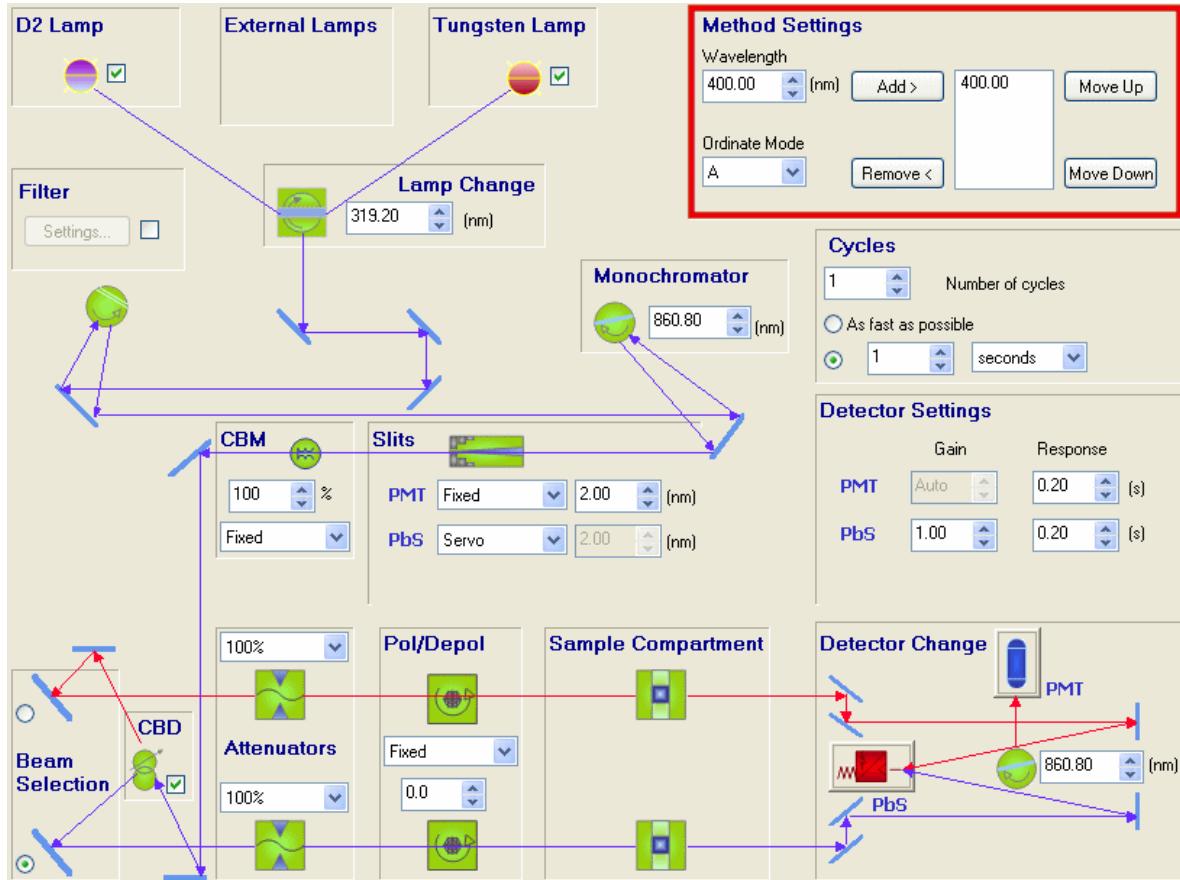
- Deselect Alignment mode.

See also

- Sample Info

Wavelength Task – Data Collection

The Data Collection page for the Wavelength Task can be used to alter the method settings such as the wavelength. For information about the generic settings on the Data Collection page, see the Data Collection page for High performance instruments. Only the method settings that are unique to the Wavelength Task are discussed below.



How do I add a Wavelength?

1. Enter the required value in the Wavelength field.

Use the **[** and **]** buttons to increase or decrease the value currently displayed. If you use a single click on a button, the value increases/decreases by 1. If you press and hold the button for more than three seconds, the wavelength increases/decreases by 10.

2. Click **Add**.

The wavelength is added to the bottom of the list.

NOTE: The order in which the wavelengths are listed will be the order in which they are scanned.

How do I delete a Wavelength?

3. From the list of wavelengths, click on the wavelength that you wish to delete.

The wavelength is highlighted to show that it is selected.

4. Click **Remove**.

The wavelength is removed from the list.

How do I re-order the list of Wavelength settings?

The order of the wavelengths in the table will determine the order in which they are scanned.

To re-order to wavelengths:

1. From the list of wavelengths, click on the wavelength that you wish to move.

The wavelength is highlighted to show that it is selected.

2. Click **Move Up** to move the selected wavelength up one position in the list of wavelengths.

OR

Click **Move Down** to move the selected wavelength down one position in the list of wavelengths.

NOTE: **Move Up** and **Move Down** are only available when a wavelength is selected from the list.

What is the maximum number of Wavelengths that can be entered?

A maximum of 99 wavelengths can be entered.

What are the available Wavelengths?

Lambda 650/650R/650S	190.0–900.0 nm
Lambda 750/750S	190.0–3300.0 nm
Lambda 800/850	175.0–900.0 nm
Lambda 900/950/1050	175.0–3300.0 nm

How do I set the Ordinate Mode?

- Select the Ordinate Mode from the drop-down list.

The available options are A (absorbance), %T (transmittance), E1 (energy from sample beam), E2 (energy from reference beam), or %R (reflectance).

If E1 is selected as the Ordinate Mode, the sample beam is set as the front beam and the beam selection is disabled. If E2 is selected as the Ordinate Mode, the sample beam is set as the rear beam and the beam selection is disabled.

NOTE: Selecting E1 or E2 does not automatically set the slits to Fixed. However, the Slit mode and Ordinate Mode fields will turn red if E1 or E2 is selected with the slits not set to fixed.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

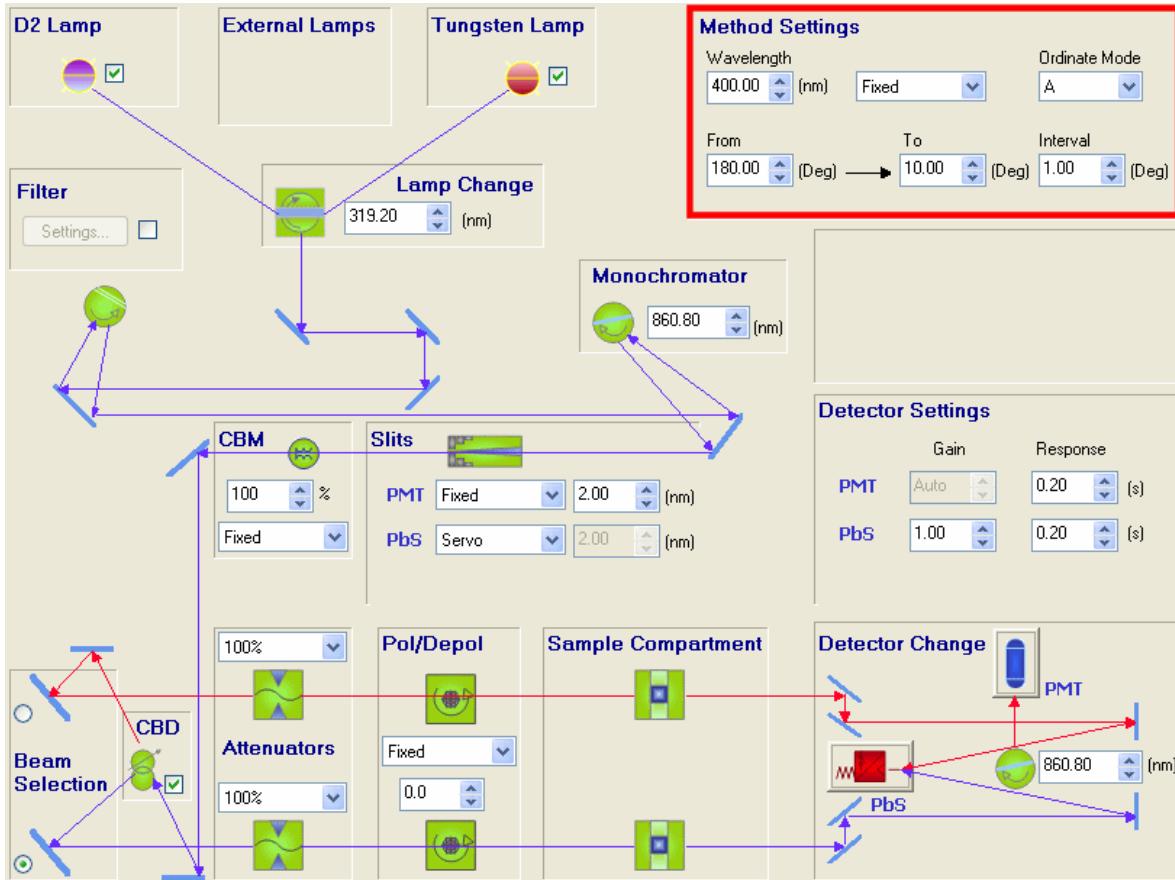
- Deselect Alignment mode.

See also

- Sample Info

Polarization Task – Data Collection

The Data Collection page for the Polarization Task can be used to alter the method settings such as the wavelength and start and end angles. For information about the generic settings on the Data Collection page, see the Data Collection page for High performance instruments. Only the method settings that are unique to the Wavelength Task are discussed below.



How do I define a polarization scan?

1. If you wish all your samples to be scanned at the same wavelength, select **Fixed**. If you wish to scan different samples at different wavelengths, select **Sample Table**.
2. If fixed is selected, enter the **Wavelength** at which the scan is performed.

OR

If Sample Table is selected, a wavelength column is added to the Sample Table. The wavelength specified on the Data Collection page is entered by default, but it can be altered on a per sample basis.

3. Enter the **From** angle.
The value must be between 10.00 and 330.00 degrees. Clicking / increases/decreases the angle by 0.05 degrees.

4. Enter the **To** angle.

The value must be between 10.00 and 330.00 degrees and must be greater than the start angle. The To angle must be less than the From angle. Clicking / increases/decreases the angle by 0.05 degrees.

5. Enter the **Interval**.

The Interval is the size of the increments between the start and ends polarization angles at which measurements will be taken. The value must be between 0.15 and 5.00 degrees. The scan range it must be a multiple of the Interval. Clicking / increases/decreases the angle by 0.05 degrees.

NOTE: When a Polarization scan is performed, the Instrument Status Bar shows the wavelength and the angle.

What are the available wavelengths?

Lambda 650/650R/650S	190.0–900.0 nm
Lambda 750/750S	190.0–3300.0 nm
Lambda 800/850	175.0–900.0 nm
Lambda 900/950/1050	175.0–3300.0 nm

How do I set the Ordinate Mode?

- Select the Ordinate Mode from the drop-down list.

The available options are A (absorbance), %T (transmittance), E1 (energy from sample beam), E2 (energy from reference beam), or %R (reflectance).

If E1 is selected as the Ordinate Mode, the sample beam is set as the front beam and the beam selection is disabled. If E2 is selected as the Ordinate Mode, the sample beam is set as the rear beam and the beam selection is disabled.

NOTE: Selecting E1 or E2 does not automatically set the slits to Fixed. However, the Slit mode and Ordinate Mode fields will turn red if E1 or E2 is selected with the slits not set to fixed.

How do I use Alignment mode?

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

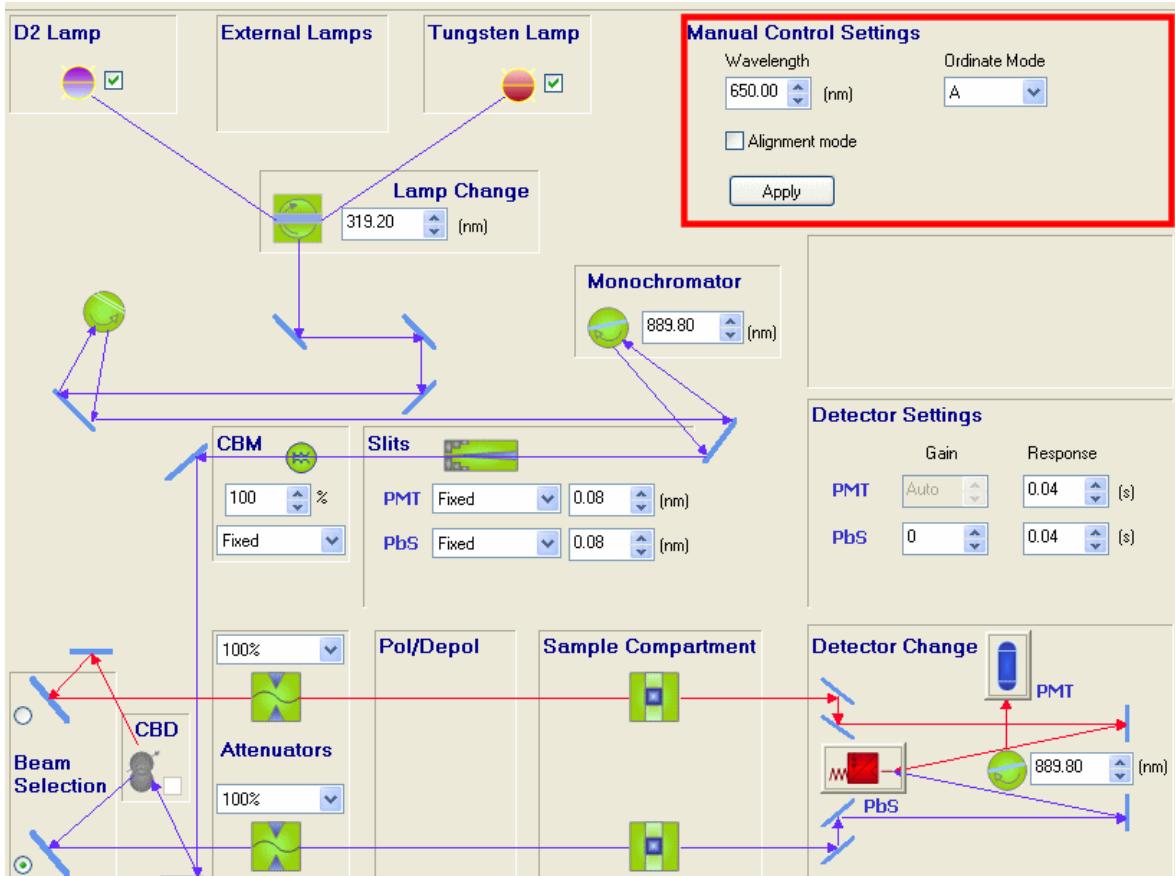
- Deselect Alignment mode.

See also

[Sample Info](#)

Manual Control – Data Collection

Manual control is a special method that allows manual control of the default instrument and enables you to look at the ordinate result at a particular wavelength, or switch to Alignment mode, without anything being saved to the database. It consists of only Instrument and Accessory settings, and cannot be used to collect data. A typical use of Manual Control would be for aligning an accessory. For information about the generic settings on this page, see the Data Collection page for High performance instruments. Only the main settings that are unique to Manual Control are discussed below.



NOTE: If any of the settings on the Data Collection page are changed, you must click **Apply** (in the top right of the dialog) for them to take effect.

How do I set the instrument to a particular wavelength?

- Enter or select the required wavelength and then click **Apply**.
The instrument will move to the required wavelength.

What is Alignment mode?

When you select **Alignment mode** and then click **Apply**, the instrument will enable the white light. The Instrument Status Bar will show the status of the Instrument during the this procedure and will read **Idle** and **0.0 nm** when it is complete.

To switch on the white light:

- Select **Alignment mode** and then click **Apply**.
The Instrument Status Bar shows 0.0 nm.

To switch the white light off:

- Deselect Alignment mode.

NOTE: When **Alignment mode** is selected, **Wavelength** is not available.

What is Optimize Gains?

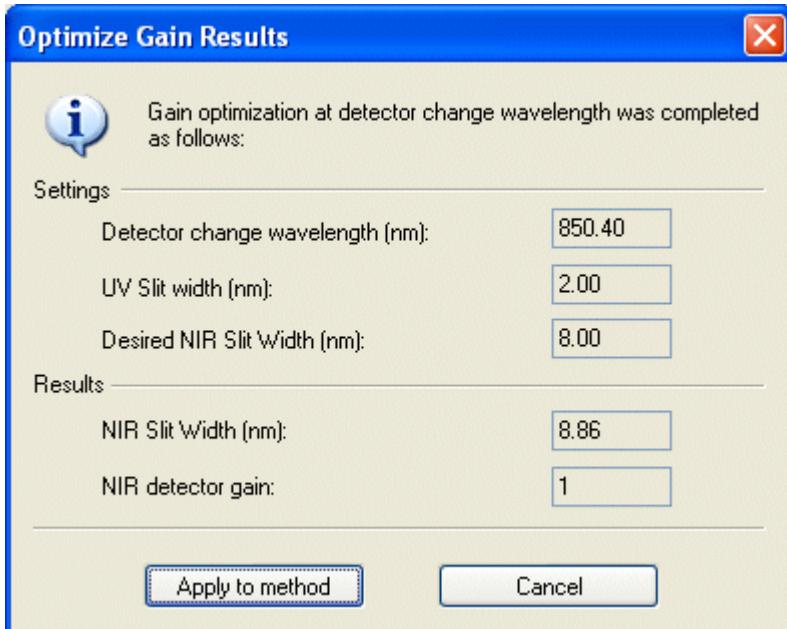
NOTE: **Optimize Gains** is only available for the Lambda 900, 950 and 1050.

Optimize gain adjusts settings to avoid any discontinuity at the detector change point caused by differences in detector linearity. The primary method is to adjust the gain to ensure that the Servo slit width control will stay within its operating range (0.2–20 nm).

The energy (and thus the slit position) is affected by the accessory fitted. Therefore, balance should be done with the accessory installed.

1. The NIR slit mode is set to **Servo**.
Servo is based on the reference beam energy level.
2. The NIR gain is set to 0.
3. The **Monochromator** is set at a point just to the NIR side of the detector change.
4. The slit width is measured.
The slit width should be at the mid-point of its range. If it is too high, the gain is increased and step 3 onwards is repeated.
5. The UV slit width is set to one-quarter of the size of the NIR slit width, due to differences in the monochromator grating.
6. The UV gain is set to 100.

When you select **Optimize Gains**, the Instrument Status Bar will update with the Instrument status. When the procedure is complete, a dialog will be displayed showing the result and asking whether you wish to apply the result to the current method:



Click **Apply to method** to apply the results to the current method, or click **Cancel** to reject the results and keep the current instrument settings.

How do I set the Ordinate Mode?

- Select the Ordinate Mode from the drop-down list.
The available options are A (absorbance), %T (transmittance), E1 (energy from sample beam), E2 (energy from reference beam), E1 & E2, or %R (reflectance).
If E1 is selected as the Ordinate Mode, the sample beam is set as the front beam and the beam selection is disabled. If E2 is selected as the Ordinate Mode, the sample beam is set as the rear beam and the beam selection is disabled.

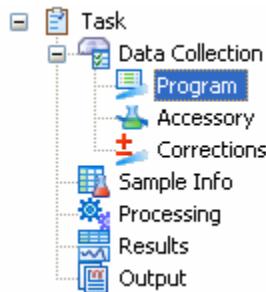
NOTE: Selecting E1 or E2 does not automatically set the slits to Fixed. However, the Slit mode and Ordinate Mode fields will turn red if E1 or E2 is selected with the slits not set to fixed.

NOTE: E1 & E2 mode allows you to monitor the energy from the sample and reference beams simultaneously.

See also

- Autosampler manual control
- Sipper manual control
- Cell changer manual control
- Peltier manual control

Program



The Program page is an "expert mode" that is only available for the High performance instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, 1050 WB and 1050 NB). The Program page allows you to create a custom setup of the instrument to reduce the detector jump on the Lambda 750, 900, 950 and 1050 spectrometers. It also enables you to create a custom slit-width profile across the scan.

For items defined on the Program page, the respective items must first be set to **Programmed** on the Data Collection page.

NOTE: The Program page is not available for Timedrive methods. In addition, the Program page is not available for E1 and E2 Ordinate Modes in Wavelength Program, Scan and Polarization methods.

NOTE: The InGaAs detector settings are only available if a Three Detector Module is installed and selected on the Accessory page.

What Types are available for programming?

The following settings are available:

Lambda 650/650R/650S – UV/Vis Detector Response, UV Slit Width.

Lambda 750/750S – UV/Vis Detector Response, UV Slit Width, NIR Detector Response, NIR Slit Width, NIR Detector Gain.

Lambda 800/850/850R – UV/Vis Detector Response, UV Slit Width, Sample Beam Attenuator, Reference Beam Attenuator.

Lambda 900/950/1050 WB/1050 NB – UV/Vis Detector Response, UV Slit Width, NIR Detector Response, NIR Slit Width, NIR Detector Gain, Sample Beam Attenuator, Reference Beam Attenuator.

NOTE: The respective items must be set to **Programmed** on the Data Collection page to define settings on the Program page.

Formatting the Program Table

How do I add a row to the table?

1. Click **Add**.

A row is added to the bottom of the table. The first column is the wavelength. The second column is the type of instrument setting, and the third column is the settings for the instrument setting.

NOTE: A cross  appears in the first column of the table until all the information for the row has been entered. It is then replaced by a tick .

2. Enter the required wavelength in the **Wavelength** field.

If the value is outside the acceptable range, the field remains red until the value is corrected. The range is 175.0–900.0 nm for Lambda 650, 650R, 650S, 800, 850 and 850R. For Lambda 750, 900, 950, 1050 WB and 1050 NB, the range is 175.0–3300.0 nm.

3. Click in the **Type** field.

A drop-down arrow appears.

4. Click  and select the type from the drop-down list.

5. Click in the **Settings** field and then click  or click **Settings**.

The appropriate Settings dialog is displayed.

How do I remove a row from the table?

- Click anywhere in the row you wish to remove and then click **Remove**.
The row is removed from the table.

How do I modify the settings for a particular row?

- Click in the **Settings** field you wish to modify and then click **Settings**.
The appropriate settings dialog is displayed.

In what order are the rows run?

The rows of the programming page are run in wavelength order starting with the lowest wavelength. This is regardless of the order in which the rows have been added to the table.

Can I sort the rows in the table?

Yes, the rows can be sorted by Wavelength or Type. The rows cannot be sorted using the Settings column.

1. Click on the title of the column (**Wavelength** or **Type**) to sort it.
An arrow appears to show how the column is ordered.
2. Click again on the column title to order the column in the reverse direction.

NOTE: This will not affect the order in which the program is run. It is just to aid viewing of the information in the table.

How many rows can be added?

If you are using a Lambda 650, 650R, 650S, 800, 850R or 850, a maximum of 40 rows can be added to the table. If you are using a Lambda 750, 750S, 900, 950, 1050 WB or 1050 NB a maximum of 60 rows can be added to the table.

It is only possible to have a maximum of 20 combined response settings (UV/Vis and NIR). For example, if you have 5 UV/Vis response values, you can only have 15 NIR (PbS, InGaAs, or PbS and InGaAs) detector response values.

It is only possible to have a maximum of 20 combined slit settings (UV and NIR). For example, if you have 12 UV slit values, you can only have 8 NIR slit values. If **Servo** is selected for the NIR slit, only one row of this type can be added to the table (giving a maximum of 19 UV/Vis rows).

It is possible to have 20 rows of NIR Gain settings. If you have a Three Detector Module installed, then this means that if you have 15 NIR PbS Gain settings you can only have 5 NIR InGaAs Gain settings.

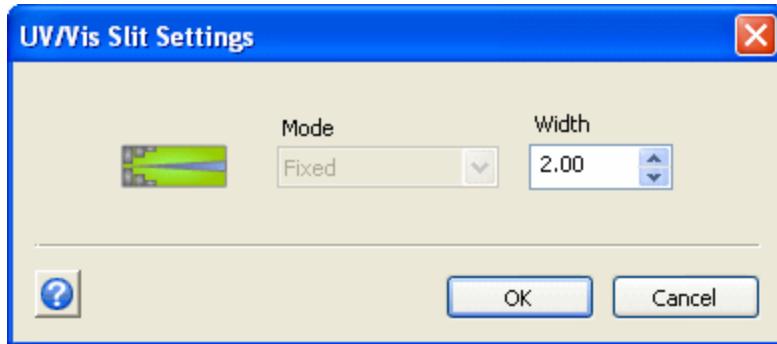
How do I select which rows of the table to run?

A tick indicates the row will be run. A cross indicates that the row will not be run.

Click to change it to and vice versa.

Instrument Settings

What settings are available for UV/Vis Slit Width?



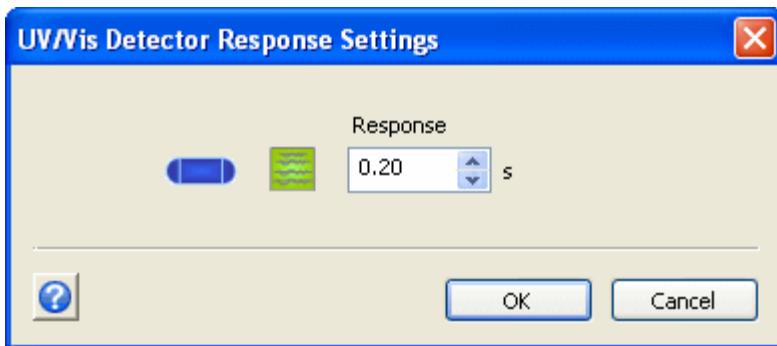
The UV/Vis Slit will operate in the range 175.0–850.0 nm.

NOTE: Only **Fixed** Mode is available.

- Select the required Slit **Width** (0.05–5 nm)

NOTE: It is only possible to have a maximum of 20 combined slit settings (UV/Vis and NIR). For example, if you have 12 UV/Vis slit values, you can only have 8 NIR slit values.

NOTE: If the Detector Change point is altered so that, a wavelength that fell in the NIR detector range now falls in the UV/Vis detector range, the Type will automatically update.

What settings are available for UV/Vis Detector Response?

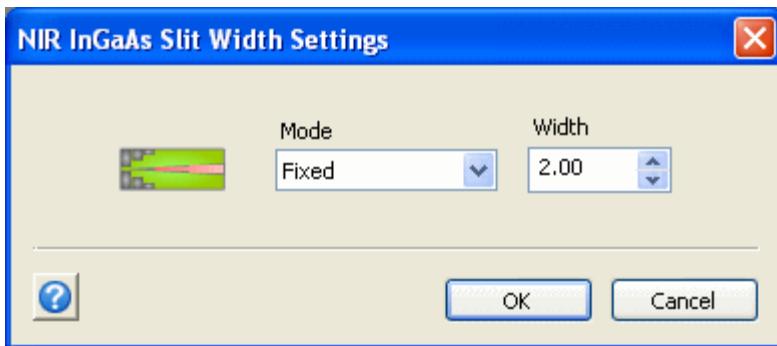
- Enter or select the required **Response** (0.04–10.00).

NOTE: It is only possible to have a maximum of 20 combined response settings (UV/Vis and NIR). For example, if you have 5 UV/Vis response values you can have only 15 NIR PbS response values.

NOTE: If the Detector Change point is altered so that, a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for NIR InGaAs Slit Width?

NOTE: The InGaAs detector settings are only available if a Three Detector Module is installed and selected on the Accessory page.



The NIR InGaAs Slit will operate in the wavelength range above the PMT to InGaAs detector change point specified on the Data Collection page and up to the InGaAs to PbS detector change point specified on the Data Collection page.

- Select the slit mode – **Fixed** or **Servo**.

If **Fixed** is selected, enter the slit width in the adjacent field.

The minimum is 0.2 and the maximum is 20.0.

If **Servo** is selected, the slit width field is grayed, and the system will monitor the reference beam energy and adjust the slits to avoid over saturation of the detectors.

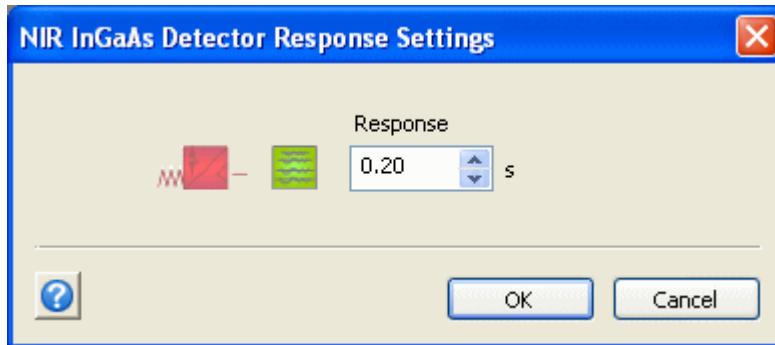
If **Servo** is selected, only one row of this type can be added to the table.

NOTE: It is only possible to have a maximum of 20 combined slit settings (UV and NIR). For example, if you have 12 UV slit values, you can have only 8 NIR slit values. If you have 6 UV slit values and 8 NIR PbS slit values, this means that you can have only 6 NIR InGaAs slit values.

NOTE: If the Detector Change point is altered so that, a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for NIR InGaAs Detector Response?

NOTE: The InGaAs detector settings are only available if a Three Detector Module is installed and selected on the Accessory page.



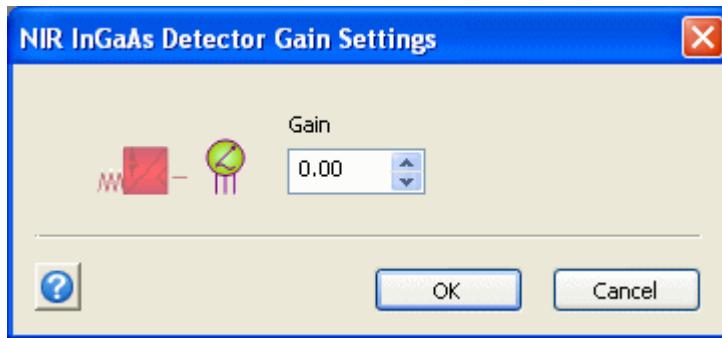
- Enter or select the required **Response** (0.04–10.00).

NOTE: It is only possible to have a maximum of 20 combined response settings (UV/Vis and NIR). For example, if you have 5 UV/Vis response values and 10 NIR PbS response values, you can have only 5 NIR InGaAs response values.

NOTE: If the Detector Change point is altered so that, a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for NIR InGaAs Detector Gain?

NOTE: The InGaAs detector settings are only available if a Three Detector Module is installed and selected on the Accessory page.



Defines the gain factor for the InGaAs detector.

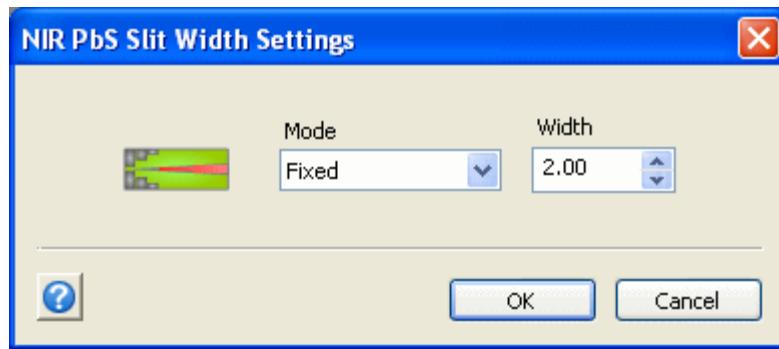
- Enter or select the required gain (0–20).

It is possible to have 20 rows of NIR Gain settings. This means that if you have 15 NIR PbS Gain settings, you can only have 5 NIR InGaAs Gain settings.

It is possible to enter 10 unique values only for the Gain of the InGaAs detector. Once 10 unique values have been used, the gain factor field becomes a drop-down list. For further gain settings you can then select a value from the drop-down list.

NOTE: If the Detector Change point is altered so that, a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for NIR (PbS) Slit Width?



If you are using a Lambda 750, 900 or 950, the NIR PbS slit will operate in the wavelength range above the PMT to PbS detector change point specified on the Data Collection page and up to the maximum wavelength of 3300 nm.

If you have a Three Detector Module installed and selected on the accessory page, the NIR PbS Slit will operate in the wavelength range above the InGaAs to PbS detector change point specified on the Data Collection page and up to the maximum wavelength of 3300 nm.

- Select the slit mode – **Fixed** or **Servo**.

If **Fixed** is selected, enter the slit width in the adjacent field.

The minimum is 0.2 and the maximum is 20.0.

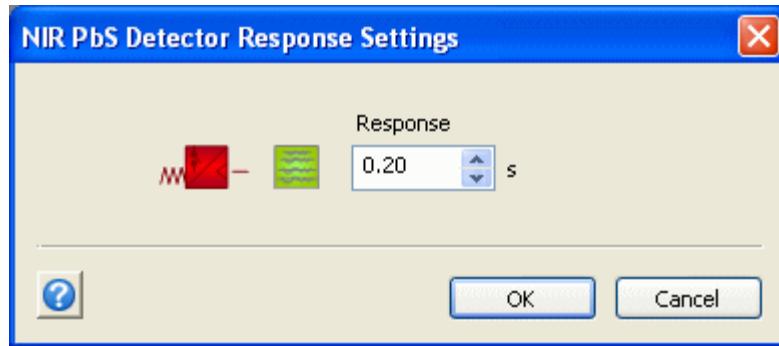
If **Servo** is selected, the slit width field is grayed, and the system will monitor the reference beam energy and adjust the slits to avoid over saturation of the detectors.

If **Servo** is selected, only one row of this type can be added to the table.

NOTE: It is only possible to have a maximum of 20 combined slit settings (UV and NIR). For example, if you have 12 UV slit values, you can have only 8 NIR slit values in total. If you have 6 UV slit values and 6 NIR InGaAs slit values, this means that you can have only 8 NIR PbS slit values.

NOTE: If the Detector Change point is altered so that a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for NIR (PbS) Detector Response?

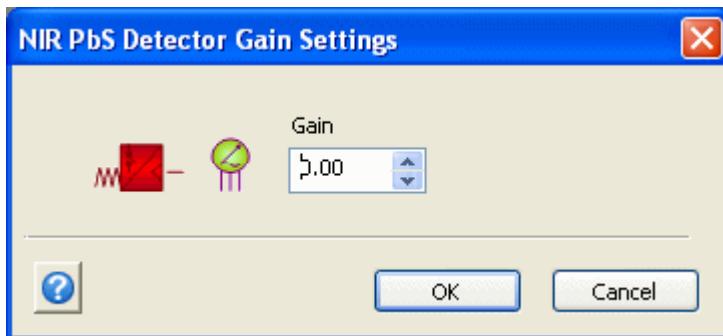


- Enter or select the required **Response** (0.04–10.00).

NOTE: It is only possible to have a maximum of 20 combined response settings (UV/Vis and NIR). For example, if you have 5 UV/Vis response values, you can only have 15 NIR response values. If you have 5 UV/Vis response values and 10 NIR InGaAs response values, you can have only 5 NIR PbS response values.

NOTE: If the Detector Change point is altered so that a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for NIR (PbS) Detector Gain?



Defines the gain factor for the lead sulfide (PbS) detector.

- Enter or select the required gain (0–12).

It is possible to have 20 rows of NIR Gain settings. This means that if you have 15 NIR InGaAs Gain settings, you can only have 5 NIR PbS Gain settings.

It is possible to enter 10 unique values only for the Gain of the PbS detector. Once 10 unique values have been used, the gain factor field becomes a drop-down list. For further gain settings you can then select a value from the drop-down list.

NOTE: If the Detector Change point is altered so that, a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

NOTE: If the Detector Change point is altered so that a wavelength that fell in the NIR detector range now falls in the UV detector range, the Type will automatically update.

What settings are available for the Sample and Reference Beam Attenuators?

Select **0%, 0.1% (Lambda 1050 only), 1%, 10% or 100%** from the drop-down lists for the front and rear beam attenuators.

The percentage values correspond to the amount of energy passing through the attenuator. 100% means that the beam is open. 0% means that the beam is closed.

For samples that have an absorbance >3 A, you should consider using the attenuators. For samples that have an absorbance >4 A, we recommend that you use the attenuators.

What happens if I specify wavelengths in the Program page that are outside the range specified for the method?

Any wavelengths that are outside the specified range for the method will not be run.

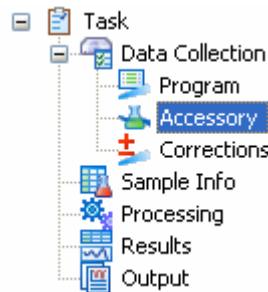
What happens if I change the Monochromator value?

The monochromator values specifies where the UV and NIR ranges change over (grating change point). Changing this value may affect the Type settings in the Program table.

For example, if a wavelength at 900 nm is specified as a UV Slit and then the monochromator value is changed to 890 nm, the UV Slit is changed automatically to NIR PbS Slit.

If a NIR gain has been specified at a wavelength that falls in the UV range due to the new monochromator value, the Type field will be updated.

Accessory



This page lists the available accessories that have been detected by the software. If an accessory is installed, it is detected and automatically selected here. Select the types of accessory that can be made available with the method.

If you wish to only allow the accessories you have selected to be available for the method, select **Only allow selected accessories**. This means that if an analyst tries to run the method without the correct accessory, an error message will be displayed informing them which accessory needs to be removed and which needs to be installed.

NOTE: Only allow selected accessories is only available when at least one accessory has been selected.

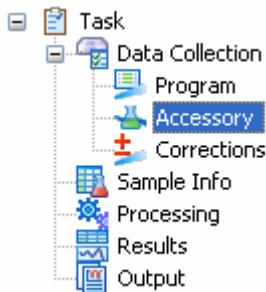
The following accessories are available. For further information see:

- Sipper
- Cell Changer
- Peltier
- Autosampler
- Universal Reflectance Accessory (URA)
- Three Detector Module.

NOTE: When you are within Manual Control, the Accessory page listing the available accessories is for reference only and cannot be edited.

If a new accessory is installed and it invalidates data in the sample table, these will automatically be replaced by intelligent defaults.

Sipper Accessory



Details for the Sipper, when installed, are shown on the Accessory page under Data Collection.

From here you can set the sample and delay times for sipping and the direction of pumping for the flush.

The Sipper can be started from either the buttons on this page or the controls on the Sipper itself.

What settings are available on the Sipper accessory page?

Type	Select the type of sipper from the drop-down list.
Description	Enter a description of the sipper. This will be saved with the method/task.
Sample time	Enter the amount of time (0.01–99.99 in seconds) that the pump pulls sample through for.
Delay time	Enter the delay time (0.01–99.99 in seconds) between the end of filling and the start of data collection. This is to allow for bubbles, turbulence, etc.
Flush	Select Flush for a forward flush. Enter the time (0.01–99.99 in seconds). Flush occurs after data collection.
Sample return	Select Sample return for a backwards flush. Enter the time (0.01–99.99 in seconds). The sample is returned to the original reservoir.
Time (seconds)	Allows you to set the time for flushing or returning the sample using the sipper.
Prompt before flushing the sample during data collection	Select for a prompt to be called before flushing the sample at the end of data collection.
Fill	Allows you to control the sipper directly. Starts to fill the flowcell as set by the Sample time .
Flush/Return	Allows you to control the sipper directly. Starts to flush the flowcell as set by the selection of Flush or Return and the Time .

What messages are displayed in the Instrument Status Bar when I am using a Sipper?

When the sipper is in use, the Instrument Status Bar shows the current status of the sipper as either **SIPPING** or **FLUSHING** depending on the button clicked.

What columns are added to the sample table when a sipper is installed?

Three columns are added to the sample table – **Fill Time**, **Flush/Return Time**, and **Delay Time**.

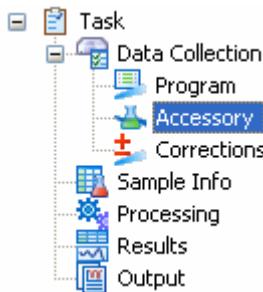
Fill Time (secs) [Sipper]	Flush/Return Time (secs) [Sipper]	Delay Time (secs) [Sipper]
Default	Default	Default

For further information about completing the sample table, see Sample info.

What is displayed on the sipper accessory page when I am in manual control?

Type	Select the type of sipper from the drop-down list.
Description	The Description field cannot be edited when you are in Manual Control.
Sample time	Enter the amount of time (0.01–99.99 in seconds) that the pump pulls sample through for.
Delay time	Enter the delay time (0.01–99.99 in seconds) between the end of sampling and the start of data collection. This is to allow for bubbles, turbulence, etc.
Flush	Select Flush for a forward flush. Enter the time (0.01–99.99 in seconds).
Sample return	Select Sample return for a backwards flush. Enter the time (0.01–99.99 in seconds).
Fill	Allows you to control the sipper directly. Starts to fill the flowcell as set by the Sample time and Delay time .
Flush	Allows you to control the sipper directly. Starts to flush the flowcell as set by the selection of Flush or Return and the Time .

Peltier Accessory



Sampling accessories such as cell changers can be used with temperature control. Peltier control cell changers heat the blocks on the cell changers to the specified value. An external probe can also be used to monitor the temperature on an individual cuvette. The temperature probe can be used on its own or with a Peltier cell changer.

How do I set up the Peltier?

1. Select the **Type** of Peltier (**Routine**) from the drop-down list.
2. Enter a Description.
The description is saved with the Method.
3. If you wish to use the Peltier, select **Peltier**.
4. If you wish to use the Temperature probe, select **Temperature Probe**.
5. Enter a Target Sample Temperature (15–45 °C).
Pressing / , increases/decreases the target temperature by 0.1 °C.
6. If you wish to use a temperature probe, click in the **Temperature probe** check box.

Can I have a temperature probe without a Peltier?

Yes, an external probe can also be used to monitor the temperature on an individual cuvette.

The temperature probe can be used on its own or with a Peltier cell changer.

NOTE: When the temperature probe is used with the Peltier, data collection will not start until the temperature probe reads the temperature specified for the Peltier. For example, if the Peltier has been set to 37 °C , data collection will not start until the temperature probe reads 37 °C.

What is the temperature range of the Peltier?

The temperature range is 15–45 °C. Pressing / , increases/decreases the target temperature by 0.1 °C.

How can I switch the Peltier on before I run a method?

If you wish to warm the cell changer before you start the method, click **Peltier on**. This will heat the cell changer to the required temperature.

When the Peltier is on, the button becomes **Peltier Off**. Click **Peltier Off** to turn the Peltier off.

What columns are added to the Sample Table when a Peltier and/or temperature probe are installed?

A new column – Target Peltier Temp °C is added to the sample table when a Peltier is installed.

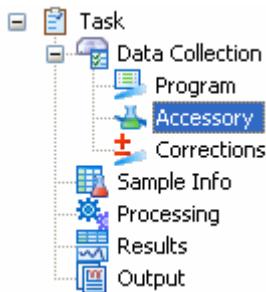
For further information about this column see Sample info.

What is displayed on the Peltier accessory page when I am in Manual Control?

The Manual Controls are the same as normally seen on the Peltier Accessory page, except that the Description field is not available.

NOTE: Within Manual Control the settings are not saved.

Autosampler Accessory



How do I select the type of Autosampler?

- Select the **Type** from the drop-down list.

Can I save a description of the Autosampler as part of my Method?

Yes, if you enter text in the **Description** field, this is saved as part of the Method.

What tray layouts are available for the Autosampler?

A tray layout is contained in a tray definition file (*.try). The tray definition file provides the system with information about the sample racks installed on the autosampler. This allows the sampling probe to be moved to the correct location for each container in the rack.

- Select the required tray layout from the drop-down list.
A graphical layout of the tray is shown below the drop-down list.

NOTE: If Custom Tray is selected, no graphical layout is displayed.

Tray Group E (200 sample positions and a rinsing port location)	<p>Comprises:</p> <ul style="list-style-type: none"> • Rack B3140621 (rinsing-port rack) with rinsing port location (location 0) and 20 locations [30 mm diameter] for 50 ml solution containers (calibration and/or test sample solutions) <p>and</p> <ul style="list-style-type: none"> • Rack B3140617 (2x), each with 90 locations [13 mm square] for 6 ml and 8 ml solution containers (test sample solutions).
--	--

	<p>Rinsing port location</p> <p>B3140621</p> <p>B3140617 (2x)</p>
Tray Group F (157 sample positions and a rinsing port location)	<p>Comprises:</p> <ul style="list-style-type: none"> • Rack B3001647 (rinsing-port rack) with rinsing port location (location 0), 8 locations [30 mm diameter] for 50 ml solution containers (calibration and/or test sample solutions) , and 29 locations [16 mm diameter] for 15 ml solution containers (test sample solutions) <p>and</p> <ul style="list-style-type: none"> • Rack B3140618 (2x), each with 60 locations [16 mm diameter] for 15 ml solution containers (test sample solutions). <p>Rinsing port location</p> <p>B3001647</p> <p>B3140618 (2x)</p>
Tray Group G (62 sample positions and a rinsing port location)	<p>Comprises:</p> <p>Rack B3140621 (3x) with 21 locations [30 mm diameter], each for 50 ml solution containers (calibration and/or test sample solutions). Location 0 in the first rack is for the rinsing port.</p>

	<p>Rinsing port location</p> <table border="1"> <tbody> <tr><td>0</td><td>7</td><td>14</td></tr> <tr><td>1</td><td>8</td><td>15</td></tr> <tr><td>2</td><td>9</td><td>16</td></tr> <tr><td>3</td><td>10</td><td>17</td></tr> <tr><td>4</td><td>11</td><td>18</td></tr> <tr><td>5</td><td>12</td><td>19</td></tr> <tr><td>6</td><td>13</td><td>20</td></tr> <tr><td>21</td><td>28</td><td>35</td></tr> <tr><td>22</td><td>29</td><td>36</td></tr> <tr><td>23</td><td>30</td><td>37</td></tr> <tr><td>24</td><td>31</td><td>38</td></tr> <tr><td>25</td><td>32</td><td>39</td></tr> <tr><td>26</td><td>33</td><td>40</td></tr> <tr><td>27</td><td>34</td><td>41</td></tr> <tr><td>42</td><td>49</td><td>56</td></tr> <tr><td>43</td><td>50</td><td>57</td></tr> <tr><td>44</td><td>51</td><td>58</td></tr> <tr><td>45</td><td>52</td><td>59</td></tr> <tr><td>46</td><td>53</td><td>60</td></tr> <tr><td>47</td><td>54</td><td>61</td></tr> <tr><td>48</td><td>55</td><td>62</td></tr> </tbody> </table> <p>B3140621 (3x)</p>	0	7	14	1	8	15	2	9	16	3	10	17	4	11	18	5	12	19	6	13	20	21	28	35	22	29	36	23	30	37	24	31	38	25	32	39	26	33	40	27	34	41	42	49	56	43	50	57	44	51	58	45	52	59	46	53	60	47	54	61	48	55	62
0	7	14																																																														
1	8	15																																																														
2	9	16																																																														
3	10	17																																																														
4	11	18																																																														
5	12	19																																																														
6	13	20																																																														
21	28	35																																																														
22	29	36																																																														
23	30	37																																																														
24	31	38																																																														
25	32	39																																																														
26	33	40																																																														
27	34	41																																																														
42	49	56																																																														
43	50	57																																																														
44	51	58																																																														
45	52	59																																																														
46	53	60																																																														
47	54	61																																																														
48	55	62																																																														
Custom Tray	<p>In addition to trays E, F and G, you can also use racks offered by Gilson. These are 'Custom Trays' within UV WinLab. When you select Custom Tray, a standard browse dialog appears allowing you to select the required tray file. Tray files are .csv files containing the tray layout, and must have the extension *.try. When the file has been selected, it will then appear in the drop-down list of tray types.</p> <p>To install the Gilson racks on the Autosampler you require an adapter plate (option). Four Gilson racks can be installed on the Autosampler.</p> <p>NOTE: Gilson racks cannot be obtained from PerkinElmer.</p>																																																															

What is the Autosampler Sipper?

This is the sipper located on the side of the Autosampler.

NOTE: If **Enable Autosampler Sipper** is de-selected, an external sipper must be installed. This external sipper is controlled (Fill / Flush) from the Sipper Accessory page which will be added to the Workspace Folder List when the accessory is installed. The sample time, delay time, and flush/return settings are all disabled on the Autosampler page.

What is the Sample time?

The **Sample time** is the time for which a sample is sipped from the sample container.

- Enter a value for the **Sample time**.
The limits are 0.00–99.99 seconds.

What is the Delay time?

The **Delay time** is the time between the end of sampling and the start of measurement. The **Delay time** allows the sample to settle before the measurement is taken.

- Enter a value for the **Delay time**.
The limits are 0.00–99.99 seconds.

What is the difference between Flush and Sample return?

NOTE: **Flush** and **Sample return** are only available if **Enable Autosampler sipper** is selected. If **Enable Autosampler sipper** is de-selected, an external Sipper Accessory must be installed, and the **Flush** and **Sample return** times have to be set on the sipper accessory page.

The **Flush** and **Sample return** radio buttons allow you to specify what should happen to the sample after it has been measured.

Flush – flushes the sample to the waste bottle for the specified period of time.

Sample return – returns the sample to the original sample container for the specified period of time.

- Enter a value for the Flush / Sample return time.
The limits are 0.00–99.99 seconds.

How do I specify the Autozero position?

- Select the position for the Autozero from the **Fill cell from position** drop-down list. This cell position is then not available as a sample position within the Position [Autosampler] column in the Sample Table (in the drop-down list). The Autozero will be performed before the samples are analyzed regardless of the position specified.

What does the Fill button do?

Fill initiates an immediate fill action. The duration of the Fill is the Sample time.

NOTE: The **Fill** button is only available if Enable Autosampler Sipper is selected. If it is de-selected, **Fill** must be initiated from the Sipper Accessory page.

What does the Flush / Return button do?

Initiates an immediate Flush or Return. What happens to the sample after it has been measured (flushed or returned) depends on the radio button selected above in the dialog. The duration of the Flush/Return is the value entered in the Time field adjacent to the Flush/Return radio buttons.

NOTE: The **Flush/Return** button is only available if Enable Autosampler Sipper is selected. If it is de-selected, **Flush/Return** must be initiated from the Sipper Accessory page.

What does the Goto Position button do?

Clicking **Goto Position** displays the Goto position dialog that allows you to send the Autosampler probe to a specific cell, XYZ position or have direct control of the Autosampler.

What does the Reset to Home button do?

Clicking **Reset** moves the Autosampler probe to the rinse port position.

What does the Initialize button do?

Clicking **Initialize** initializes the Autosampler and moves the probe to the rinse port position.

What does the Depth button do?

Clicking **Depth** displays the Depth dialog allowing you to set the depth of the probe in the sample tubes.

Which methods can be used with the Autosampler?

The Autosampler can be used with the following methods:

- Scan
- Wavelength Program
- Timedrive
- Scanning Quant
- (Wavelength Program) Quant
- Polarization.

NOTE: The Polarization method is only available for High Performance Instruments (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, 1050 WB and 1050 NB).

What columns are added to the Sample Table when the Autosampler is installed?

Six columns are added to the Sample Table – **Position**, **Probe Depth**, **Return Cell**, **Fill Time**, **Flush/Return Time** and **Delay Time**. The values in each of these columns are specified on a per sample basis. This means that you can use different settings for every sample if you require.

Position [Autosampler]	Probe Depth [Autosampler]	Return Cell [Autosampler]	Fill Time (secs) [Autosampler]	Flush/Return Time (secs) [Autosampler]	Delay Time (secs) [Autosampler]
1. Default	Rinse Port	Default	Default	Default	Default

If **Default** is specified in the sample table, the Autosampler Accessory page settings are used.

For further information about completing the Sample Table, see Sample info.

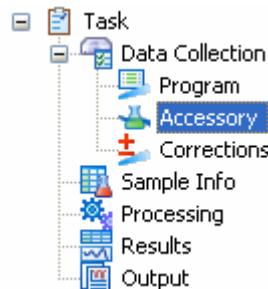
What can I do with the Autosampler when I am in Manual Control?

All the settings described above are available. However, a description cannot be entered and information is not saved. Manual Control allows you to control the Autosampler without performing a task.

See also

- Autosampler Goto Position
- Autosampler Depth dialog

Autosampler Accessory – Goto position



What does the Goto position dialog do?

The dialog allows you to move the Autosampler probe to a specified position by either entering a **cell position**, **XYZ position** or using **direct control**.

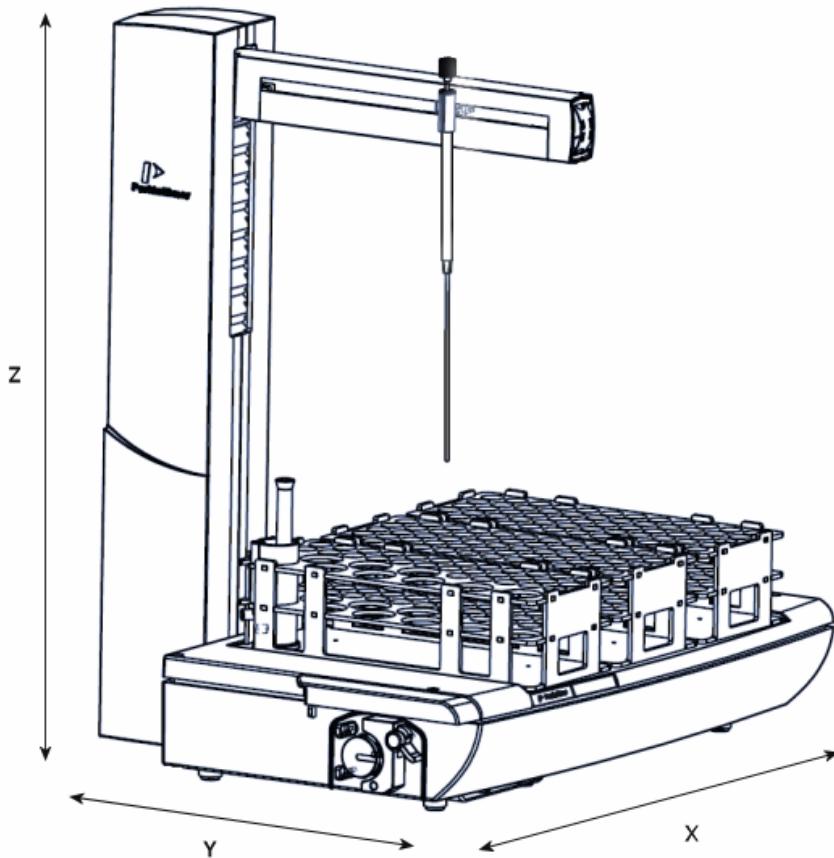
How do I move the autosampler probe to a cell position?

1. Select Cell position.
2. Enter a value in the field below.
The allowable range depends on the currently selected tray units. The current position is displayed in the XYZ position fields.
3. Click **Apply** to move the probe to the specified position and keep the dialog open, or click **OK**, to move the probe to the specified position and close the dialog.

How do I move the autosampler probe to a specific XYZ position?

This allows the user to move the autosampler probe using XYZ coordinates.

It can be used with Trays E, F and G, where slim sample tubes may be being used, and this will enable the probe to be successfully directed into the tube. It can also be used to determine a custom tray layout where cell positions are not defined. For example, your sample may contain solution and sediment and you wish to sample from the solution and prevent the probe entering the sediment. By using XYZ you can determine the required height of the probe for the measurement.



1. Select XYZ position.
2. Enter the required values or use and to scroll to the values in the X, Y and Z fields.

The limits are:

X	0– 310 mm
Y	0–255 mm
Z	0–146 mm

A single click of or increases / decreases the value by 1.0 mm.

3. Click **Apply** to move the probe to the specified position and keep the dialog open, or click **OK**, to move the probe to the specified position and close the dialog.

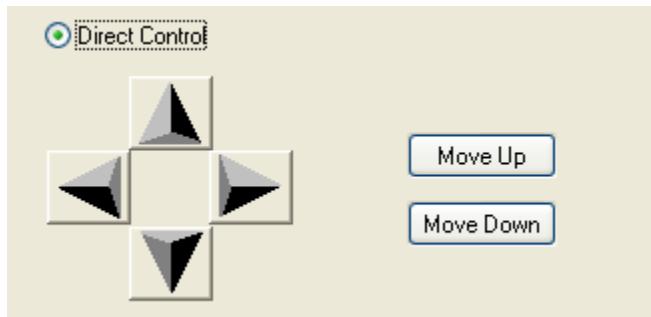
NOTE: If the probe is in a cell and you click **Apply** or **OK**, the probe will move up out of the cell before moving to the new location.

How do I move the autosampler probe using direct control?

Direct control allows you to move the probe in increments of 1 mm.

- Click the required arrow to move the probe one unit (1 mm).
The probe moves instantly and the XYZ coordinates are also updated.

NOTE: The longer the button is held down, the faster the probe will move (until the limits are met or the button is released).



The left and right arrows control movement in the X direction.

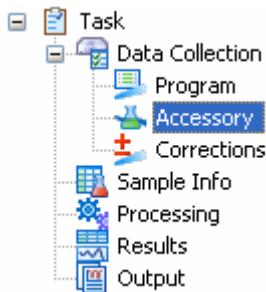
The up and down arrows control movement in the Z direction.

Move Up and **Move Down** control movement in the Y direction.

This allows you to fine tune a custom analysis or produce a custom tray layout. You can only move +/- 10 units past the minimum height values for the cells or centered positions for the cells.

NOTE: You must ensure that the value on the Goto depth dialog does not exceed the value set on the Autosampler Depth dialog. No warning message will be displayed.

Autosampler Accessory – Depth



What does the Depth dialog do?

The depth dialog allows you to specify the depth of the probe in the solution containers. This can be used to prevent the probe from stirring up sediment that may be in the samples.

How do I set the depth of the probe?

There are two ways of setting the probe on the dialog:

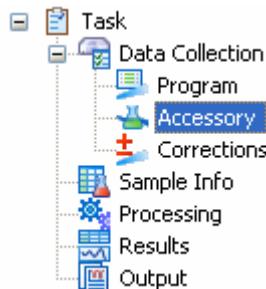
- Drag the slider control to the required position between the maximum and minimum heights.
The maximum height (0 mm) is the fully withdrawn probe. 146 mm is the maximum distance that the probe can descend into the tube.

The value in the field below the slider control automatically updates.

OR

- Enter the required value, or use and to update the value in the field.
The slider control updates automatically to reflect the new value.

Cell Changer Accessory



The Accessory page for the Cell Changer allows you to select the cell positions for the samples and to define when an auto zero will be performed.

The selections on this page will determine what cell positions are available in the Sample Table. By default the table will be populated in order, but this can be changed unless the method is locked.

NOTE: When installing a cell changer, first switch off the Lambda spectrometer. Install the accessory and then reboot the spectrometer so that the Cell Changer is initialized.

Cell Changer for Medium Performance Lambda Spectrometers (Lambda 20, 40, 40P, 20Bio, 40Bio, 25, 35, 45)

What Cell Changers are available?

The following cell changers are available:

- 5 + 1
- 6 + 1
- 8 + 1
- 9 + 1
- 13 + 1.

What does X + 1 mean?

X + 1 is the number of positions in the cellchanger. There is 1 reference position and X sample positions.

How do I setup the Cell Changer?

1. Select the **Type** from the drop-down list.

Auto Type allows you to set up a method with a cell changer without knowing the type of cell changer that will be used when the method is run. Auto is displayed in the Cell and Carousel columns in the sample table. When the task is run, the type of cell changer is automatically detected and the Sample Table is updated and you can then specify cell positions.

2. Enter a **Description** of the Cell Changer.

3. Select Autozero every sample or Autozero at position.

Autozero every sample – Select this option to perform an autozero before every sample.

Autozero at position – Select this option to perform an autozero before specified sample positions.

4. If **Autozero at position** is selected, select the position from the drop-down list.
5. Select which carousel positions are to be enabled.
Click in the check box. indicates that the position is selected and it will appear in the Sample Table.

NOTE: Enabled Positions is not available when Auto Type is selected.

How do I perform an autozero at every sample position?

- If you wish to perform an auto zero at every selected position, select **Autozero every sample**.
Before the samples are run, an autozero will be run at each selected position.

How do I perform one autozero per task with a cell changer installed?

1. Select Autozero at position.
2. Select the position from the drop-down list.

How do I define which sample is at a particular cell changer position?

When you are using a cell changer, a column called Cell is added to the Sample Table.

3. Select a sample in the Sample Table and click in the Cell field.
An drop-down arrow  appears.
4. Click on the arrow and select the cell position for the sample from the drop-down list.
The list of available cell positions is dependent on the check boxes selected on the Accessory page.

NOTE: If you have more samples than available positions in the cellchanger, you can select a position more than once. The software will prompt you to insert the correct sample.

How do I specify which cell positions are enabled?

- In the Enabled Positions section of the dialog, click in the check box.
 indicates that the position is selected and it will appear in the Sample Table.

NOTE: This is not available when **Auto Type** is selected.

Cell Changer for High Performance Lambda Spectrometers (Lambda 650, 650R, 650S, 750, 750S, 800, 850, 900, 950, 1050 WB and 1050 NB)

What Cell Changers are available?

The following cell changers are available:

- 8+8
- 5+5
- 6+6
- 9+9.

How do I set up the Cell Changer?

1. Select the **Type** from the drop-down list

Auto Type allows you to set up a method with a cellchanger without knowing the type of cell changer that will be used when the method is run. Generic entries are made in the Sample Table. When the task is run, the type of cell changer is automatically detected and the Sample Table is updated. If a method is created when the Cell Changer is installed, the software will automatically detect the type of cell changer.

2. Enter a **Description** of the Cell Changer.

This information is saved as part of the method.

3. Select Single reference in position, Single reference in front beam And single reference in rear beam, or Matched pairing of samples to reference.

Single reference in position –

Select this option to specify a single reference cell position. This can be in the front or rear carousel.

➤ Select the position of the reference from the drop-down list.

Rear or front is specified depending on the position of the reference beam as specified on the Data Collection page.

Single reference in front beam And single reference in rear beam –

Select this option to specify two reference cell positions.

4. Select the front beam reference position from the drop-down list.

5. Select the rear beam reference position from the drop-down list.

Matched pairing of samples to reference

Select this option to use matched pair sample and references. This means that the reference for sample 1 will be at position 1, the reference for sample 2 will be at position 2 and so on.

- Select Autozero every sample or Autozero at position.

Autozero every sample –

Select this option to perform an autozero before every sample in the selected (enabled) cell positions.

Autozero at position –

Select this option to perform an autozero at selected position(s).

6. Select the **Front** position from the drop-down list.
7. Select the **Rear** position from the drop-down list.

The software will prompt you to remove all samples before the autozero(s) are performed.

NOTE: It is not possible to perform an autozero in the reference position.

Sample positions

- Select which carousel positions are to be enabled.
Click in the check box. indicates that the position is selected and it will appear in the Sample Table.

NOTE: Autozero options are ignored if baseline Corrections are turned off

Enable front positions and Enable rear positions are available for Single reference in front beam and single reference in rear beam.

Either **Enable front positions** or **Enable rear positions** will be available for **Single reference in position** depending on the position of the reference beam. For example, if the reference beam is specified as the rear beam, the Single reference in position option will be for the rear beam, and **Enabled front positions** will be available.

By definition, only Enable front positions is available for Matched pairing of samples to reference.

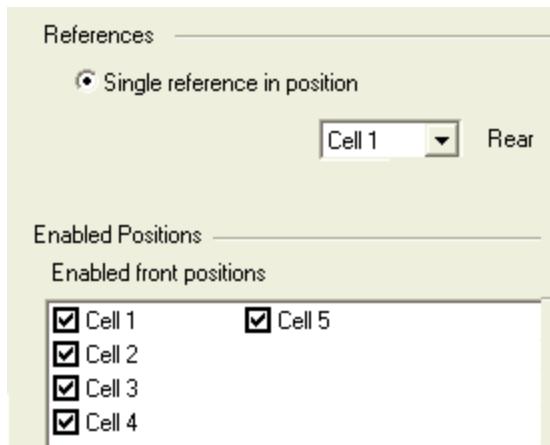
NOTE: Enabled Positions is not available when **Auto** Type is selected.

How do I define the positions in the Cell Changer that will contain samples?

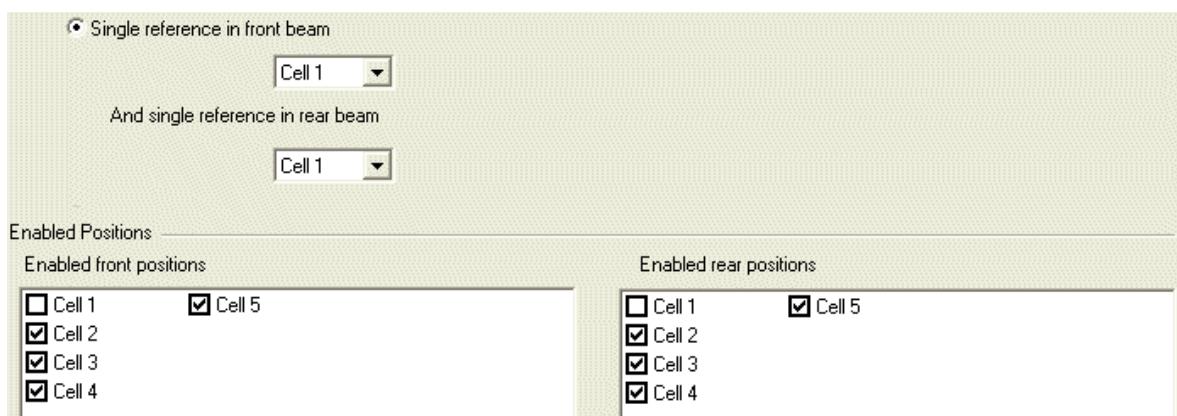
The number of available cells (represented by check boxes) depend on the type of cell changer installed and the type of reference specified.

Example – a 5+5 Cell Changer is installed.

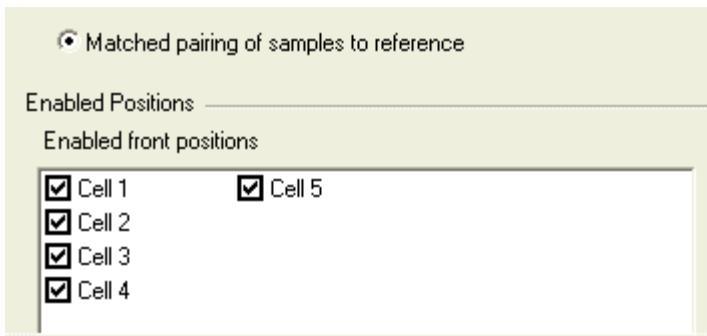
If **Single reference in position** is selected, and the reference beam is the rear beam, there will be 5 cell positions in the front carousel available for samples. The Cell Changer is acting as a X + 1 cell changer (X samples and 1 reference). The reference in the back carousel is used for all samples.



If **Single reference in front beam and single reference in rear beam** is selected, all positions except those specified as reference positions will be available. Therefore, there will be 4 sample positions in the front carousel and 4 sample positions in the rear carousel. The Cell Changer is acting as a $(2X-2)+2$ Cell Changer [$(2X-2)$ sample positions and 2 reference positions]. The reference in the back carousel is used for all the samples in the front carousel, and then the reference in the front carousel is used for all the samples in the back carousel.



If **Matched pairing of samples to reference** is selected, and the reference beam is the rear beam, all 5 samples positions will be available in the front carousel. The Cell Changer is acting as a 5 + 5 Cell Changer (5 samples and 5 references). Each sample in the front carousel uses the reference in the corresponding position in the rear carousel cell. For example, the sample in cell 1 will use the reference in cell 1 in the rear carousel.



- To select the cells that will contain samples, select the check box adjacent to the cell number.
A tick mark indicates that the cell will contain a sample.
On the Sample Table, only these positions will be available from the drop down list in the Cell column.

How do I perform an autozero at every sample position?

- If you wish to perform an autozero at every selected position, select **Autozero every sample**.
Before the samples are run, an autozero will be run at each selected position. The software will prompt you to remove the samples before the autozeros are performed.

NOTE: It is not possible to perform an autozero in the reference position.

How do I perform an autozero at a specific position(s)?

1. Select the **Front** position from the drop-down list.
2. Select the **Rear** position from the drop-down list.
The software will prompt you to remove all samples before the autozero(s) are performed.

NOTE: It is not possible to perform an autozero in the reference position.

How do I specify which cell positions are enabled?

- In the Enabled Positions section of the dialog, click in the check box.
 indicates that the position is selected and it will appear in the Sample Table. Only the positions selected appear in the sample table. If there are more samples specified than the number of available cell positions, the cell changer will re-index (back to the first specified sample position) and the software will prompt you for the new samples.

NOTE: This is not available when **Auto Type** is selected. The sample table is populated with the number of samples specified on the sample info page and the table is completed at run time.

What columns are added to the Sample Table when the Cell Changer is installed?

When you are using a cell changer, columns called Cell and Carousel are added to the Sample Table.

Cell	Carousel
2 Front	

For information on completing the Sample Table see Sample info.

What is displayed in the Sample Table if Auto is selected as the Type?

Two columns are added to the table – **Cell** and **Carousel**. Both are populated as **Auto** for every sample and this cannot be edited. At run time, the software will detect the type of cell changer and populate the sample table after data collection.

Cell	Carousel
Auto	Auto

Manual Control for Cell Changer

What is Cell Changer Manual Control?

The Cell Changer Manual Control allows direct control of the Cell Changer. It enables you to go to a specific position on the Cell Changer.

When you are within Manual Control select the Cell Changer node of the Folder List.

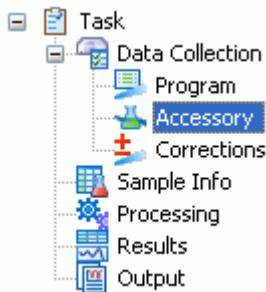
How do I go to a specific position?

NOTE: The Type of Cell Changer installed is automatically detected by the software and cannot be changed within Manual Control.

1. Select **Front** or **Rear** carousel from the drop-down list.
2. Select the cell position.
3. Click **Goto**.

The Cell Changer moves the selected cell into the beam.

Universal Reflectance Accessory (URA)



The Universal Reflectance Accessory (URA) is a variable angle reflectance accessory that loads directly into the detector compartment of the Lambda 650/850/750/950/1050 spectrometers. It can also be used with the Lambda 800 and 900.

The URA can be used to collect absolute or relative reflectance measurements of samples measuring from 5 mm to greater than 150 mm. Samples less than 5 mm in size can be used with a sample holder.

Data Collection

What types of data collection are available?

There are three types of data collection – Absolute Reflectance Mode, Relative Reflectance Mode, and Transmission (Sample Compartment) Mode.

Absolute Reflectance Mode – The baseline is collected from the baseline position. The sample spectra are collected from the sample position.

NOTE: When Absolute Reflectance Mode is selected, you are not prompted for the baseline correction as it will be collected automatically by the software.

Relative Reflectance Mode – The baseline and the sample spectra are collected from the sample position. In Relative Reflectance Mode, you are prompted to place a reference material at the sample position when the baseline (autozero) is collected.

Transmission (Sample Compartment) Mode – This allows you to use the spectrometer sample compartment with the URA still installed. The cover must be placed over the URA sample position to prevent stray light, although it is not used for data collection.

NOTE: Transmission (Sample Compartment) Mode is only available when %T or A Ordinate mode is selected on the Data Collection page.

Angle

What range of incident angles can be used?

The URA can be used to collect absolute or relative reflective measurements with an angle of incidence in the range 8°–65° in 0.5° increments.

How do I measure many angles from one sample?

It is possible to measure many angles of the same sample without user intervention using Measurements within the Sample Table.

See Sample Info for further information on how to set up the Sample Table.

How do I use the Angle setting on the URA page?

The Angle setting on the URA is the default value that will appear in the sample table for all samples. This value can be altered for each sample from within the sample table.

You can set the angle on this page and select **Sample** or **Baseline**. If you then click **Apply** the URA will move to the selected angle in the Sample or Baseline position (depending on the radio button selected) and set the spot size.

Enter values for the Angle or click / to increase / decrease the value by 0.5 degrees.

Spot size

How do I specify the spot size?

You can select Physical setting or Instrument setting:

Physical setting – Enter values for the **Width** and **Length** or click / to increase / decrease the value by 0.1 mm.

The width does not have to be the same value as the length.

If the width and/or length value is altered on the Sample Info page, this supercedes the value set on the Accessory page.

Altering the Physical setting (Width and Length) automatically alters the Instrument settings (Slit width and CBM).

Instrument setting – Enter values for the Slit width and CBM (common beam mask) or click / to increase / decrease the values (by 0.1 for the Slit width, and 1 for the CBM(%)).

Altering the Instrument setting (Slit width and CBM) automatically alters the Physical setting (Width and Length).

What is the effect of reducing the spot size?

Reducing the spot size increases noise in the data collected. Reducing the spot width (slit width) increases spectral resolution.

How does the spot size remain constant as the angle is increased?

To maintain a constant spot size as the angle is increased, the Common Beam Mask (CBM) is reduced.

Actions

What does the Transit button do?

This button is only for use when you wish to ship the URA. Clicking **Transit** moves the internal hardware into a safe position to avoid damage during shipment.

NOTE: Clicking **Transit** is not necessary if you are only removing the URA and placing it on the bench.

What does the Alignment button do?

This button is only available if you have Service permission. Clicking **Alignment** displays the Alignment dialog. **DO NOT ALTER ANY OF THE SETTINGS ON THIS DIALOG AS THIS WILL RESULT IN YOUR URA BECOMING MISALIGNED. THIS DIALOG IS FOR USE BY PERKINELMER SERVICE ENGINEERS ONLY.**

What does the Apply button do?

Clicking **Apply** applies the current settings on the URA page to the URA. For example, if the angle is set to 50° and Sample selected, the URA will move to an angle of 50° in the sample position, and set the spot size.

Corrections and baselines

What type of corrections should I use when I have a Universal Reflectance Accessory (URA) installed?

If you are using a URA, it is likely that you will be making many measurements – for example, measuring many angles of one sample. Correction spectra (**100 %T / 0A Baseline (Autozero)** and/or **0 %T / Blocked Beam Baseline**) will be measured for each angle and this could take some time depending on the number of samples/measurements specified in the sample table. We therefore recommend that you do not select **Always at task start**, as all correction spectra will be recorded before any sample spectra are recorded. If you select **Always before next measurement**, the same number of correction spectra will be recorded but the correction spectra will be recorded for a particular sample (or measurement) and then the sample run. This means that you will not have to wait as long before seeing your results.

You can save correction spectra as part of a method so that they do not have to be run each time you run the method.

NOTE: You must open a method in Edit mode.

1. Define the corrections you require.

2. Click  .
Clicking Autozero will perform the selected corrections.

3. From the File menu select **Save Method**.

4. On the Save Method dialog, select **Save Corrections**.
The corrections will be stored with the method.

NOTE: You should not save corrections with the method if you select **Always at task start** or **Always before next measurement**, as by definition, previous corrections are discarded.

When you run the task from this method:

If **As required at task start** is selected, the corrections that were saved with the method are used and only any additional corrections that are required are collected at the start of the task provided the expiry time has not elapsed.

If **As required before next measurement** is selected, the corrections that were saved with the method are used and only any additional corrections that are required are collected before each measurement.

NOTE: If you press  when you are in a task that contains corrections that were saved as part of the method, these previously saved corrections are discarded.

How do I perform an autozero within Manual Control?

We recommend that you do not perform this action unless you fully understand how the URA operates.

 Clicking  in Manual Control with a URA installed will collect a baseline using the URA in its current position.

If the URA is currently in the sample position the baseline will be collected in the sample position. You must place a reference mirror at the sample position to collect the baseline with the URA in the sample position. Unlike when you are running a method, you will not be prompted to do this.

If you wish to autozero in the baseline position you must first drive the URA to the baseline position by selecting **Baseline** and then clicking **Apply**. After performing the autozero you must then drive the URA to the sample position (by selecting **Sample** and then clicking **Apply**) before scanning a sample within Manual Control. If you fail to do this the data will be meaningless.

Small Samples

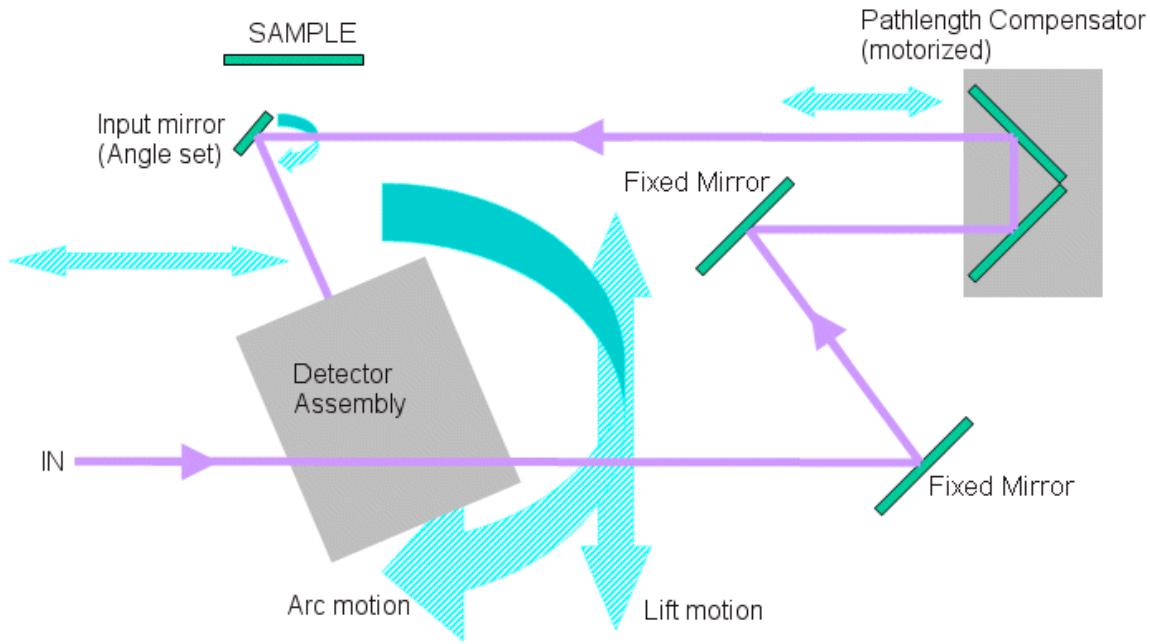
How do I align a small sample?

1. Click 
2. Enter a value of **0 nm** to get white light at sample position.
This will allow you to align a small sample.

URA Optics

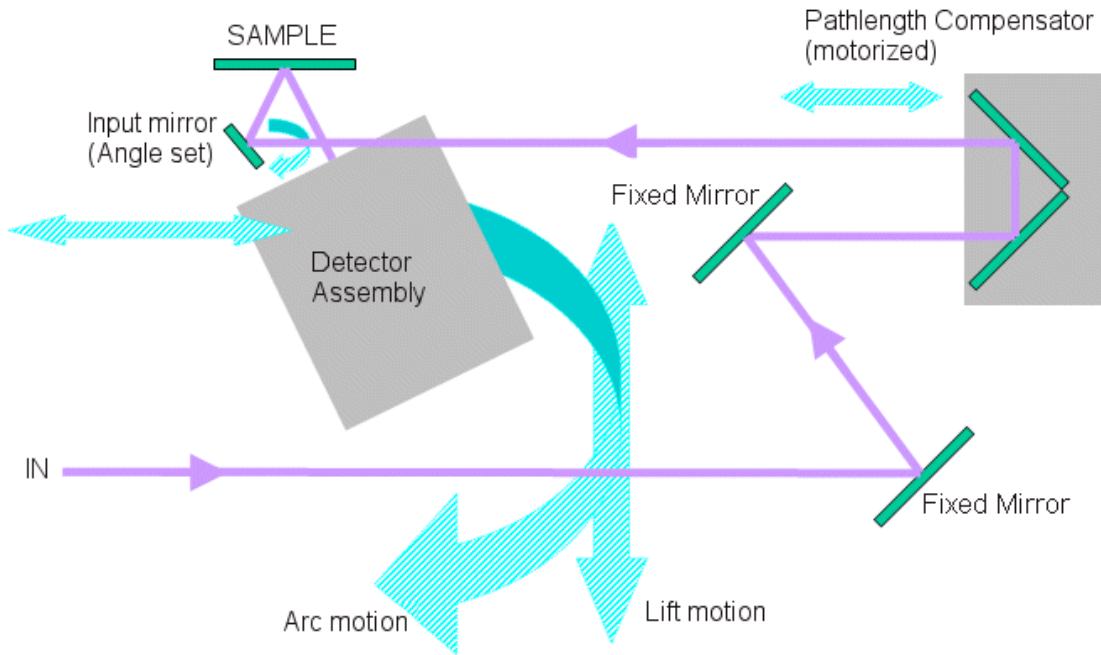
What are the positions of the optics when an angle of incidence of 8° is specified?

The schematic diagram below shows the optics in the URA positioned to collect a baseline measurement when the angle of incidence is 8°. The available movements of the pathlength compensator, input mirror and detector arc assembly are displayed using dashed arrows.



URA baseline measurement when angle of incidence is 8°

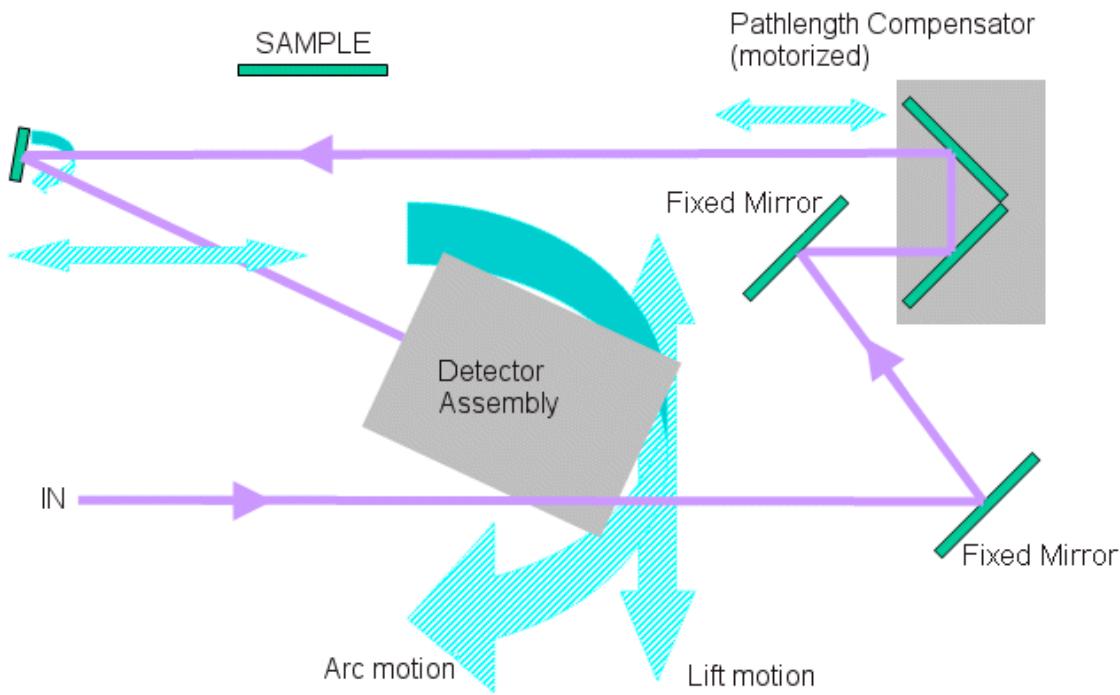
To collect the sample measurement, the Input mirror rotates to direct the beam onto the sample. The detector is raised up to the correct position to capture the light.



URA sample measurement when the angle of incidence is 8°

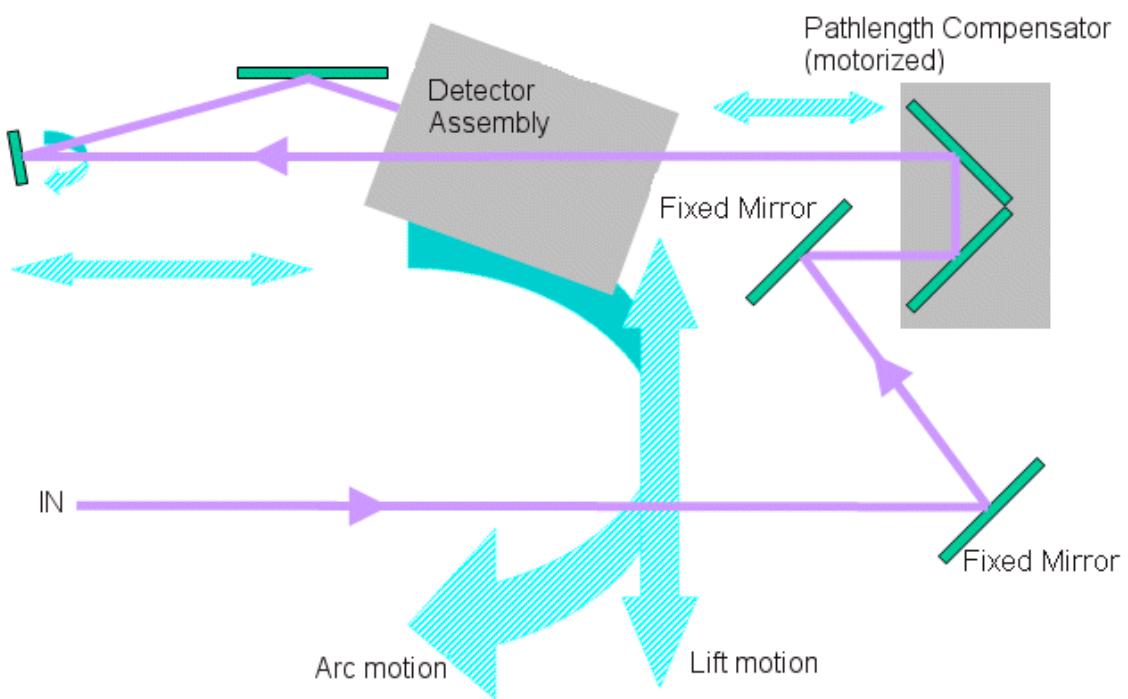
What are the positions of the optics when an angle of incidence of 65° is specified?

The schematic diagram below shows the optics in the URA positioned to collect a baseline measurement when the angle of incidence is 65°. The available movements of the pathlength compensator, input mirror and detector arc assembly are displayed using dashed arrows.



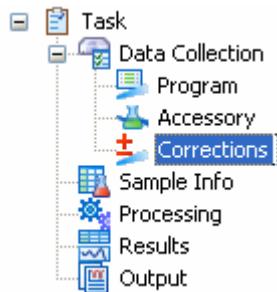
URA baseline measurement when the angle of incidence is 65°

To collect the sample measurement, the Input mirror rotates to direct the beam onto the sample. The detector is raised up to the correct position to capture the light.



URA sample measurement when the angle of incidence is 65°

Corrections



Corrections is used to specify how often a baseline must be collected, the type of baseline, reflections settings and absorptance.

NOTE: Reflection Corrections is only available when the Ordinate mode is %R.

Baseline Corrections

What types of baseline correction are available?

Always at task start – When data collection is first started for the task, corrections will always be collected for all samples before the sample measurements are made. If the task remains open and more samples are added, the task will re-use the corrections it had collected previously. If any new corrections are required, these will be collected before the sample measurements are made. However, if a task is closed and then re-opened and continued at a later date, new corrections will be collected before the new sample measurements.

Always before next measurement – Corrections for the sample will always be collected before each sample measurement. Even if previously collected corrections are still valid, these are discarded and new corrections collected.

As required at task start – When data collection is first started for the task, corrections will be collected for all samples only where there are no existing, valid corrections. Any existing, suitable corrections will be re-used. Corrections need to be saved with the method for them to be available to tasks that are run from the method. Corrections can be collected for a method using the Autozero command and then saved by selecting **Save Corrections** on the Save Method or Save As Method dialog.

As required before next measurement – Corrections for the next sample are collected prior to collecting the sample measurement only if there are no existing, suitable corrections.

Import – Corrections can be imported for use with all samples. Select Import from the drop-down list. A Browse dialog is displayed enabling you to explore and select a spectrum (or spectra). Each spectrum must cover the spectral range of the spectrum to be corrected.

If Always at task start, As required before next measurement, or As required at task start are selected, you must define when the corrections will expire.

- Select the number (1–999) and time period (hours, days, weeks) from the drop-down lists.

NOTE: For **Always at task start** the expiry time only applies while the task remains open.

What types of baseline can be collected?

1. Select 100 %T / 0A Baseline (Autozero) and / or 0 %T / Blocked Beam Baseline.

NOTE: 100 %T is 0 Abs.

2. If **0 %T / Blocked Beam Baseline** is selected, select whether to use the **internal attenuator**.

Attenuators are used to improve noise levels at high absorbance / low transmittance. The internal attenuator option will only be available if it is installed. If you use the internal attenuator, the sample beam will automatically be blocked for the correction. If not selected, you will be prompted to block the beam.

3. If required, select Do not invalidate baselines with respect to instrument settings.

NOTE: Do not invalidate baselines with respect to instrument settings and Do not invalidate attenuator corrections are not available when Always before next measurement is selected as they are not applicable.

If Do not invalidate baselines with respect to instrument settings is selected and you change any instrument settings after collecting corrections, the corrections will not be invalidated. The only exception to this is if you increase the data range or reduce the data interval. In these cases, there is insufficient data within the corrections and so they must be recollected. If this occurs, the software will automatically collect the corrections it requires.

If Do not invalidate baselines with respect to instrument settings is not selected, and you change any of the instrument settings, new baselines will be collected. However, if the range or interval is changed such that they are a subset of the previous range or interval (for example the old range was 800–300 nm, and the new range is 700–400 nm) then existing corrections will be re-sampled to the new instrument conditions and hence new corrections will not be collected.

If Always at task start or As required at task start is selected, when a change is made to the instrument settings, the corrections are collected for all pending samples before the sample measurements. If As required before next measurement is selected, the necessary corrections will be collected before each sample measurement.

How do I save baselines with a method, and how does this affect the corrections collected when I run a task?

NOTE: You must open a method in Edit mode.

To save corrections to the existing method:

1. Define the corrections you require.
2. Click  .
Clicking Autozero will perform the selected corrections.
3. From the File menu select **Save Settings**, and then from the submenu select **To Method**.
4. On the Save Method dialog, select **Save Corrections**.
The corrections will be stored with the method.

NOTE: You should not save corrections with the method if you select **Always at task start** or **Always before next measurement**, as by definition, previous corrections are discarded.

When you run the task from this method:

If **As required at task start** is selected, the corrections that were saved with the method are used and only any additional corrections that are required are collected at the start of the task provided the expiry time has not elapsed.

If **As required before next measurement** is selected, the corrections that were saved with the method are used and only any additional corrections that are required are collected before each measurement.

NOTE: If you press  when you are in a task that contains corrections that were saved as part of the method, these previously saved corrections are discarded.

To save corrections to a new method:

1. Define the corrections you require.

2. Click  .
Clicking Autozero will perform the selected corrections.

3. From the File menu select **Save Settings**, and then from the submenu select **As New Method**.

4. On the Save As Method dialog, select **Save Corrections**.

5. Select the location for the new method from the list of folders.

NOTE: The folders available in the Folder List will depend on the permissions you have.

6. Enter a **Name** and **Description** and then click **Save**.
A Save As Method dialog is displayed.

7. If you are using the Enhanced Security version of UV WinLab, enter your **User name**, **Password**, **Reason**, and **Comment**.
The fields that appear on this dialog depend on the settings previously defined by your UV WinLab Administrator.

8. Click **OK**.
The dialog closes and the settings and corrections are saved as a new method. The status of the method in the Explorer is draft.

What happens if I change any instrument settings after collecting baselines?

If **Do not invalidate baselines with respect to instrument settings** is selected and you change any instrument settings after collecting corrections, the corrections will not be invalidated. The only exception to this is if you increase the data range or reduce the data interval. In these cases, there is insufficient data within the corrections and so they must be recollected. If this occurs, the software will automatically collect the corrections required.

If **Do not invalidate baselines with respect to instrument settings** is not selected, and you change any of the instrument settings, new baselines will be collected. If **Always at task start** or **As required at task start** is selected, when a change is made to the instrument settings, the corrections are collected for all future samples before the sample measurements. If **As required before next measurement** is selected, the necessary corrections will be collected before each sample measurement.

How is 100% T performed and calculated?

For Medium performance instruments you will be prompted to remove any samples prior to the scan. The instrument will scan at 100%T.

For High performance instruments, if attenuators have been defined on the Data Collection page the sample beam attenuator is set to the value of the reference beam attenuator and then the instrument will scan at 100%T.

How is 0% T performed and calculated?

NOTE: 0%T is only available for High performance instruments.

100%T and 0%T spectra must be collected for this correction.

If attenuators are installed and **Use internal attenuator** is selected:

100%T – The sample beam attenuator is set to the value of the reference beam defined on the Data Collection page and the 100% correction is collected.

0% T – The sample beam attenuator is then set to 0% and the 0%T correction spectrum collected.

If attenuators are not installed, or **Use internal attenuator** is deselected:

100%T – You will be prompted to remove sample(s) to perform a 100%T correction.

0% T – You will then be prompted to block the sample beam. The 0%T spectrum is then collected.

$$T(\%) = \frac{T_{\text{raw}} - B_0}{B_{100} - B_0} \times 100$$

where

$T(\%)$ = value in %T units

T_{raw} = transmittance value

B_0 = baseline value at 0%T

B_{100} = baseline value at 100%T

How are 100%/0% baselines collected using internal attenuators?

If attenuators are installed and **Use internal attenuator** is selected:

100%T – The sample beam attenuator is set to the value of the reference beam defined on the Data Collection page and the 100% correction is collected.

0% T – The sample beam attenuator is then set to 0% and the 0%T correction spectrum collected.

Reflection Corrections

When are the Reflection Corrections settings available?

The Reflection settings are only available when %R is specified as the Ordinate mode in the Method settings section of the Data Collection page. You must specify one of the three Reflection settings options – **Reflectance corrected for reference (%RC)**, **Reflectance corrected for IV and VW accessories (%RA)**, or **None**.

What is Reflectance corrected for reference (%RC)?

Reflectance corrected for IV and VW accessories (%RA) allows you to collect absolute reflectance data. This is the square root of reflectance data. %RA should be used for internally reflective samples.

IV- and VW-type absolute specular reflectance accessories work by measuring two reflections from the sample, and so the detector measures the square of the true reflectance. This mode uses the square root of the measured spectrum so that the user collects the true reflectivity of the sample.

NOTE: This options should not be selected for the URA, PELA 1030, or VN-type reflectance accessories.

Reflectance corrected for reference (%RC) corrects a reflectance spectrum for dark and white values.

1. Select the **Light Spectral Reference** from the drop-down list.
The options are Spectralon, BaSO₄, or Select - Import.
2. If you select **Select - Import**, a Browse dialog is displayed and you can select the spectrum to import.
The spectrum must cover the spectral range of the spectrum to be corrected.
3. Click **Open** to import the spectrum.

NOTE: When a spectrum is imported, the file name and path is displayed in the Spectral Reference drop-down list.

4. Select the **Dark Spectral Reference** from the drop-down list.
The options are None, Spectralon, BaSO₄, or Select - Import.
5. If you select **Select-Import**, a Browse dialog is displayed and you can select the spectrum to import.
The spectrum must cover the spectral range of the spectrum to be corrected.

6. Click **Open** to import the spectrum.

NOTE: When a spectrum is imported, the file name and path is displayed in the spectral Reference drop-down list.

What is Reflectance corrected for IV and VW accessories (%RA)?

Reflectance corrected for IV and VW accessories (%RA) allows you to collect absolute reflectance data. This is the square root of reflectance data. %RA should be used for internally reflective samples.

IV- and VW-type absolute specular reflectance accessories work by measuring two reflections from the sample, and so the detector measures the square of the true reflectance. This mode uses the square root of the measured spectrum so that the user collects the true reflectivity of the sample.

NOTE: This options should not be selected for the URA, PELA 1030, or VN-type reflectance accessories.

How is the Reflection Correction calculated?

The following equation is used:

$$R_{\text{actual}} = \frac{[R_{\text{measured}} - (R_0)] \times (R_{100})}{100 - (R_0)}$$

where R_{100} is the Light Spectral Reference, and R_0 is the Dark Spectral Reference.

Attenuator Corrections

What do attenuators do?

The Front and Rear beam attenuators are used to select the attenuation in the sample and reference beam. Reference beam attenuation is used to improve noise levels at high absorbance/low transmittance. The instrument uses a single detector (photomultiplier for UV/Vis, and InGaAs or PbS for NIR). A ratio is measured between the sample and reference beam. With highly absorbing samples this means there is a ratio between a very small signal (highly absorbing sample) and a big signal (the unblocked reference beam). The instrument cycles between the two readings every 40 milliseconds and the rapidly changing light levels can cause the detector to become noisy. In addition, there is the mathematical problem of performing a ratio with a large difference between the numerator and the denominator.

When are the attenuator correction settings available?

Attenuator corrections are only collected if the Sample Beam attenuator is set to **Automatic**. The option always remains visible if attenuators are installed, but is only activated in the software when the Sample Beam attenuator is set to **Automatic**.

The options are **Measure** or **Import**.

What happens if I change any instrument settings after collecting attenuator corrections?

If **Do not invalidate attenuator corrections** is selected and you change any instrument settings after collecting corrections, the corrections will not be invalidated. The only exception to this is if you increase the data range or reduce the data interval. In these cases, there is insufficient data within the corrections and so they must be recollected. If this occurs, the software will automatically collect the corrections required.

If **Do not invalidate attenuator corrections** is not selected, and you change any of the instrument settings, new baselines will be collected. If **Always at task start** or **As required at task start** is selected, when a change is made to the instrument settings, the corrections are collected for all future samples before the sample measurements. If **As required before next measurement** is selected, the necessary corrections will be collected before each sample measurement.

Expire Corrections

Can I set when a correction will expire?

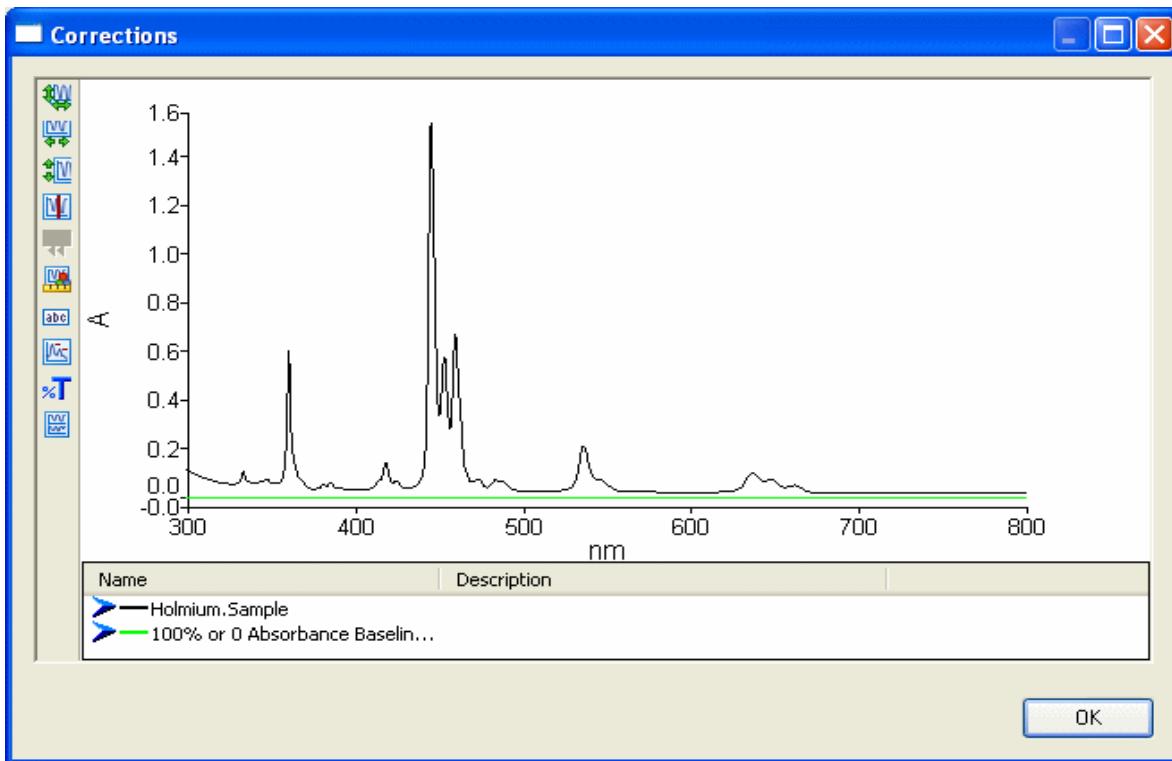
- Enter the time period after which you would like the corrections to expire in **Hours, Weeks or Days**.
If you do not wish the corrections to expire, select Never.

Viewing Corrections

How do I view the correction spectrum/spectra/data for a sample/standard?

The correction spectra can be viewed within the Sample Table, Standards Table or Results Table.

- Using the right mouse button, click on the sample/standard whose correction spectrum you wish to view, and select **Show Corrections**.
A second window is displayed showing the sample and correction spectra:



If the corrections are for a wavelength program or a wavelength quant task, a corrections table is displayed when you select **Show Corrections**:

The figure shows the 'Corrections' dialog box with a table of correction values:

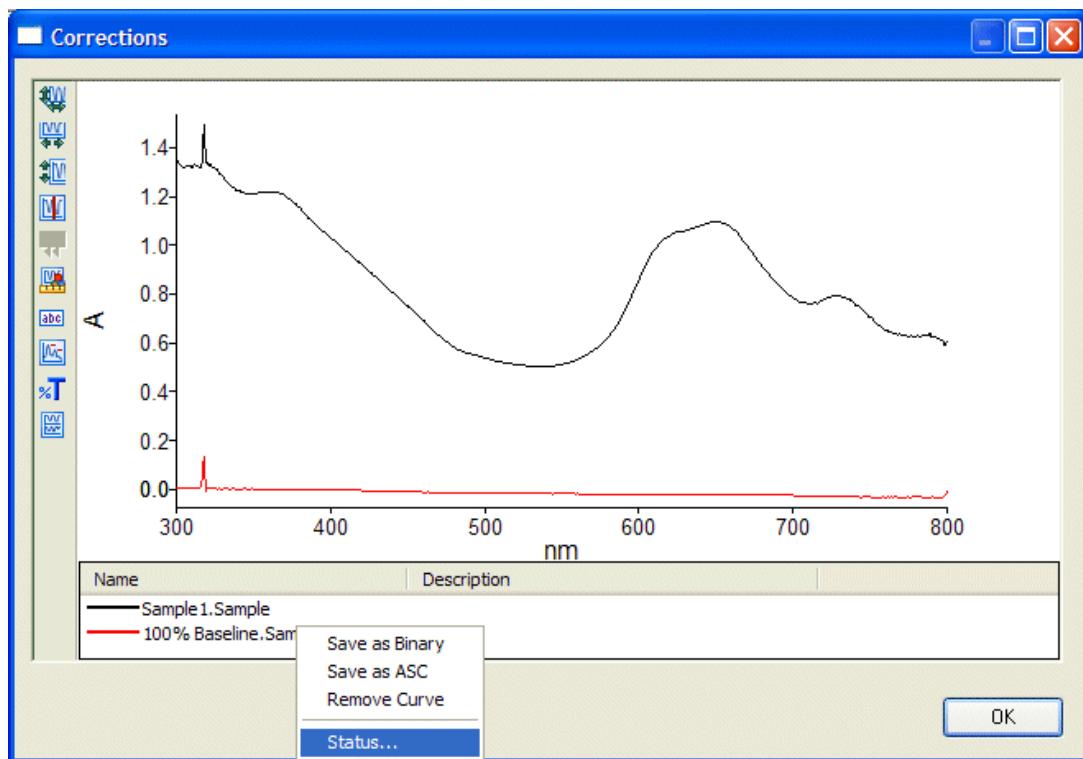
	550
Sample2.Replicate 2	1,317
100% or 0 Absorbance Baseline	0,031

At the bottom right of the dialog box is an 'OK' button.

How do I know how long a correction is valid for in a particular method?

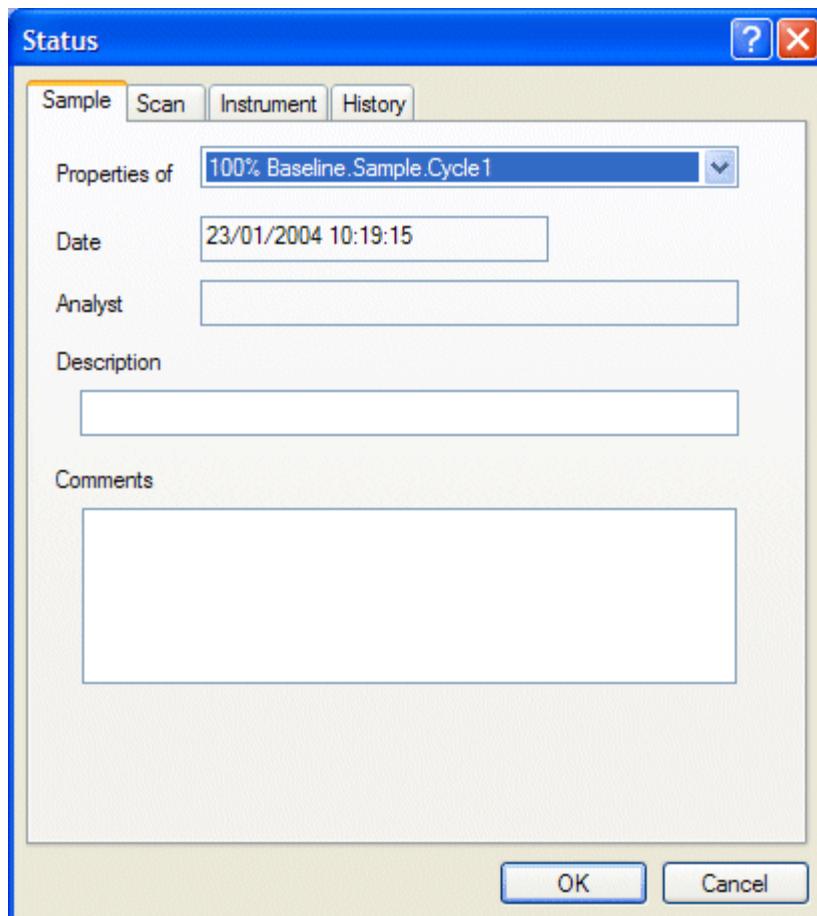
To determine the length of time remaining before a correction expires, you need to view the status of the correction spectrum and the Correction settings within the method used to create the task.

1. Open a task that has been run using the method whose correction expiry you wish to determine.
2. From the sample table, click the right mouse button on a sample and select **Show Corrections**.
The Corrections window is displayed with the sample and correction spectra.
3. Using the right mouse button, click on the name of the correction spectrum in the legend below the graph and select **Status**:



The Status dialog is displayed.

4. Select the Sample tab:



This shows the date and time that the correction was collected.

5. Close the Status dialog and then close the Correction window.
6. Select the Corrections page in the Workspace.
This will show how long the correction is valid for before it expires.
7. From the date of the task and the expiry length, you can then calculate how much longer the correction is valid for.

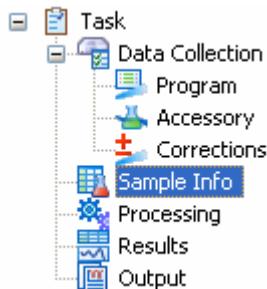
Accessories

What type of corrections should I use when I have a Universal Reflectance Accessory (URA) installed?

If you are using a URA, it is likely that you will be making many measurements – for example, measuring many angles of one sample. Correction spectra (**100 %T / 0A Baseline (Autozero)**) and / or **0 %T / Blocked Beam Baseline**) will be measured for each angle and this could take some time depending on the number of samples / measurements specified in the sample table. We therefore recommend that you do not select **Always at task start** as all correction spectra will be recorded before any sample spectra are recorded. If you select **Always before next measurement**, the same number of correction spectra will be recorded but the correction spectra will be recorded for a particular sample (or measurement) and then the sample run so you will not have to wait as long before seeing your results.

You can save correction spectra as part of a method so that they do not have to be run each time you run the method. See "How do I save baselines with a method, and how does this affect the corrections collected when I run a task?" above.

Sample Info



The Samples tab on the Sample Info page enables you to format the Sample Table with all the information needed prior to collecting the samples, such as the type of sample preparation, the Sample ID, the number and frequency of control samples, any Standards (quant methods) and the columns that will appear in the Table. The spectra are displayed on the Graphs tab.

Adding and deleting samples

How do I enter the number of samples?

- Enter the number of samples in the **Samples** field:

1	Samples
---	---------

Should I enter the number of samples before or after formatting the Sample Table?

It does not matter whether you enter the number of samples before or after formatting the Sample Table for replicates and measurements; the table will be updated accordingly. If more than one replicate is specified, the correct number of replicates are added to or removed from the table when the number of samples is changed. For example, if you specify three replicates per sample and you increase the number of samples by 2, then 6 entries are added to the table.

However, if your sample table contains columns such as Initial Volume for a dilution, entering the number of samples before or after formatting the sample table will make a difference. For example, you have 3 samples in the table and set the Initial Volume to 10 ml. If you then format the table and change the Volume to 15 ml, the samples currently in the table will not be updated with the new volume. However, any new samples added to the table will have a Volume of 15 ml. It is possible though to edit the previous Volumes in the sample table by clicking in the required cell and editing the value.

NOTE: Adding samples using **Insert** adds samples to the top of the list. Increasing the number of samples by amending the number of samples in the field to the left of the Format Sample Table button adds samples to the bottom of the list.

NOTE: The number of samples is determined by the Analyst after a Method has been opened, rather than when the Developer is creating the Method.

How do I insert a sample at a particular point in the Sample Table?

After you have defined the number of samples, it is possible to add samples if you have forgotten to do so.

1. Click in the Sample row where you want to add the sample.

NOTE: The new row is added above the selected row.

2. Click **Insert**.
3. The new sample is added above the selected row.
It is possible to then change the sample Type if you want the sample to be a control or blank rather than sample.

How do I delete a sample from the Sample Table?

If data has not been collected:

4. Click in any cell of the row of the sample that you wish to delete.
A dotted line appears around the cell to show that it is selected:

	Sample ID	Description	Type
1	Sample1	Supplier ABC12	Measurement 1
2	Sample1	Supplier ABC12	Measurement 2
3	Sample2	Supplier ABC12	Measurement 1
4	Sample2	Supplier ABC12	Measurement 2

5. Click **Delete**.

The sample is deleted from the Sample Table. When a replicate is deleted, the remaining replicates are renamed and ordered accordingly.

NOTE: If replicates are defined, Replicate 1 must be selected to delete the entire sample.
Otherwise, only the selected replicate is deleted.

If data has not been collected, the sample ID of a deleted sample can be re-used.

If data has been collected:

If you are using the Standard Security version of UV WinLab, you are prompted to confirm the deletion. The sample is then deleted.

If you are using the Enhanced Security version of UV WinLab, you are prompted to confirm the deletion and provide a reason to be recorded with the sample. This information together with the User name and the Date/Time is recorded with the sample data. The row is grayed. The behavior is the same for replicates.

What sample types are available?

The sample types: **Sample**, **Blank**, **Control** and **Standard** are available.

If Replicates are used, **Sample** is replaced by **Replicate 1** to Replicate n (where n is the number of replicates).

If Measurements are used, **Sample** is replaced by **Measurement 1** to Measurement n (where n is the number of measurements).

How do I select a sample type?

1. Click in the **Type** cell of the sample whose **Type** you wish to specify.
A drop down arrow  appears.
2. Click on  to display a drop-down box with the list of available Types.
The options are Blank, Control, Sample or Standard (available for quant methods only)..
3. Select the sample **Type**.
The cell is updated with the selected sample Type.

If Replicates are used, **Sample** is replaced by **Replicate**:

2 Samples			
	Sample ID	Description	Type
1	Sample1		Replicate 1
2	Sample1		Replicate 2
3	Sample2		Replicate 1
4	Sample2		Replicate 2

If Measurements are used, **Sample** is replaced by **Measurement**:

	Sample ID	Description	Type
1	Sample1		Measurement 1
2	Sample1		Measurement 2
3	Sample2		Measurement 1
4	Sample2		Measurement 2

How do I import a sample?

1. From the File menu, select **Open**.
The Open dialog is displayed.

NOTE: The **Open** menu item is only available if the Sample Info page is selected.

2. In the left panel, click .
3. Using the **Folders** and **Name** sections, navigate to the spectrum you want to import into UV WinLab.
When you select a spectrum it is displayed in the lower panel.
4. Click **Open**.
The spectrum is imported into the method and can be manipulated in the same way as collected spectra.

How do I import a Sample Table?

It is possible to import sample tables created in *.csv, *.prn and *.txt files. The imported table does not need to contain all the columns that are in the Sample Table, but if it contains more columns than the UV WinLab Sample Table these will not be imported. Also the order of the columns in the imported file must match that in the UV WinLab Sample Table.

1. From the File menu, select **Import Table**.

The Open dialog is displayed.

NOTE: The **Import Table** menu item is only available if the Sample Info page is selected.

2. Select the file type you would like to import.

The options are *.csv, *.txt and *.prn.

NOTE: When creating your file for import you will need to ensure that the appropriate list separator is used. The list separator allowed will depend on what is specified in the **Regional and Language Options** set up via the **Control Panel** in Windows.

3. Select the file you would like to import.

4. Click **Open**.

The samples are imported into the Sample Table.

If the file includes information on the sample type, this will be automatically recognized in the UV WinLab Sample Table.

NOTE: When you have imported a Sample Table you can edit it the columns as described on this page.

How do I export a Sample Table?

It is possible to export sample tables as *.csv, *.prn and *.txt files.

1. From the File menu, select **Export Table**.

The Save As dialog is displayed.

NOTE: The **Export Table** menu item is only available if the Sample Info page is selected.

2. Select the file type you would like to export the sample table as.

The options are *.csv, *.txt and *.prn.

NOTE: The list separator used when creating the text files will be that specified in the **Regional and Language Options** set up via the **Control Panel** in Windows.

3. Click **Save**.

How do I add a comment to a sample?

1. Select the sample to which you wish to add a comment.

2. Right-click and from the menu select **Add Comment**.

The Add Comment dialog is displayed.

3. Enter the text and then click **OK**.

NOTE: A comment can only be added to a sample via the **Add Comment** menu item that is displayed when you right-click on a sample.

The comment can be viewed in the Sample Event Log – right-click on the sample and select **View Sample Event Log**. The Sample Event Log records the Event, Date and Time, User, and Reason/Comment.

It can also be viewed via the results of a Query. Select the result from the table and then select **Signatures and Comments** from the Folder List. Any comments added to the sample are listed.

How does the Fill Down button work?

If you want several cells to contain the same entry, for example, if the Description is the same, Fill Down allows you to enter the text once and then copy it into the other cells.

1. Enter the text in the top cell of the range of cells that will contain the same information:

Description
Supplier ABC12

2. Select all the cells to contain the information – including the cell you have just entered the information in:

Description
Supplier ABC12

Clicking on the column header will select all the cells in the column.

3. Click Fill Down.

The information is copied into all the selected cells:

Description
Supplier ABC12

NOTE: **Fill Down** is not applicable to the Sample ID and Standard ID columns.

What columns will appear in the Sample Table?

By default the first column will be **Sample ID**. The other columns depend on the formatting of the table and what accessories are installed. See the questions on this page relating to each accessory and how the sample table is affected.

How do I edit a sample description?

1. Click in the **Description** cell that you wish to edit.

A dotted line appears around the cell to show that it is selected:

	Sample ID	Description	Type
1	Sample1	Supplier ABC12	Measurement 1
2	Sample1	Supplier ABC12	Measurement 2
3	Sample2	Supplier ABC12	Measurement 1
4	Sample2	Supplier ABC12	Measurement 2

2. Enter a description of the sample.

Click outside of the cell when you have finished.

What menu items are available when I right mouse-click on a row in the Sample Table?

The following menu items are available:

Copy	Copies information from the selected cell to the clipboard.
Paste	Pastes information from the clipboard to the selected cell.
View Sample Event Log	Displays the Sample Event Log.
Add Comment	Allows you to add a comment to a sample.
Show Corrections	Displays the corrected spectra collected for the sample. These are specified on the Corrections dialog.

What is the Sample Event Log?

In the Enhanced Security version of UV WinLab:

The Sample Event Log records all events connected with the sample. For example, if a description is added after the sample has been run, this event is recorded in the Event Log. The original value and new value are recorded.

The Sample Event Log records the event, time, user, and reason/comment.

NOTE: If a field has been changed from being empty to containing a value, empty quotes "" are used to show that the field was initially empty. For example, if the description field was empty and then changed to read Batch 1, the Event Log would record: Description changed from "" to "Batch 1".

In the Standard version of UV WinLab:

The Sample Event Log only records any comments associated with the sample.

Formatting the Sample Table

How do I format the Sample Table?

Formatting the Sample Table allows you to define the Columns and layout of the Sample Table for a particular Method. In the Enhanced Security version of UV WinLab, Analysts will not be able to change which columns are displayed or which columns are mandatory to complete.

1. Click Format Sample Table.
The Table Builder dialog is displayed.
2. Edit the details as required.
Each of the four tabs are explained in turn below.

Preparation

NOTE: The types of Preparation available depend on the method selected.

Select the type of sample preparation to be used. Columns will be added to the table depending on the type of sample preparation selected:

None	No change is made to the Sample Table.
Factor	A column called Factor is added to the Sample Table.
Dilution	Adds columns called Factor , Initial Volume (ml) and Final Volume (ml) to the Sample Table.
Dissolved solid	Adds columns called Factor , Weight (mg) and Volume (ml) to the Sample Table.
Dissolved solid and dilution	Adds columns called Factor, Weight (mg), Volume (ml), Initial Volume (ml) and Final Volume (ml) to the Sample Table.
Concentration	A column called Concentration is added to the Sample Table. If this column is not automatically added, select the Columns tab of the Table Builder dialog and select the Concentration column. When the dialog is closed, the column is added to the table. NOTE: Concentration is only available for Quant Methods.
Substrate concentration	Adds columns called Starting Volume (ml) and Added Volume (ml) to the Sample Table. The options on the Preparation tab are different when Substrate Concentration is selected. NOTE: Substrate concentration is only available for Timedrive methods.

Set preparation volume (ml)	This is only available when Dissolved solid or Dissolved solid and dilution is selected. Enter the volume used (ml). This value will appear in the Volume (ml) column of the table.
Set volume taken for dilution (ml)	This is only available when Dilution or Dissolved solid and dilution is selected. Enter the volume used (ml). This value will appear in the Initial Volume (ml) column of the table.
Set final volume (ml)	This is only available when Dilution or Dissolved solid and dilution is selected. Enter the volume used (ml). This value will appear in the Final Volume (ml) column of the table.

NOTE: When a value is entered for **Set preparation volume (ml)**, **Set volume taken for dilution (ml)**, or **Set final volume (ml)**, it applies to all samples in the table. However, you can edit these values. Alternatively, you can de-select these options and manually enter any values in the Sample Table for each sample. Values set by the Developer cannot be edited by the Analyst within the Enhanced Security version of UV WinLab.

Set starting volume (ml)	This is only available when Substrate concentration is selected. Enter the starting volume used (ml). This value will appear in the Starting Volume (ml) column of the table.
Set added volume (ml)	This is only available when Substrate concentration is selected. Enter the added volume. This value will appear in the Added Volume (ml) column of the table.

The volume information does not have to be entered in the Sample Table. However, if **Table must be completed before run** is selected, this information must be entered by the Analyst before running the task.

Design

Select whether you wish to include replicates and control samples. Rows will be added to the table depending on the options selected.

Replicates per sample	<p>Enter the number of replicates per sample. The default is one. If more than one replicate is entered, each sample will have the appropriate number of replicates added. The cell in the Type column then contains a drop-down list of Blank, Control and Replicate 1.</p> <p>If Blank or Control are selected the associate replicates are removed from the table.</p>
Sequence by sample	<p>Select Sequence by sample to run the samples in sample order rather than replicate order. For example, if you had 2 samples (S1 and S2) and each sample had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the samples would be run in the order: S1R1, S1R2, S2R1, S2R2.</p> <p>NOTE: The number of replicates must be between 1 and 9.</p>
Sequence by replicate	<p>Select Sequence by replicate to run the samples in replicate order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the samples would be run in the order: S1R1, S2R1, S1R2, S2R2.</p> <p>NOTE: The number of replicates must be between 1 and 9.</p>
Sequence by measurement	<p>Select Sequence by measurement to run the samples in measurement order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S2M1, S1M2, S2M2.</p> <p>NOTE: The number of replicates must be between 1 and 200.</p>
Control sample at start of run	Inserts a control sample at the start of the run (that is, the first entry in the Sample Table)
Control sample every X samples	Inserts a control sample after every Xth sample. Enter a value for X. The default is 1.
Control sample at end of run	Inserts a control sample at the end of the run (that is, the last entry in the Sample Table).

NOTE: The order of the samples in the table (which is determined by the options on the Design tab) is the order in which the samples will be run.

NOTE: It is possible to select **Control sample at start of run**, **Control sample every X samples**, and **Control sample at end of run** within one Sample Table. This will allow the analyst to always run a Control at the beginning, Xth position, and at the end of a run.

Columns

Select the columns to appear in the Sample Table and the order of these columns.

Add	Displays the Column type dialog which enables you to add a custom column to the table. See Custom Columns.
Move Up	Moves the selected column one space up. This means the column is moved one space to the left in the Sample Table. NOTE: Move Up is only available if a column is selected (a dotted line appears around the column name when it is selected).
Move Down	Moves the selected column one space down. This means the column is moved one space to the right in the Sample Table. NOTE: Move Down is only available if a column is selected (a dotted line appears around the column name when it is selected).
Format	Displays a Format Column dialog depending upon the type of column selected. See Formatting Columns.

ID

User must enter	When User must enter is selected, the user must fill in the Sample ID column of the Sample Table for each sample. NOTE: If replicates are included, when you enter the name for the first replicate the name of the other replicates for the sample are automatically filled in.
Use this format	When Use this format is selected, the Sample ID column of the Sample Table is automatically filled in by incrementing the prefix which you should enter in the field below. For example, if you enter the prefix Trial , the samples will be numbered Trial1 to Trialn (where n is the number of samples). The numbering is remembered across all tasks created by this method.

3. When you have selected all the required settings, click **OK**.

The Table Builder dialog closes. Any new samples added to the Sample Table are populated using the chosen settings.

NOTE: When reprocessing a task, only the Columns tab of the Table Builder dialog is available.

Why can't I format the Sample Table?

The Sample Table can only be formatted using the **Format Sample Table** button.

The **Format Sample Table** button is not available in the Enhanced Security version of UV WinLab if the Method has been locked, if the samples have been run, or if you do not have the correct permission. However, information such as the sample description can still be edited.

NOTE: Permissions are defined by the UV WinLab Administrator. Please contact them for further information about your permissions.

How do I select which columns will appear in the Sample Table?

1. Select the Columns tab on the Table Builder dialog.
 2. Select the check box next to the name of a column that you want to appear in the table.
- A tick indicates that the column will appear in the Sample Table.

NOTE: The column to the right of the sample table will automatically expand to fill the pane.

How do I reorder the columns in the Sample Table?

The order of the columns listed on the Columns tab is the order in which the columns will appear in the Sample Table.

1. Click on a column name to select it.
A dotted line appears around the name of the column to show that it is selected.
2. Click **Move Up** or **Move Down** to move the selected column up or down the list.
The columns in the Sample Table are updated accordingly.
Move Up moves the column to the left in the table.
Move Down moves the column to the right in the table.

Measurements and replicates

How do I add replicates to the Sample Table?

1. Enter the number of **Samples** to be run.
2. Click **Format Sample Table**.
The Table Builder dialog is displayed.
3. Select the Design tab.
4. Select **Replicates** from the drop-down list.
5. Enter the number of **replicates per sample**.

6. Select whether to Sequence by sample or Sequence by replicate.

Select Sequence by sample to run the samples in sample order rather than replicate order. For example, if you had 2 samples (S1 and S2) and each sample had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the samples would be run in the order: S1R1, S1R2, S2R1, S2R2.

Select Sequence by replicate to run the samples in replicate order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the samples would be run in the order: S1R1, S2R1, S1R2, S2R2.

7. Click **OK**.

The replicates are added to the sample table.

NOTE: The number of replicates must be between 1 and 9.

If I have replicate samples, can I choose the order in which the samples are run?

The Design tab of the Table Builder dialog allows you to specify the order in which the samples should be run when you have replicates. You can either sequence by sample or sequence by replicate.

Select **Sequence by sample** to run the samples in sample order rather than replicate order. For example, if you had 2 samples (S1 and S2) and each sample had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the samples would be run in the order: S1R1, S1R2, S2R1, S2R2.

Select **Sequence by replicate** to run the samples in replicate order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the samples would be run in the order: S1R1, S2R1, S1R2, S2R2.

Samples are run in the order they appear in the Sample Table.

How do I add measurements to the Sample Table?

1. Enter the number of **Samples** to be run.
2. Click Format Sample Table.
The Table Builder dialog is displayed.
3. Select the Design tab.
4. Select **Measurements** from the drop-down list.
5. Enter the number of measurements per sample.

6. Select whether to Sequence by sample or Sequence by measurement.

Select Sequence by sample to run the samples in sample order rather than measurement order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S1M2, S2M1, S2M2.

Select Sequence by replicate to run the samples in replicate order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S2M1, S1M2, S2M2.

7. Click **OK**.

The measurements are added to the sample table.

NOTE: A maximum of 200 measurements can be added for each sample.

If I have measurement samples, can I choose the order in which the samples are run?

The Design tab of the Table Builder dialog allows you to specify the order in which the samples should be run when you have replicates. You can either sequence by sample or sequence by measurement.

Select **Sequence by sample** to run the samples in sample order rather than measurement order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S1M2, S2M1, S2M2.

Select **Sequence by measurement** to run the samples in measurement order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S2M1, S1M2, S2M2.

Samples are run in the order they appear in the Sample Table.

When would I use Replicates and when would I use Measurements?

Replicates should be used when you have identical samples. For example if you have 3 solutions of a sample and they are all the same concentration, you would enter 1 sample in the number of samples and 3 in the number of replicates. You cannot use varying concentrations with replicates of a sample. If concentration is specified in the sample table, all replicates of a sample must have the same concentration.

Measurements should be used when you have similar samples. For example, if you have 3 solutions that are of varying concentrations having been made up by diluting 1 original solution. In this case you would enter 1 in the number of samples and 3 in the number of measurements. If concentration is specified in the sample table you can define the concentration of each measurement.

Can I change Replicate 1 to a Blank or Control?

Yes.

1. Click in the **Type** field.

An arrow  appears.

2. Click on the arrow and then select **Blank** or **Control** from the drop-down list.
Replicate 1 is changed and all other replicates for the sample are deleted.

NOTE: Only Replicate 1 can be changed. The **Type** field is not selectable for the other replicates.

Control samples

When would I use Control samples?

There are three uses for Control samples:

- The simplest use is simply to flag the sample as a control so you can manually check that the results are correct.
- Or, you can set up a conditional response using Equation builder so that calculations on Control samples are flagged if they fall outside set limits. For example, create an equation of $\text{Area}[\text{Control}, 260, 300]$, then on the Conditional tab select that equation and set the criteria for a failure and decide what happens in the case of that failure. See Equations and Conditional Formatting for more information.

Or, in Quant when you select Control as the sample type you can then enter the expected concentration in the Sample Table in Sample Info and then on the Beer's Law Quant page you set the Control samples tolerance and any samples that give a calculated results more than the tolerance limit away from the expected concentration are flagged up.

How do I include Control samples?

Control samples can be included at the start of the run, after every X samples, and/or at the end of the run.

1. Click Format Sample Table.
2. Select the Design tab of the Sample Table Builder dialog.
3. Select where you want your control samples within your run – Control sample at start of run, Control sample every X samples, and/or Control sample at end of run.
4. If you select **Control sample every X samples**, enter the number of samples between each control in the field below:

The screenshot shows a dialog box with a light beige background. At the top left is a checked checkbox labeled "Control sample every X samples". To its right is a small input field containing the number "1". Below this section is a horizontal line with a vertical line extending downwards from the center, indicating a continuation of the table structure.

NOTE: It is possible to select more than one or all of the options.

5. Click **OK**.
- The Control samples are added to the table as specified. The Sample ID is Control and the Type is also Control.

NOTE: You can also set the Sample type for any sample in the table to **Control** using the drop-down list.

Data Collection Settings

How do I specify the values for Wavelength if 'Sample Table' is specified on the Data Collection page of a Polarization scan?

NOTE: This is only applicable to high-performance instruments.

If **Sample Table** is specified for the Wavelength in the Method Settings section on the Data Collection page, then a Wavelength column is added to the Sample Table.

By default, the wavelength specified on the Data Collection page is listed for each sample.

A different value can be specified for each sample.

- Click in the wavelength field you wish to amend and enter the new wavelength.

NOTE: If the wavelength is changed on the Data Collection page and then more samples are added to the sample table, the default wavelength for the added samples is the new wavelength that was specified. Only samples added to the Sample Table after this change will display the new value.

How do I specify the values for the Common Beam Mask (CBM) if 'Sample Table' is specified on the Data Collection page?

NOTE: This is only applicable to high-performance instruments.

If **Sample Table** is specified for the Common Beam Mask (CBM) on the Data Collection page, then a CBM% column is added to the Sample Table.

CBM %
100
100
100

By default the value is set to the value specified on the Data Collection page.

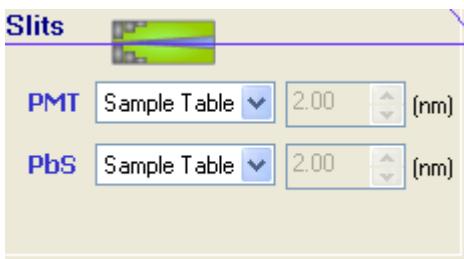
A different value can be specified for each sample.

- Click in the required field and edit the value.

If the field is left empty, the default value on the instrument's page will be used.

NOTE: If **Fixed** is re-selected on the Data Collection page, the CBM value altered, and then **Sample Table** re-selected, only samples added to the Sample Table after this change will display the new CBM value.

How do I specify the values for the UV/Vis slits if 'Sample Table' is specified on the Data Collection page?



If **Sample Table** is specified for the **UV/Vis Slit** on the Data Collection page, **UV/Vis Mode** and **UV/Vis Slit** columns are added to the Sample Table.

A different **Mode** and **Slit** can be specified for each sample.

UV/Vis Mode	UV/Vis Slit
Fixed	2.00
Fixed	2.00
Fixed	2.00

1. Select a sample and click in the **UV/Vis Mode** field.
A drop-down arrow appears .
2. Click  to display a drop-down list, and select **Fixed** or **Program**.

Fixed – Enter the required slit width in the **UV/Vis** field. Different slit widths can be specified for each sample that has a **Fixed UV/Vis Mode**. The default value is the value on the Instruments page.

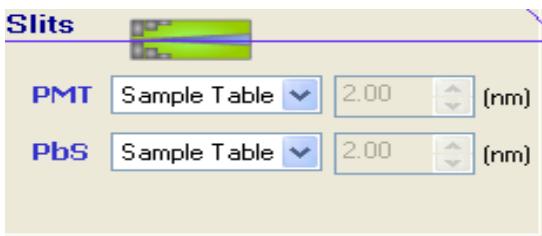
If the **UV/Vis Slit** is left blank, the default value on the Data Collection page will be used.

NOTE: These values will override the value displayed on the Data Collection page.

Program – If **Program** is selected, the software will use the pre-defined program from the Program page, and the **UV/Vis slit** column is set to **<None>**.

How do I specify the values for the NIR slits if 'Sample Table' is specified on the Data Collection page?

NOTE: This is only available for the Lambda 750, 950 and 1050.



If **Sample Table** is specified for the **NIR Slit** on the Data Collection page, **NIR Mode** and **NIR Slit** columns are added to the Sample Table.

NOTE: If you have a Three Detector Module installed, and selected on the Accessory page, then there will be two NIR slit options, one for the InGaAs detector (**NIR InGaAs Slit**) and one for the PbS detector (**NIR Slit**). If Sample Table is specified for the NIR InGaAs slit on the Data Collection page, **NIR InGaAs Mode** and **NIR InGaAs Slit** columns are added to the table.

A different **Mode** and **Slit** can be specified for each sample.

NIR Mode	NIR Slit
Fixed	2.00

1. Select a sample and click in the **UV/Vis Mode** field.
A drop-down arrow appears ▾.
2. Click ▾ to display a drop-down list, and select **Fixed** or **Program**.

Fixed – Enter the required slit width in the **UV/Vis** field. Different slit widths can be specified for each sample that has a **Fixed UV/Vis Mode**. The default value is the value on the Instruments page.

If the **NIR Slit** is left blank, the default value on the Data Collection page will be used.

NOTE: These values will override the value displayed on the Data Collection page.

Program – If **Program** is selected, the software will use the pre-defined program from the Program page, and the **UV/Vis slit** column is set to <None>.

Servo – If Servo is selected, the system will monitor the reference beam energy and adjust the slits oversaturation of the detectors. The NIR Slit column is set to <None>.

NOTE: These values will override the value displayed on the Data Collection page.

If no mode is selected the system will use the default from the Data Collection page.

How do I specify the setting for the Common Beam Depolarizer (CBD) in the Sample Table?

If the Common Beam Depolarizer is installed, a column called **CBD** is added to the sample table. This means that, for each sample, you can specify whether to have the polarization settings on or off.

CBD
On

1. Select a sample and click in the **CBD** field.
A drop-down arrow appears ▾.
2. Click ▾ to display a drop-down list, and select **On** or **Off**.

How do I specify the values for the front and/or rear beam attenuators if 'Sample Table' is specified on the Data Collection page?

If **Sample Table** is specified on the Data Collection page for the front beam attenuator, a column called **SBAtt** (sample beam attenuator) is added to the sample table.

If **Sample Table** is specified on the Data Collection page for the rear beam attenuator, a column called **RBAtt** (rear beam attenuator) is added to the sample table.

1. Select a sample and click in the **SBAtt** or **RBAtt** field.
A drop-down arrow appears .
2. Click  to display a drop-down list, and select **0%, 0.1% (L1050 only), 1%, 10%, 100%** or **Automatic**.

NOTE: **Automatic** is only available for the sample beam.

0%, 0.1%, 1%, 10%, 100% correspond to the amount of energy passing through the attenuator. 100% means that the beam is free.

What is displayed in the Sample Table when a polarizer/depolarizer is installed?

When a polarizer/depolarizer is installed, a Pol. Angle column is added to the sample table. The polarization angle can be defined for each sample.

Pol. Angle – Enter the polarization angle to be used for the sample.

When a Universal Reflectance Accessory (URA) is also installed, a Polarizer Type column is also added to the sample table.

Pol. Angle	Polarizer Type
0.00	None
0.00	None
0.00	None

The Polarizer Type is used to alter the effective pathlength to allow for the polarizer / depolarizer installed.

- Select the type from the drop-down list – **Thick, Thin, None, 1 mm, 2 mm, 5 mm, 10 mm, 20 mm, or 30 mm**.

Accessory settings

What is displayed in the Sample Table when a cell changer is installed in a Medium performance instrument?

Two columns are added to the sample table – **Cell** and **Carousel**. **Carousel** indicates the carousel being used (this will always read **Front** for a Cell Changer installed in a Medium-performance spectrometer), and **Cell** indicates the position within the carousel.

Cell	Carousel
2	Front

1. Click in the **Cell** column for a sample.
An arrow  appears.
2. Click on the arrow to display a drop-down list.
The list contains all the sample positions selected on the Cell Changer Accessory page.
3. Select the cell position for the sample.
The sample cell position can be specified for more than one sample. The cell changer will re-index to the position and prompt for the sample.

NOTE: If the available cell positions are updated on the Accessory page after the sample table has been setup, the updated available/not available positions will only apply to samples subsequently added to the sample table.

What is displayed in the Sample Table when a cell changer is installed in a High performance instrument?

Two columns are added to the sample table – **Cell** and **Carousel**. **Carousel** indicates the carousel being used (front or rear), and **Cell** indicates the position within the carousel. How these columns are filled in depends on the type of reference(s) specified on the Cell Changer Accessory page.

Cell	Carousel
2	Front

NOTE: If the available cell positions are updated on the Accessory page after the sample table has been setup, the updated available / not available positions will only apply to samples subsequently added to the sample table.

If **Single reference in position** is specified, and the reference beam is specified as the rear beam, the Carousel column is defined as **Front** for all samples and cannot be edited.

1. Click in the **Cell** column for a sample.
An arrow  appears.
2. Click on the arrow to display a drop-down list.
The list contains all the sample positions selected on the Cell Changer Accessory page.
3. Select the cell position for the sample.
The sample cell position can be specified for more than one sample. The cell changer will re-index to the position and prompt for the sample.

If Single reference in front beam and single reference in rear beam is specified, the Carousel can be Front or Rear.

1. Click in the **Cell** column for a sample.

An arrow  appears.

2. Click on the arrow to display a drop-down list.

The list contains all the sample positions selected on the Cell Changer Accessory page.

3. Select the cell position for the sample.

The sample cell position can be specified for more than one sample. The cell changer will re-index to the position and prompt for the sample.

4. Click in the **Carousel** column.

An arrow  appears.

5. Click on the arrow to display a drop-down list of **Front** and **Rear**.

Select which carousel the sample is in.

If **Matched pairing of samples to reference** is selected, and the reference beam is specified as the rear beam, the Carousel column is defined as **Front** for all samples and cannot be edited.

1. Click in the **Cell** column for a sample.

An arrow  appears.

2. Click on the arrow to display a drop-down list.

The list contains all the sample positions selected on the Cell Changer Accessory page.

3. Select the cell position for the sample.

The sample cell position can be specified for more than one sample. The cell changer will re-index to the position and prompt for the sample.

What is displayed in the Sample Table if Auto is selected as the Cell Changer Type?

Two columns are added to the table – **Cell** and **Carousel** (this applies to Medium- and High-performance instruments). Both are populated as **Auto** for every sample and this cannot be edited. At run time, the software will detect the type of cell changer and populate the sample table. The Cell positions can then be changed.

Cell	Carousel
Auto	Auto

What columns are added to the Sample Table when a sipper is installed?

Three columns are added to the sample table – **Fill Time (secs) [Sipper]**, **Flush/Return Time (secs) [Sipper]**, and **Delay Time (secs) [Sipper]**. The values in each of these columns are specified on a per sample basis. This means that you can use different settings for every sample if you require.

Fill Time (secs) [Sipper]	Flush/Return Time (secs) [Sipper]	Delay Time (secs) [Sipper]
Default	Default	Default

Flush/Return column means the sipper will flush OR return depending on the option selected on the Sipper Accessory page.

1. Click in a field (Fill Time, Flush/Return Time or Delay Time).
An arrow  appears.
2. You can type directly into the field without first selecting an item from the drop-down list. You can enter any numeric value or the word **Default**.
Default will use the value specified on the Sipper Accessory page.

OR

Click on the arrow to display a drop-down list.

The list contains Default and a number (the number is the value currently entered on the Sipper accessory page (the default value)).

3. If you wish to use the Default value specified on the Sipper Accessory page, select **Default**.

OR

If you wish to enter your own value in the field, select the only value currently listed in the drop-down list (this will subsequently be edited).

When the value is entered in the field, it is highlighted to show that it can be edited.

4. Enter the required value.
This will override the setting on the sipper accessory page.
5. Repeat for Fill Time, Flush/Return Time and Delay Time.
6. Repeat for all samples in the sample table.

What columns are added to the Sample Table when a Peltier and/or temperature probe are installed?

One column – **Target Peltier Temp** is added to the Sample Table when a Peltier is installed. The **Target Peltier Temp** column is the requested temperature, and this can be set on a per sample basis.

Target Peltier Temp. °C	Actual Peltier Temp. °C
Default	

1. Click in the required field.

A drop-down arrow  appears.

Target Peltier Temp. °C
Default
Default
25

2. Select **Default** or **25**.

Default will use the value specified on the Peltier Accessory page.

NOTE: Even if you want to specify another temperature, you MUST select one of these options from the drop-down list first.

3. If you wish to enter another value, click in the field and edit the value.

The minimum is 15 °C and the maximum is 45 °C.

The following read-only columns are added to the Results Table.

Actual Peltier Temp. °C	External Probe Temp. °C	Average Peltier Cycles Temp. °C	Average Ext. Probe Cycles Temp. °C

Actual Peltier Temp – displays the Peltier temperature for the measurement.

Average Peltier Cycles Temp – displays the average Peltier temperature for the sample over the sample measurement cycles. This column is present only when using more than one measurement cycle.

Two extra columns are added when a temperature probe is installed:

External Probe Temp – displays the external probe temperature for the measurement.

Average Ext. Probe Cycles Temp – displays the average external probe temperature over the sample measurement cycles. This column is present only when using more than one measurement cycle.

What columns are added to the Sample Table when an autosampler is installed?

Six columns are added to the sample table – Position [Autosampler], Probe Depth (mm) [Autosampler], Return Position [Autosampler], Fill Time (s) [Autosampler], Flush / Return Time(s) [Autosampler], and Delay Time(s) [Autosampler]. The values in each of these columns are specified on a per sample basis. This means that you can use different settings for every sample if you require.

	Position [Autosampler]	Probe Depth (mm) [Autosampler]	Return Position [Autosampler]	Fill Time (s) [Autosampler]	Flush / Return Time (s) [Autosampler]	Delay Time (s) [Autosampler]
1	2	Default	Rinse Port	Default	Default	Default

Flush / Return column means the autosampler will flush OR return depending on the option selected on the Autosampler Accessory page.

Position numbering starts at 1 as this is the first sample position. Position 0 is the rinse port and cannot be used in the sample table.

When the number of samples is specified, the Position column is automatically populated sequentially. The software will prompt for new trays if required when the samples are run.

NOTE: The sample tray selected on the Autosampler Accessory page cannot be altered once the sample table has been completed.

1. Click in a field (Position, Probe Depth, Return Position, Fill Time, Flush/Return Time, or Delay Time).
An arrow  appears.
2. You can type directly into the field without first selecting an item from the drop-down list. You can enter any numeric value or the word **Default**.
Default will use the value specified on the Autosampler Accessory page.

NOTE: If **Flush** has been specified, all entries in the **Return Position** column are set to **Rinse Port**. The **Return Position** column can only be edited if **Sample return** is selected on the Autosampler Accessory page. The **Return Position** column does not have a 'Default' option. Instead, if you wish the sample to return to the vial from which it came, select **Original**, otherwise select or enter the number of the vial into which it should go after it has been run.

OR

Click on the arrow to display a drop-down list.

The list contains Default and a number (the number is the value currently entered on the autosampler accessory page (the default value)).

3. If you wish to use the Default value specified on the Autosampler Accessory page, select **Default**.

OR

If you wish to enter your own value in the field, select the only value currently listed in the drop-down list (this will subsequently be edited).

When the value is entered in the field, it is highlighted to show that it can be edited.

4. Enter the required value.
This will override the setting on the autosampler accessory page.
5. Repeat for all the other autosampler columns.
6. Repeat for all samples in the sample table.

NOTE: All sample information that has not been completed at runtime will have the default values used.

What columns are added to the Sample Table when a Universal Reflectance Accessory (URA) is installed?

When in %R, three columns are added to the sample table – **Width**, **Length**, and **Angle**. These columns are removed when in A or %T ordinate mode.

By default, the values on the accessory page are used on the sample info page.

Width	Length	Angle
5.0	5.0	8.0
4.0	4.0	45.0

All these values can be altered by sample (or by measurement, if there are many measurements for one sample).

- Click in a field and edit the value as required.

The maximum width is 5 mm and the maximum length is 5 mm. The width does not have to be the same value as the length. The angle of incidence can be in the range 8° to 70°. The URA operates at step intervals of 0.5°.

When measuring several angles with the URA, do the angles have to be specified incrementally in the Sample Table?

No, the angles can be specified in any order. The accessory will move to the required angle for each measurement.

How should I set up the Sample Table if I want to measure many angles of one sample using the URA?

We recommend that you use measurements to measure many angles of one sample.

1. Enter the number of **Samples** to be run.
2. Click Format Sample Table.
The Table Builder dialog is displayed.
3. Select the Design tab.
4. Select **Measurements** from the drop-down list.
5. Enter the number of measurements per sample.
6. Select whether to Sequence by sample or Sequence by measurement.
Select Sequence by sample to run the samples in sample order rather than measurement order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S1M2, S2M1, S2M2.
Select Sequence by replicate to run the samples in replicate order rather than sample order. For example, if you had 2 samples (S1 and S2) and each sample had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the samples would be run in the order: S1M1, S2M1, S1M2, S2M2.
7. Click **OK**.

The measurements are added to the sample table.

NOTE: A maximum of 200 measurements can be added for each sample. You can edit the width, length and angle fields for each measurement.

What columns are added to the Sample Table if a URA is installed but the instrument is used in %T or A Ordinate mode?

The instrument can be used in %T or A ordinate mode with the URA installed. When %T or A ordinate mode is selected, two columns are added to the sample table – **Sample Pathlength** and **Baseline Pathlength**.

Sample Pathlength	Baseline Pathlength
0.0	0.0
0.0	0.0
0.0	0.0

These columns are used to adjust the pathlengths. This may be particularly useful if you have thick samples and wish to improve the Signal:Noise ratio. You can independently adjust the sample and/or baseline pathlength for each sample (or each measurement or replicate if either of these are being used).

Custom columns

What is a custom column?

A custom column is a user-defined column that can be added to the Sample Table.

How do I add a custom column?

1. Click Format Sample Table.
The Table Builder dialog is displayed.
2. Select the Columns tab.
3. Click **Add**.
The Column Type dialog is displayed.
4. Select Data entry, Text selection or Sample tag and click OK.
The appropriate Format dialog is displayed.
5. Edit the details as required.

How do I delete a custom column?

1. On the columns tab of the Table Builder dialog, select the column you wish to delete by clicking on it.
A dotted line appears around the name to show that it is selected.
2. Press the **Delete** key on your keyboard.
The column is removed.

NOTE: Sample ID, Description and Type are mandatory columns and cannot be deleted.

Formatting columns

What types of column are present in the Sample Table?

There are four types of column:

System Column	System Columns are the first three columns that appear by default in a table – Sample ID, Description, and Type. It is not possible to define any other System Columns.
	NOTE: System Columns cannot be deleted.
Data Entry Column	A Data Entry Column is a custom column added by the user. Numeric values that can then be used as variables in equations are expected in a Data Entry Column.
Text Selection Column	A Text Selection Column is a custom column added by the user. Each cell in a Text Selection column contains a drop-down list of options. These options are defined within the Format Text Selection dialog.
Sample Tag	A special text selection column that is used as an identifier in special equations that use variations on a sample. The Sample Tag becomes part of the spectrum name.
	NOTE: Only one sample tag column may be added to a table.

The type of formatting that can be performed on a column depends on the type of column.

What is a Sample Tag?

A sample tag is a special custom column that can be added to assign an alias to a sample. For instance, if you have a group of samples that each have their own particular identifier that you need to use as the sample name, but you also want to pick out samples as belonging to special groups, you can use a sample tag.

Sample tags are like other text selection columns in that the method developer assigns the entries available and then when samples are entered in the table there is a drop-down list of options for the user to select from. Where this differs from standard text is that these options then become available in the Variables list in the Equation Builder so you can specifically select an equation to operate only on samples with that tag, or perform an operation that differentiates between samples with different tags.

NOTE: You should not use the words Sample , Blank or Control as sample tags as this will cause confusion with the sample types.

An example use of sample tags is where you have several matched pairs of samples and you want to subtract one from the other:

1. Add the **Sample Tag** column and format it with the list reading **A** and **B**.
2. In your Sample Info table use the drop-down list in the new Sample Tag column to select which samples are A and which are B.
In this example, they must run in sequence one after the other, and there must be an even number of each.
3. In Processing, set up an equation using these new variables, for example **Height[A, 265]-Height[B, 265]**.

When samples are run a Custom table will appear in the Results page that gives the result of this equation for each pair of samples.

How do I format a Column?

1. On the Sample Info page click **Format Sample Table**.
The Table Builder dialog is displayed.
2. Select the Columns tab.
3. From the list of columns highlight the column you want to format and click **Format**.
The appropriate formatting dialog is displayed:

System Column

NOTE: Only the font can be changed in a System Column.

The Format System Column dialog is displayed.

1. Click Change Font.
The Font dialog is displayed.
2. Select the **Font**, **Font style**, **Size**, **Effect**, **Color** and **Script**, and then click **OK**.
An example of the selected font is shown on the Format System Column dialog.

Data Entry Column

The Format Data Column dialog is displayed.

1. If you wish to format a Data Entry Column that already exists, select the **Name** of the Data Entry Column that you wish to format from the drop-down list of all available Data Entry Columns.

OR

To create a new Data Entry Column, enter a new **Name**.

NOTE: It is not possible to edit the **Name** of a previously saved column. If you try to edit the **Name** of a previously saved column, a new column is created.

2. If you wish to add units to a column header, enter the Units in the field:

Column Details –	
Name	
Initial Volume	
Units	
ml	

The units are automatically placed in brackets after the column name.

3. If you want the analyst to enter information in the selected column before they can run a sample, select **Mandatory**.
4. If you want the analyst to only be allowed to enter numbers in the selected column, select **Numbers only**.

NOTE: Significant figures and Decimal places are only available when Numbers only is selected.

5. Click Change Font to change the Font type, Font style, Size, Effect, Color and Script.
6. If **Numbers only** has been selected, select the number of **Significant figures** (0 to 9) or the number of **Decimal places** (1 to 9) from the drop-down list.

Text Selection Column

The Format Text Selection Column dialog is displayed.

1. If you wish to format a Text Selection Column that already exists, select the **Name** of the Text Selection Column that you wish to format from the drop-down list of all available Text Selection Columns.

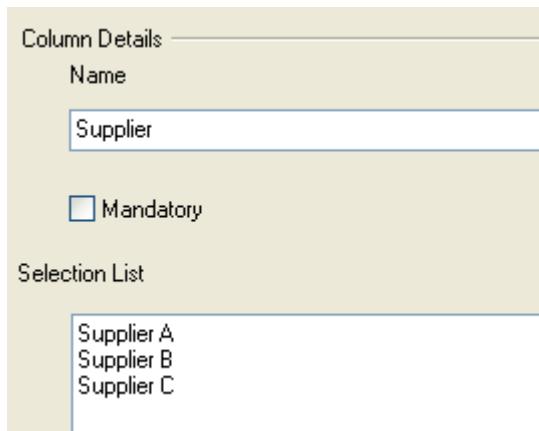
OR

To create a new Text Selection Column, enter a new **Name**.

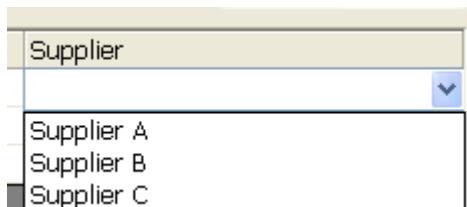
2. If you want the analyst to enter information in the selected column before they can run a sample, select **Mandatory**.
3. Enter the words to be selectable (from a drop-down list) for each cell in the column in the **Selection listfield**:

In the Supplier column you want the analyst to be able to choose from Supplier A, Supplier B and Supplier C.

Enter Supplier A, Supplier B and Supplier C in the Selection list.



When the user clicks on a cell in the **Supplier** column of the table a drop-down list is displayed containing Supplier A, Supplier B and Supplier C.



Click Change Font to change the Font type, Font style, Size, Effect, Color and Script.

How do I force the user to enter information in a custom column I have created?

- When creating the custom column, select **Mandatory** within the Format column dialog. The user then must fill in the column prior to data collection.

How do I auto-increment Sample IDs?

1. Click Format Sample Table.
The Table Builder dialog is displayed.
2. Select the ID tab.
3. Select Use this format

When Use this format is selected, the Sample ID column of the Sample Table is automatically filled in by incrementing the prefix which you should enter in the field below. For example, if you enter the prefix Trial, the samples will be numbered Trial1 to Trialn (where n is the number of samples). The numbering is remembered across all tasks created by this method.

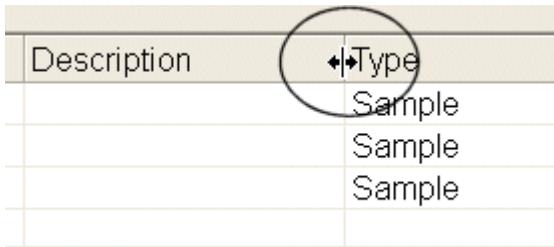
How do I force the user to enter a sample ID before they run a sample?

1. Click Format Sample Table.
The Table Builder dialog is displayed.
2. Select the ID tab and select **User Must Enter**.
The Sample ID column remains empty and the user is then forced to enter a sample ID before they run the sample.

How do I alter the width of the columns in the Sample Table?

1. Position the mouse pointer between two columns.

The mouse pointer changes:



Description	Type
	Sample
	Sample
	Sample
	Sample

2. Drag the edge of the column in the required direction.

Can I copy and paste between cells?

3. Select the cell you wish to copy.

A dotted line appears around the cell to show that it is selected.

4. From the Edit menu select **Copy**.

5. Position the cursor in the cell into which the information is to be pasted and then select **Paste** from the Edit menu.

The information is pasted into the selected cell.

NOTE: It is also possible to paste information from the Windows clipboard into the Sample Table.

Copy and **Paste** are also available from the context menu shown by right-clicking in a cell.

Can I select adjacent cells?

Yes.

- You can select adjacent cells by clicking on the first cell and then dragging the mouse over all the required cells.

OR

You can click in the first cell and then hold down the Shift key and use the arrow keys on the keyboard to select the block of cells.

NOTE: It is not possible to select non-adjacent cells.

What do I see in the Sample Table when a sample has been run?

When a sample has been run, a green tick mark appears in the first column:

	Sample ID
✓	1 Sample1
✓	2 Sample1
✓	3 Sample2
✓	4 Sample2

How do I select all the cells in the Sample Table?

1. Click in the first cell of the table:

(marked red in this example)

	Sample ID	Description	Type
✓ 1	Sample1		Replicate 1
✓ 2	Sample1		Replicate 2
✓ 3	Sample2		Replicate 1
✓ 4	Sample2		Replicate 2

All the table is highlighted blue to show that it is selected.

2. Use CTRL+C to copy the data to the clipboard.

How do I select an entire row?

1. Click in the first cell of the row (the row number):

✓ 1	Sample1	Replicate 1
-----	---------	-------------

The row is highlighted blue (apart from the first cell in the field) to show that it is selected.

2. Use CTRL+C to copy the row to the clipboard.

How do I select adjacent rows?

1. Click in the first cell of the first row you wish to select (the row number):

✓ 1	Sample1	Replicate 1
-----	---------	-------------

The row is highlighted blue (apart from the first cell) to show that it is selected.

2. Hold down the SHIFT key and then click in the first cell of the final row you wish to select.

All rows in between (and including the selected rows) are highlighted blue.

3. Use CTRL+C to copy the rows to the clipboard.

How do I select an entire column?

1. Click on the header (column title) of the column you wish to select.

The column is highlighted blue (apart from the first cell below the title) to show that it is selected.

2. Use CTRL+C to copy the column to the clipboard.

How do I select adjacent columns?

1. Click on the header (column title) of the column you wish to select.

The column is highlighted blue (apart from the first cell below the header) to show that it is selected.

2. Hold down the SHIFT key and then click in the header of the final column you wish to select.

All columns in between (and including the selected columns) are highlighted blue (except the first cell below the header in the first column).

3. Use CTRL+C to copy the columns to the clipboard.

Are hidden table columns available for export?

Yes, if a column is added to a table but is then hidden, it will be available for export as the software will populate all specified columns even if they are not displayed. The Data Export dialog (obtained via the Output page in the Workspace) will automatically list all table columns (displayed and hidden).

Using a pre-defined Sample Table

Can an analyst alter the number of replicates specified in the Sample Table?

Yes in the Standard version of UV WinLab.

In the Enhanced Security version of UV WinLab, the analyst cannot add or remove the replicate sample rows.

Can an analyst alter the position of the control samples?

In the Standard version of UV WinLab, the analyst can alter the position of any control samples.

In the Enhanced Security version of UV WinLab, the analyst is not able to alter the position of any control samples.

Can an analyst change a control sample to any other type of sample?

Yes, a control sample can be changed to any other type.

Can an analyst overwrite an auto-incrementing prefix?

Yes, the analyst can overwrite the prefix.

Running a Sample Table

In most cases, a Developer will create a method from an existing method, or one of the methods provided with UV WinLab will be used to create the task. The Analyst will open the method, enter the required information in the Sample Table and then run the samples. Once the data has been collected (and, where specified, processed), the task is saved.

NOTE: The settings on the Data Collection page MUST NOT be altered while the Sample Table is being run.

How does the analyst use the Sample Table at run time?

1. On the Sample Info page enter the number of **Samples** to be run.
2. Enter any other mandatory information required in the Sample Table.
For example, a concentration might need to be specified.

NOTE: The Analyst will be informed if a mandatory field is left blank.

3. Click .

The samples are run in the order listed in the Sample Table. The Graphs tab displays automatically at the start of the run and the spectrum appears in the graph display. A green tick appears next to the sample name in the Sample Table when it has been run.

Does the analyst have to complete the Sample Table before a run?

The analyst will be required to enter the number of samples to be run, and any information that has previously been specified as mandatory by the Developer. The sample table will be checked for this information before the samples are run.

NOTE: Samples may be added to the Sample Table during data acquisition. DO NOT ADD REPLICATES TO SAMPLES WHEN THE SAMPLE TABLE IS RUNNING.

Will the analyst be prompted to perform an autozero before the samples in the Sample Table are run?

Yes, the analyst will be prompted to perform an autozero before the samples in the Sample Table are run.

NOTE: There is no graphical display for the autozero but the Instrument Status Bar is updated appropriately.

Can samples be added to the table whilst the Sample Table is being run?

No. While the Sample Table is running it is not possible to edit the Sample Table in any way.

Can samples be added to the table when the sample list has been run?

Yes, you can add samples and then run these additional samples.

Can information in the Sample Table be edited whilst the Sample Table is being run?

No, it is not possible to edit information while data is being collected.

Can information in the Sample Table be changed if data has been collected?

Yes, it is possible to edit information after data has been collected:

- Select the cell and amend it as required.

In the Enhanced Security version of UV WinLab, you will be asked to confirm the edit request and provide a comment to be recorded with the sample in the Sample Event Log.

How do I exclude a sample once the Sample Table has been run?

NOTE: This is only applicable to the Enhanced Security version of UV WinLab.

1. Select the sample to be excluded.
2. Click **Delete**.

OR

Right-click on the sample and select **Exclude** from the menu.

You are asked to confirm that you wish to exclude the sample.

3. Click **OK**.
- If you are using the Enhanced Security version of UV WinLab, the green tick is replaced by a red cross and the row is grayed.
- If you are using the Standard version, the sample is removed from the Sample Table.

How do I include a sample that I have accidentally excluded?

NOTE: This is only applicable to the Enhanced Security version of UV WinLab.

1. Select the sample that has been excluded.
2. Right-click anywhere in the row and from the menu select **Include**.
The row is no longer grayed.

Can I view reference spectra while samples in the Sample Table are being run?

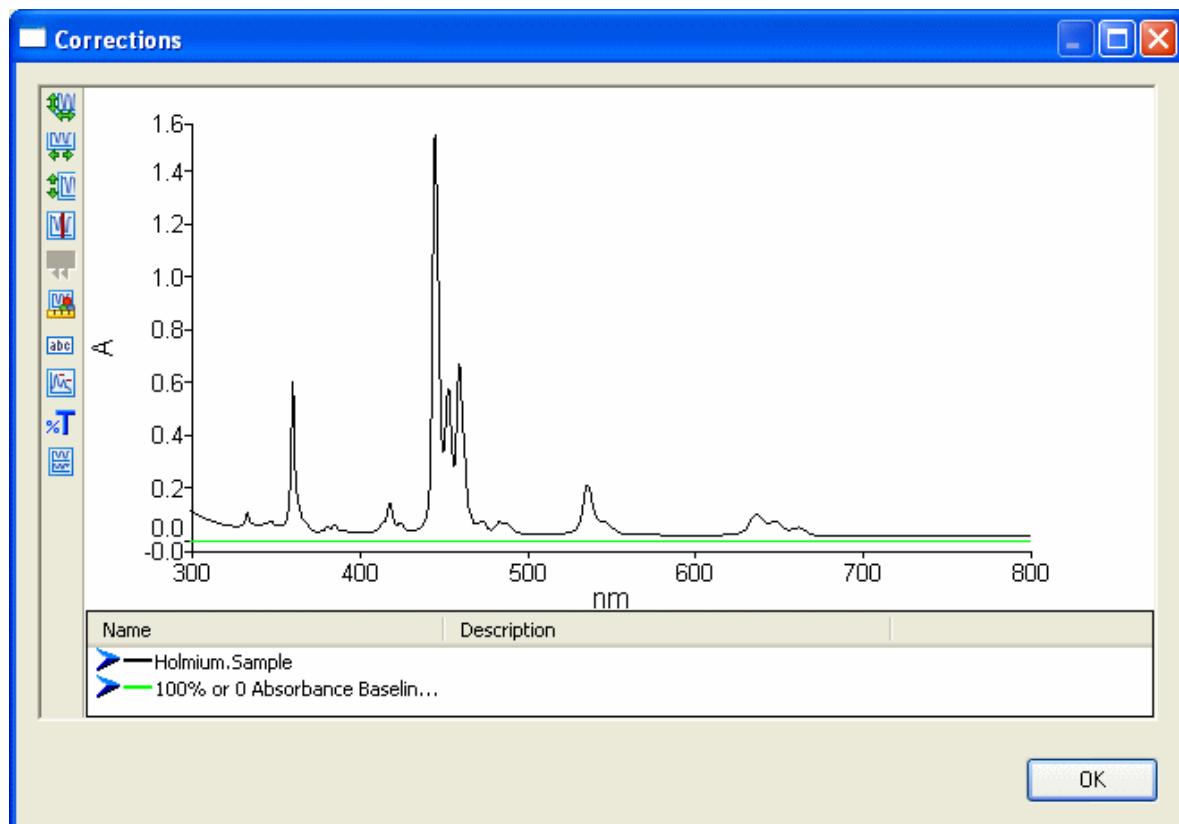
Yes. Reference spectra can be added to the Graphs tab for comparison using the Add Reference button. These are not saved as part of the task.

How do I view the correction spectrum/spectra for a sample?

The correction spectra can be viewed within the Sample Table, Standards Table or Results Table.

- Right-click on the sample whose correction spectrum you wish to view and select **View Corrections**.

A graph window is displayed showing the sample and correction spectra:



How do I run a Simulation?

NOTE: The Simulator must have been setup correctly before simulating running the Sample Table. Ensure that the Simulated instrument is selected under Instruments in the UV WinLab Explorer. The Simulator is only available for Medium-performance instruments.

1. Enter the names of the data in the Sample Table.

The names must be identical to the names listed in Windows Explorer but the extension should not be included. For example, if you have the sample Holmium.asc you should enter Holmium in the Sample Table.

NOTE: If the name is not entered correctly the simulator will report an error when it tries to run.

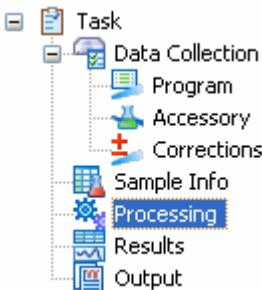
2. Click .

The samples are 'run' and the graphs displayed.

See also

- Accessories
- Quant Calibration

Processing



Processing enables a spectrum to be changed in some way or another spectrum can be created from the original spectrum if the method involves spectra rather than discrete wavelengths. Data manipulation can be performed by setting up equations.

Processing is achieved through a series of chained events, with parameters for each event being separately configured. Spectral processing performs actions on whole spectra.

Pre-processing and Post-processing can be defined for Quant Methods and Timedrive Methods. Pre-processing applies to all samples and standards before a Quant or Rate calculation is performed. Post-processing allows processing to be applied to the Quant or Rate results.

NOTE: Pre-Processing is simply called **Processing** in Timedrive Methods.

NOTE: Pre-processing and Post-processing are defined for scanning Quant methods. For Wavelength Quant, there is only Post-processing which is simply called Processing.

What is displayed on the Processing page?

The table defining the processing parameters and the order they will be performed in is displayed.

What processes are available?

The following options are available:

Equation	Difference
----------	------------

Conditional Formatting	Interpolate
------------------------	-------------

(Conditional Formatting is part
of Equation settings)

Arithmetic	Normalize
------------	-----------

Convert X	Peak Table
-----------	------------

Convert Y	Smooth
-----------	--------

Derivative

NOTE: The processing options that are available depend on the type of Method selected. For example, smooth is not applicable to a Wavelength Program method.

What do the buttons on the Table do?

Up	Moves the selected row up the table and therefore changes the order in which the processes occur.
Down	Moves the selected row down the table and therefore changes the order in which the processes occur.
Settings	Displays a dialog enabling you to change the settings of the selected process. The dialog displayed depends on the Process represented by the selected row.
Clear	Removes the selected row from the table.
Add	Adds a new row to the bottom of the table.

How do I add the processes to the Table?

1. Click **Add**.
A row is added to the Table. By default, Process is set to Select and Settings is set to None.
2. In the **Process** column, click  and select the type of Process from the drop-down list.
The Process field is updated and the Properties field displays the default settings for the selected Process.
3. To change the default settings, click **Settings**.
The Settings dialog for the Process is displayed.
4. Edit the settings and click **OK**.
The Settings are updated.
5. Repeat steps 1 to 4 for all the processes you wish to apply to the collected data.

How do I select a row in the table?

- Click anywhere in the row that you wish to alter.
The row becomes blue to show that it is selected.

How do I select a process?

A process is selected from a drop-down list of all available processes. By default, **Select** is displayed in the field.

1. Click  on the right hand side of the process field.
A drop-down list of all available processes is displayed.
2. Select the required process.
The process is displayed in the field and the Settings are updated with the default settings.

How do I change the Settings?

1. Select the required row.
The row is highlighted blue to show that it is selected.
2. Click **Settings**.
The appropriate settings dialog is displayed.
3. Select the required Settings and click **OK**.
The dialog closes and the table updates with the new settings.

NOTE: It is not possible to edit the Settings field within the table. You must click **Settings** and select the settings from the dialog.

Double-clicking in the Settings field also displays the appropriate settings dialog.

Does it matter in which order the processes are listed in the table?

The order in which the processes are listed is the order in which they will be performed. It is therefore important that you place them in the sequence that you want the processing to be applied.

How do I reorder the processes in the table?

It is possible to reorder the processes using the Up and Down buttons.

- Select the required row from the table.
Click Up or Down to move the row in the required direction.

How do I clear a row from the table?

1. Select the required row.
2. Click **Clear**.
The row is removed from the table.

How do I add a row to the table?

- Click **Add**.
A row is added at the bottom of the table.

Equation

Equation is used to perform calculations on raw data or on numerical data obtained from other processing. Equations are constructed from **Functions** and **Operators** applied to all samples or specific samples as set by **Variables**. The results of one equation can be used as **Variables** in another, so that complex calculations can be carried out as a sequence of simpler operations.

How do I create equations?

An equation is constructed as follows:

1. From the Processing page, select **Equation** as the type of process and click **Settings**.

The Settings dialog for Equation is displayed.

2. Enter an **Equation** name.

This is used to label the equation on the Processing page and is also used as the tooltip for the equation in subsequent equations.

OR

Select a previous equation from the drop-down list.

3. Enter a **Variable** name.

This is the name that will be seen in the Variables list on the dialog. This will be used when inserting one equation in another – that is, using an equation as a variable in another equation.

The default Variable name is the same as the default equation name.

4. The **Result Column** name is the name of that will be used as the column header in the Results Table. This field is read only. The default Result Column name is the same as the default equation name. If you wish to change the default name, click **Edit**.

The Format Data column dialog is displayed. See 'How do I format the result column name ?' below for information on how to change the name.

5. Select **Add to favorites** if you want to be able to re-use this equation in other Methods.

The equation will then appear in the Equation name drop-down list next time you open the Equation Settings dialog.

NOTE: When you select **Add to favorites**, the information on the Equation page is locked and cannot be edited. To edit the Equation you must first enter a new **Equation name**. It is not possible to de-select **Add to favorites** once it has been selected.

6. Enter the details of the **Equation**, either by typing directly or clicking on items in the **Operators**, **Functions** and **Variables** lists.

Further information on each of the Functions is given below, but basically each function has a number of settings that are required or optional (as shown in the tooltip that is displayed when a function is added to the equation). The first of these settings is usually a <spectrum variable> which by default will be set to All. This means that the function will be applied to each sample, however, if you want to create an equation that performs a function on a specific type of sample or even a specific spectrum, change the All to one of the other Variables or click Browse to read in a specific spectrum.

7. Click **Check** to ensure that the syntax used in the equation is correct.

8. Click **OK**.

The new column is created with the equation providing the data for that column. The column will now be shown in the Organize columns dialog on the Results page where you can choose whether it is displayed or hidden, and where in the table it appears.

How do I format the result column name?

1. Click **Edit**.

The Format Data Column dialog is displayed.

2. If required, change the **Name**.

This is what will be displayed in the top of the column in the Results Table.

NOTE: It is not possible to create two columns with the same name.

3. If required, add **Units** to the column header.

The units are automatically placed in brackets after the column name.

4. Click **Change Font** to change the font type, style, size and color.

5. Select the number of **Significant figures** (0 to 9) or the number of **Decimal places** (0 to 9) from the drop-down lists.

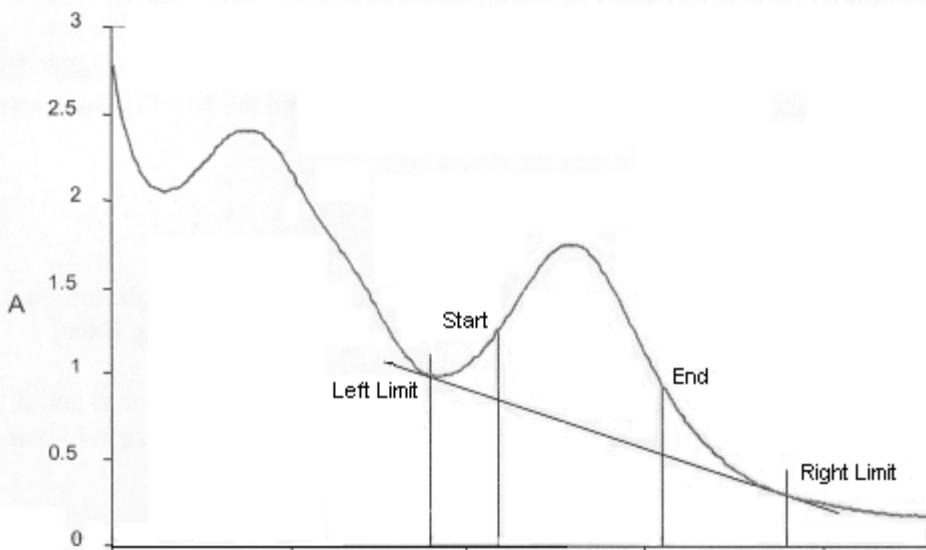
6. Click **OK**.

These column settings will be applied to the Results Table.

How do I use Area?

This calculates the area under the curve between two set points. The formatting of the command is "Area[<spectrum variable>, Left Limit, Right Limit, Left Base Point(optional), Right Base Point(optional)]"

For example: **Area(All,800,900,700,1000)** calculates the area between abscissa positions 800 and 900 above a baseline with base points at 700 and 1000.



The area reported is that between the **Left Limit** and **Right Limit**, either relative to a zero baseline or relative to a linear baseline intersecting the spectrum at the specified baseline points. If no baseline points are specified, the area reported is that above zero. If only one baseline point is specified, a horizontal baseline is constructed from that point.

How do I use Yval?

Yval is used to obtain the ordinate value at a specified abscissa position. The formatting of the command is "**Yval[<spectrum variable>, Abscissa position]**", for example, **Yval[All, 360]** would give the ordinate value for each spectrum at 360 nm.

It is likely to be used to obtain a net absorbance from the difference in the values at two positions. A typical use (instead of **Ymax** or **Ymin**) would be to determine the amplitude of a band on a varying sloping background, where apparent shifts in the position of the band maximum should be ignored.

How do I use PeakX?

PeakX is used to find the position of a peak or valley within a specified range. The formatting of the command is "**PeakX[<spectrum variable>, Start Range, End Range, Peak Threshold, 1=Peak/-1=Base, 1=Interpolated(default)/0=sampled position (optional)]**".

A peak is identified by having a valley that exceeds the selected threshold on either side. **PeakX** gives the abscissa position of a peak or valley over the specified range. If there is more than one peak that exceeds the threshold the routine returns the value of the first peak or valley it finds.

For example: **PeakX[All, 450, 350, 0.1, 1, 0]** will return the first peak between 350 and 450 over the 0.1 threshold, and return the abscissa value of the sampled data point (as opposed to the peak position when a smooth curve is fitted to the sampled data points).

How do I use Ymax?

Ymax finds the maximum ordinate value within a specified range instead of measuring the height at a fixed position. This has the advantage in the processing of multiple spectra where there are shifts in the locations of the maximum. The formatting of the command is "**Ymax[<spectrum variable>, Start Range, End Range]**", where **Ymax[All, 450, 350]** will give the maximum ordinate value found between 450 and 350.

How do I use Ymin?

Ymin finds the minimum ordinate value within a specified range. This has the advantage in the processing of multiple spectra where there are shifts in the locations of the minimum. The formatting of the command is "**Ymin[<spectrum variable>, Start Range, End Range]**", where **Ymin[All, 450, 350]** will give the minimum ordinate value found between 450 and 350.

How do I use XYmax?

XYmax is used to return the abscissa position of the maximum ordinate value found by interpolation over a specified range. The formatting of the command is "**XYmax[<spectrum variable>, Start Range, End Range]**", where **XYmax[All, 450, 350]** will give the abscissa position for the maximum ordinate value found between 450 and 350.

How do I use XYmin?

XYmin is used to return the abscissa position of the minimum ordinate value found by interpolation over the specified range. The formatting of the command is "**XYmin[<spectrum variable>, Start Range, End Range]**", where **XYmin[All, 450, 350]** will give the abscissa position for the minimum ordinate value found between 450 and 350.

How do I use Height?

Height is used to obtain the ordinate value at a specified abscissa position relative to an optional baseline. The formatting of the command is "**Height[<spectrum variable>, Abscissa position, Left Base Point(optional), Right Base Point(optional)]**" where **Height[All, 400, 450, 350]** would give the height at 400 corrected for a baseline drawn between 450 and 350. If one base point is specified, a horizontal baseline is calculated from that point. If no base points are specified then the height is not corrected for baseline.

How do I use Interval?

Calculates the data interval of the spectrum. The formatting is "**Interval[<spectrum variable>]**", where **Interval[All]** simply gives the data interval of each spectrum.

How do I use Npts?

Calculates the number of data points in the spectrum. The formatting is "**Npts[<spectrum variable>]**", where **Npts[All]** simply gives the number of data points in each spectrum.

How do I use End?

Calculates the last abscissa point of the spectrum. The formatting is "**End[<spectrum variable>]**", where **End[All]** simply gives the last abscissa point of each spectrum.

End can also be used nested within a command in place of the actual start value for a command, for example **XYmax[All, Start[All], End [All]]** will give the abscissa value for the maximum ordinate anywhere in the spectrum without having to set start and end limits.

If End is used with Wavelength Programmed data it will return the last wavelength in the list, so combined with Height, **Height[All, End[All]]** will give the height at the last wavelength in the list without pre-setting that value.

How do I use Start?

Calculates the first abscissa point of the spectrum. The formatting is "**Start[<spectrum variable>]**", where **Start[All]** simply gives the first abscissa point of each spectrum.

Start can also be used nested within a command in place of the actual start value for a command, for example **XYmax[All, Start[All], End [All]]** will give the abscissa value for the maximum ordinate anywhere in the spectrum without having to set start and end limits.

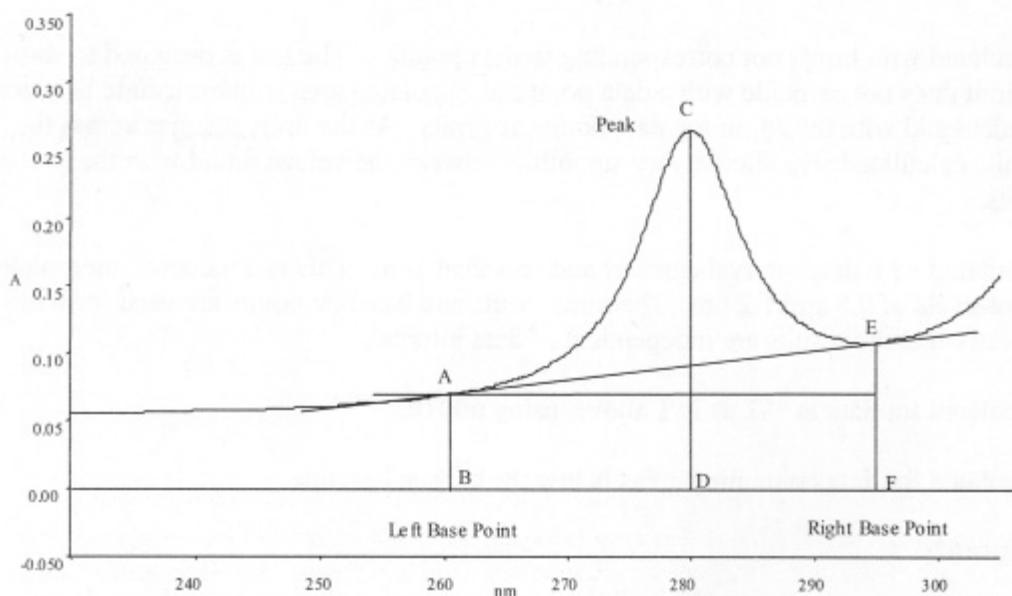
If Start is used with Wavelength Programmed data it will return the first wavelength in the list, so combined with Height, **Height[All, Start[All]]** will give the height at the first wavelength in the list without pre-setting that value.

How do I use MaxHt?

MaxHt determines the height of the biggest peak in a range, relative to a baseline if required. The most common use is to obtain absorbance values for quantitative applications. The most common use of %T data is for measurements of optical filters.

The formatting of the command is "**MaxHt[<spectrum variable>, Start Range, End Range, Left Base Point, Right Base Point]**" where **MaxHt[All, 400, 350, 450, 300]** gives the height of the biggest peak between 450 and 350 corrected for the baseline constructed between 450 and 300, using cubic interpolation between data points.

When **Left Base Point** and **Right Base Point** are given the same value, a horizontal baseline is used, as shown below.



How do I use Mean, SD, and RMS?

Mean,SD (Standard deviation) and **RMS** (Root mean square deviation) are standard statistical operators. When applied to a spectrum or wavelength programmed data they produce the average, standard deviation or RMS deviation ordinate value over all wavelengths.

Mean and SD are mostly used to evaluate results from repeat measurements in order to improve precision and estimate uncertainty.

In each case the format of the command is "**[<spectrum variable> or <number variable>]**" which means that Mean, SD and RMS can be applied to spectra or to numbers calculated by another equation. For example if you had an equation to find the area of a peak in each replicate you could then use these commands to find say the mean of the replicate areas. In this case the results would then appear in the replicates table.

How do I use Trend?

Trend is used to measure variation that is slow relative to the frequency of measurement, that would be regarded as drift rather than noise. It operates on a timedrive measurement or a spectrum. It is used in noise measurements to separate out any longer term drift.

Trend calculates the slope of a linear fit to the data on which it operates. It can be applied to the complete spectrum or timedrive measurement or to a limited region defined by **Start Range** and **End Range**. Unlike other statistical functions it cannot be applied to the results of calculations on a group of spectra.

The formatting of the command is "**Trend[<spectrum variable>, Start Range (optional), End Range (optional)]**" where **Trend[All, 450, 350]** calculates the slope of a linear fit to all data between 450 nm and 350 nm.

How do I use Exp, Log, Sqr, Sqrt and Ln?

These are standard mathematical operations that can be applied to spectra or numerical data. For example they can be used to further manipulate the results of a different equation.

NOTE: A value of zero is returned if you try to take the log, natural log (Ln), or square root (Sqrt) of a negative number.

Log returns the base <n> logarithm of a number or a spectrum. The formatting of the Log command is "**Log[<spectrum variable> or number, base]**", for example, **Log[All, 10]** would give the logarithm (base 10) of all spectra. Base 10 can be replaced by any base if required.

How do I use XVal?

XVal is used to determine the abscissa value at a specified Y value. If interpolation is used (this is the default), the equation will report the actual abscissa value at the specified Y value. If interpolation is not used (by entering 0 in the syntax), the equation will report the abscissa value of the nearest data point.

XVal will report the first abscissa value at the specified Y value. The Search from Start / End options in the equation syntax are useful if the spectrum has more than one abscissa value at the specified Y value (for example, if there is more than one peak in the spectrum). You can start the search from the end of the spectrum that ensures the required value is reported.

The formatting of the command is "**XVal[<spectrum variable>, Ordinate value, Start Range, End Range, 1=Search from Start(default)/-1=Search from End(optional), 1=Interpolated(default)/0=sample position (optional)]**"

NOTE: If you include the Interpolated command in the equation, you must include the Search from command to ensure the syntax is correct. **XVal[All, 1.2, 900, 300, 1]** means that the Interpolated value has not been defined. **XVal[All, 1.2, 900, 300, ,1]** where no value has been entered for the Search from value (only a space between 2 commas) would return an incorrect syntax message.

Examples:

XVal[All, 1.2, 900, 300, 1, 1]

The above equation will calculate the abscissa value for all spectra at the Y value of 1.2, between 900 and 300 nm. It will start the search from 900 nm, and it will be interpolated (give the actual value rather than the nearest data point).

XVal[All, 1.1, 800, 700, -1, 0]

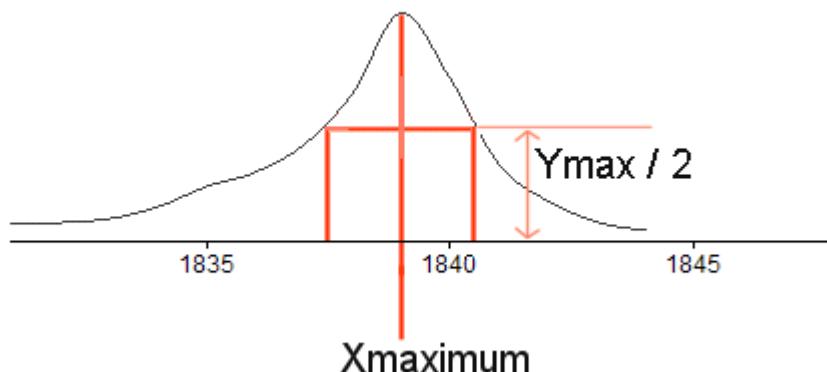
The above equation will calculate the abscissa value for all spectra at the Y value of 1.1, between 800 and 700 nm. It will start the search from 700 nm (Search from End), and it will not be interpolated (give the nearest data point).

XVal[All, 1.2, 900, 300]

The above equation will calculate the abscissa value for all spectra at the Y value of 1.2, between 900 and 300 nm. It will start the search from 900 nm (the default if nothing is specified), and it will be interpolated (the default if nothing is specified).

XVal can be used to calculate the width of a peak at half height.

Example:



Equation	Conditional Formatting
Equation name	Equation1
Variable name	Equation1
Result column name	Width at half height
Equation	$XVal[All, Ymax[All, 1835, 1845] / 2, XYMax[All, 1835, 1845], 1845] - XVal[All, Ymax[All, 1835, 1845] / 2, XYMax[All, 1835, 1845], 1835]$

In this example, we are looking at the peak between 1835 and 1845 nm. The width at half height will be reported for all spectra. The functions Ymax and XYMax are used to calculate XVal. Ymax calculates the maximum ordinate value within the range. XYMax calculates the abscissa position of the ordinate maximum.

In the first part of the equation – $XVal[All, Ymax[All, 1835, 1845] / 2, XYMax[All, 1835, 1845], 1845]$ -

1845 means the first XVal is looked for starting from 1845.

In the second part of the equation – $XVal[All, Ymax[All, 1835, 1845] / 2, XYMax[All, 1835, 1845], 1835]$ – 1835 means the first XVal is looked for starting from 1835.

To calculate the width at half height for any data, use the equation above and simply replace the values (1835, 1845).

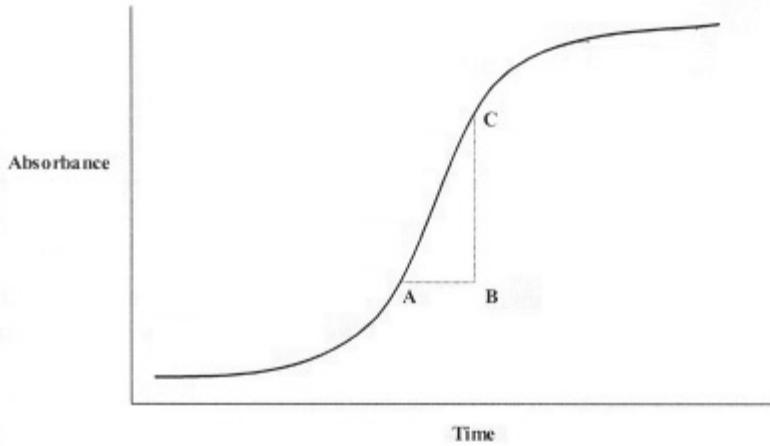
How do I use RCoeff?

This is the correlation coefficient.

The formatting of the command is "RCoeff[<spectrum variable>,Start Range(optional), End Range(optional)]"

If the Start Range and End Range are not included, the whole range scanned for the spectrum is used.

The graph below is an example of the type of data that may be produced from a Timedrive task.



The **Trend** function can be used to find the slope AC.

RCoeff can be used to find the goodness of fit of the slope to the actual curve.

How do I use Ri?

Ri is the refractive index.

The formatting of the command is "Ri[<spectrum variable>, Start Range, End Range, Thickness, Num Fringes (-1=Auto), Angle of Incidence, Thickness Units: 1=Å/2=µm, Peak Threshold(optional)]"

How do I use Tcalc?

Tcalc is the equation used to calculate thickness.

The formatting of the command is "Tcalc[<spectrum variable>, Start Range, End Range, Refractive Index, Num Fringes(-1=Auto), Angle of Incidence, Result Units: 1=Å/2=µm, Peak Threshold(optional)]"

What sorts of variables are there?

There are three distinct types of variables that will appear in the Variables list:

- Sample variables – **All** (which is used to apply a function to every sample), **Sample**, **Blank** and **Control** (which can be used to pick a certain sample type only), **Replicates**, if replicates are being used this will group results by the replicates of each sample, and an individual entry for each sample in the task at this time (so you can use an individual sample in an equation).
- Sample tags – If you set up a Sample Tag column, the entries for that column will appear as variables so that you can apply functions only to samples with a certain tag. There will also be a **Sample Tags** entry that enables you to group by sample tag, as per **Replicates** above.
- Other columns – If you have other columns, either equations you've already set up or Data Entry columns that are set to **Numbers only** added to the sample table. If this is a post-processing page, the Rate and Quant results columns will also be available for equations.

What is the Replicates variable?

The Replicates variable is used to process all samples that are of the type **Replicate** but keeps the results grouped by **Sample ID**. This allows the results of the equation to be used in the Mean, SD and RMS functions so that these calculations are performed on all replicates of the same sample and the result is then put in the Replicates table

For example, if it was required that the maximum ordinate value of each replicate is calculated and then the mean and standard deviation of the results for each Sample ID is produced, this can be achieved with the following equations:

`Ymax[Replicates,Start[Replicates],End[Replicates]]`

Equation variable **YMaxResult**

Column header **YMax**

This will calculate the maximum ordinate value for each sample marked replicate and put the result in the Replicate Means table in a column called **YMax**.

`Mean[YMaxResult]`

Column header **YMaxMean**

This will calculate the mean result for the replicates of each sample and put the result in the Replicate Means table in a column called **YMaxMean**.

`SD[YMaxResult]`

Column header **YMaxStdDev**

This will calculate the standard deviation result for the replicates of each sample and put the result in the Replicate Means table in a column called **YMaxStdDev**.

How do I perform a calculation on individual measurements that results in a spectrum and then these resultant spectra are displayed in the measurement table?

For example, perform the absorptance calculation manually and then setup the method to have 2 measurements per sample. The first measurement is the %T measurement made by inserting the sample in front of the reflectance sphere. The second is the %R measurement made by putting the sample on the reflectance sphere.

Create an equation called 'Absorptance' as

100 – Measurement1 – Measurement2

The result is a spectrum and this will appear as the spectrum for that sample in the measurements table replacing the 'mean' spectrum that normally appears. The description of the spectrum is the equation description.

NOTE: If you perform several equations that output a spectrum for the measurements table, only the last equation's results are displayed.

Conditional Formatting

Conditional formatting is part of the Equation settings and enables you to apply special formatting to the result when it meets certain criteria.

In this way you can highlight, for example, results that are below a certain limit.

The format for this is an "if...then...else" type of construction. For example "If the result of Equation 1 (**'If' condition**) is less than (**Operator**) 15 (**Criterion**) then turn the result red ('Then' conditional formatting) else leave it as it is ('Else' conditional formatting)".

How do I set up conditional formatting?

1. From the Processing page, select **Equation** as the type of process and click **Settings**.
The Settings dialog for Equations is displayed.
2. From the Settings dialog, select the Result Formatting tab.
3. Select the result you are going to apply the conditions to as the **'If' condition**.
The drop down list contains equations that have been constructed previously.
4. Select the required **Operator** from the drop down list.
5. Select the required **Criterion** for the condition to be met.
This can be a number or the result from another equation (selected from the drop down list).
If the Operator selected is Between or Not between, then Criterion 2 must be selected as well.
6. Select the **'Then' conditional formatting** to be applied if the criterion is met and select the font and color, if required.
Formatting options are discussed below.
7. Select the **'Else' conditional formatting** to be applied to results that do not meet the criterion, and select the font and color, if required.
The default is No formatting.

What Operators are available?

<>	Anything that does not equal the criterion
<=	Less than or equal to the criterion
>=	Greater than or equal to the criterion
<	Less than the criterion
>	Greater than the criterion
=	The same as the criterion
Not between	Does not lie between the two criteria
Between	Does lie between the two criteria

What formatting options are available?

The formatting options are:

- **No formatting** – the result is displayed just as it is.
- **Replace result with text** – the result will not be displayed, the text entered in the field will be displayed instead.
- **Add text to result** – the result will be displayed followed by the text entered in the field.
- **Only change font** – the result is displayed as calculated but in a different font as specified here.

To set the formatting:

1. Select the required formatting type.
2. If the type was **Replace result with text** or **Add text to result**, enter the text in the field below.
3. To change the font, click **Change Font**, and then use the dialog displayed to set the font required.
The current font settings are displayed under Current format.
4. Select the **Fill color** to be applied to the cell in the table.
5. Select **Apply to row** if you want the font formatting and fill color to be applied to the whole row in the table rather than just the cell with the result in it.

Arithmetic

Arithmetic is a processing command.

Arithmetic is the application of an arithmetic function on a spectrum. It can be the application of a constant (for example, multiply the spectrum's ordinate by a factor of 2) or spectral (for example, divide one spectrum by another spectrum).

How do I perform arithmetic on all samples?

1. Select the operation to be performed from the drop-down list.
2. If +, -, x or / is selected, select **Import** for a previous spectrum to use in the calculation or enter a value to be used.

For example, Current Sample x 2 will multiply the ordinate values of the current spectrum by a factor of 2.

NOTE: The input field and **Import** button are not available when **log**, **ln**, **sqr**, or **sqrt** is selected as this operation is performed on the spectrum itself.

What arithmetic operations are available?

+	addition
-	subtraction
x	multiplication
/	division
log	log base 10
ln	log base e
sqr	square
sqrt	square root

NOTE: For logarithmic and square root functions, negative values are set to zero.

What happens when I perform an arithmetic calculations using spectra that have different units?

When adding subtracting or dividing two spectra, the resulting spectrum is the sum, difference or ratio (as appropriate) of the two original spectra and the ordinate axis is designated the units of the first spectrum.

Units of Spectrum 1	Units of Spectrum 2 (spectrum chosen using Import)	Units of Result
A	%T	A
%T	A	%T
%T	Egy	%T
Egy	%T	Egy
A	Egy	A
Egy	A	Egy

The following table shows the special cases that may arise and the units of the resulting spectrum:

Units of Spectrum 1	Units of Spectrum 2 (spectrum chosen using Browse)	Operation	Post-Treatment	Units of Result
Any	%T	divide	x100	Units of Spectrum 1
Any	%R	divide	x100	Units of Spectrum 1
Egy	Egy	divide	x100	%T
%T	%T	multiply	/100	%T
%T	%R	multiply	/100	Arbitrary
%R	%T	multiply	/100	Arbitrary
%R	%R	multiply	/100	%R

NOTE: Multiplying two spectra in %T produces a result where the value is divided by 100.
For example, $50\text{ \%T} \times 50\text{ \%T} = 25\text{ \%T}$

NOTE: Dividing two energy spectra produces a result in %T.

Adding Spectra

What happens when I add two spectra with the same units and the same range but different data intervals?

The resulting spectrum is the sum constructed from the smaller data interval.

What happens when I add two spectra with the same units and data intervals but different ranges?

The resulting spectrum is the sum of the overlapping regions only.

Subtracting Spectra

What happens when I subtract two spectra with the same units and data intervals but different ranges?

The resulting spectrum is the difference of the overlapping regions only.

What happens when I subtract two spectra with the same units and range but different data intervals?

The resulting spectrum is the difference constructed from the smaller data interval.

Multiplying Spectra

What happens when I multiply two spectra with the same units, data interval and range?

The resulting spectrum is the product of the two spectra, except when multiplying two spectra in %T.

NOTE: Multiplying two spectra in %T produces a result where the value is divided by 100.
For example, $50\text{ \%T} \times 50\text{ \%T} = 25\text{ \%T}$

What happens when I multiply two spectra with the same units and data intervals but different ranges?

The resulting spectrum is the product of the overlapping regions only.

NOTE: Multiplying two spectra in %T produces a result where the value is divided by 100.
For example, $50\text{ \%T} \times 50\text{ \%T} = 25\text{ \%T}$

What happens when I multiply two spectra with the same units and ranges but different data intervals?

The resulting spectrum is the product constructed from the smaller data interval.

NOTE: Multiplying two spectra in %T produces a result where the value is divided by 100.
For example, $50\text{ \%T} \times 50\text{ \%T} = 25\text{ \%T}$

Dividing Spectra

What happens when I divide two spectra with the same units, data interval and range?

The resulting spectrum is the ratio of the two spectra, except when dividing two spectra in Egy.

NOTE: Dividing two energy spectra produces a result in %T.

NOTE: Dividing any spectrum by a spectrum in %T, produces a result where the value is multiplied by 100.

What happens when I divide two spectra with the units and data intervals but different ranges?

The resulting spectrum is the ratio of the overlapping regions only.

NOTE: Dividing two energy spectra produces a result in %T.

NOTE: Dividing any spectrum by a spectrum in %T, produces a result where the value is multiplied by 100.

What happens when I divide two spectra with the same units and ranges but different data intervals?

The resulting spectrum is the ratio constructed from the smaller data interval.

NOTE: Dividing two energy spectra produces a result in %T.

NOTE: Dividing any spectrum by a spectrum in %T, produces a result where the value is multiplied by 100.

Convert X and Convert Y

Convert X and Convert Y are processing commands. Convert X allows you to convert the abscissa scale. Convert Y allows you to convert the ordinate scale.

Convert X

When would I use Convert X?

Spectra are commonly displayed with abscissa scales that may be linear either in wavelength or in frequency. Linear wavelength is preferred for the UV/Vis region, while a linear frequency scale is employed for most applications in the mid-IR region. In the near IR region, both kinds of scale are in common use. **Convert X** is used to change between linear wavelength and linear frequency.

What are the available Ablscissa units?

- Select Wavenumbers (cm^{-1}), Microns (μm), Angstrom (\AA), Nanometers (nm), Hours (h), Minutes (min) or Seconds (s).

Timedrives can be converted between hours, minutes and seconds.

What is the Data Interval?

The data interval, or digital resolution, refers to the abscissa spacing of data points in a spectrum.

A default **Data Interval** is displayed for the selected Ablscissa unit.

- To change the default value, enter a new value in the **Data Interval** field.

What are the relationships between data linear in wavenumber (cm^{-1}) and data linear in wavelength – micrometers (μm) or nanometers (nm)?

$$\text{nm} = 10^3 \times \mu\text{m}$$

$$\text{cm}^{-1} = 10^4 / (\mu\text{m})$$

$$\text{cm}^{-1} = 10^7 / (\text{nm})$$

Convert Y

When would I use Convert Y?

Spectra are typically generated in transmittance or reflectance. However, for work involving quantitative determinations, it is appropriate to work in units that can be directly related to concentrations. This involves a logarithmic conversion to units of absorbance or log(1/R). For diffuse reflectance measurements, an equivalent conversion is to Kubelka-Munk units.

What are the available Ordinate units?

Select Transmittance (%T), Absorbance (A), Kubelka-Munk (K-M), Reflectance (%R), or Log 1/R.

Spectra can be converted between **A** and **%T**, **%R** and **K-M**, **%R** and **Log 1/R**.

NOTE: The software treats data in %T and %R identically except for the purposes of labeling the ordinate axis.

How is Absorbance (A) related to Transmission (% T)?

Absorbance (A) is related to Transmission (%T) by the equation:

$$A = 2 - \log_{10} (\%T)$$

NOTE: The conversion is truncated for transmission values below 1×10^{-6} %T, which are converted to 8 absorbance.

How is Log (1/R) related to % reflection (% R)?

Log (1/R) is related to % reflection (%R) by the equation:

$$\log (1/R) = 2 - \log_{10} (\%R)$$

NOTE: The conversion is truncated for transmission values below 1×10^{-6} %R, which are converted to 8 absorbance.

How is intensity in Kubelka-Munk units (K-M) related to the measured reflectance of the sample (R)?

The K-M command converts a reflectance spectrum, R, to K-M units, using the relationship:

$$K-M = (1-R)^2 / 2R$$

where R is the ratio of the reflectance of the sample to that of a suitable reference material. Typically, reflectance spectra may be initially represented on a %R scale from 0 to 100. The ordinate value used in the Kubelka-Munk calculation is equal to $0.01 \times \%R$. As R approaches zero, the results are truncated at approximately 0.7 %R to avoid division by zero.

What does Kubelka-Munk do to a spectrum?

Converts a spectrum with reflectance as its ordinate to Kubelka-Munk units.

When should I use a Kubelka-Munk scale?

In theory, after K-M (Kubelka-Munk) transformation, the band intensities vary linearly with concentration (obey Beer's Law). So, you would use K-M for diffuse reflectance spectra just as you would use absorbance for spectra measured in transmittance. The K-M theory is strictly applicable only to very weak absorption.

NOTE: Performing the Kubelka-Munk transform on a spectrum that is already in K-M units will convert it to Reflectance units.

What does Log(1/R) do to the data?

Log(1/R) converts spectral data from %T or %R to Log(1/R).

What is Reflectance (% R)?

Reflectance (%R) = Reflectance / 100

Derivative

Derivative is a processing command.

Derivative spectra are used for two main reasons:

- the identification of band centers where bands are not well separated;
- to reduce the influence of broad features, typically those associated with the wings of neighbouring bands, in quantitative analysis.

Derivative applies a 1st–4th order derivative function to the spectrum. A fixed width can be applied, where the width is the number of data points (of the original spectrum) over which the derivative calculation is applied. The amplitude of a feature in a derivative spectrum should be proportional to the amplitude of the original band.

What is Derivative?

Derivative uses the Savitzky-Golay procedure to estimate the derivative of a smooth curve, constructed through the original data points of your original spectrum. It uses a number of neighboring data points to estimate the curve. As the number of data points used in the calculation is increased, the contributions of broader features increase relative to narrow features.

Derivative spectra have both negative and positive features, which make them difficult to interpret.

Derivative spectra emphasize narrow features, including noise, relative to broad ones. You can control this to some extent by varying the width of the derivative function. This is equivalent to applying some smoothing to the derivative spectrum.

What are the properties of the derivatives?

First derivative – the first derivative is zero at the band center and has single sidelobes of equal amplitude, one positive and one negative on either side.

Second derivative – the second derivative has a minimum at the center of the band and single maxima of equal amplitude on either side.

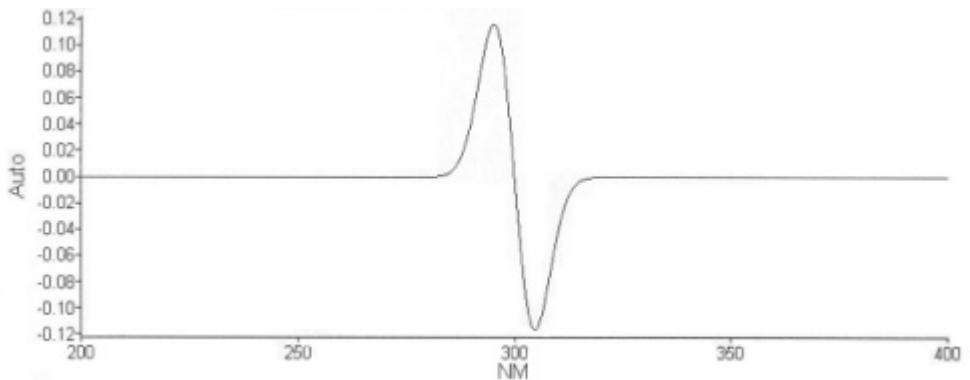
Third derivative – the third derivative is zero at the band center and has two sidelobes, one positive and one negative, on either side. The corresponding sidelobes on the two sides are of opposite sign.

Fourth derivative – the fourth derivative has a maximum at the band center and has two sidelobes on either side. The first sidelobes are negative and the second are positive.

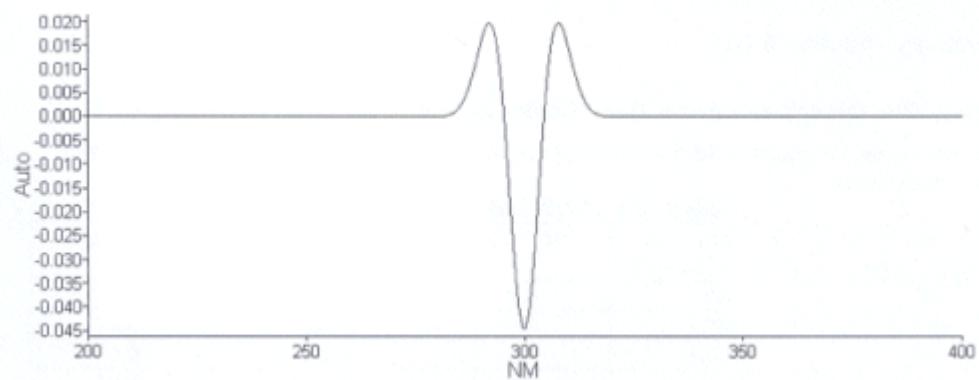
Original spectrum



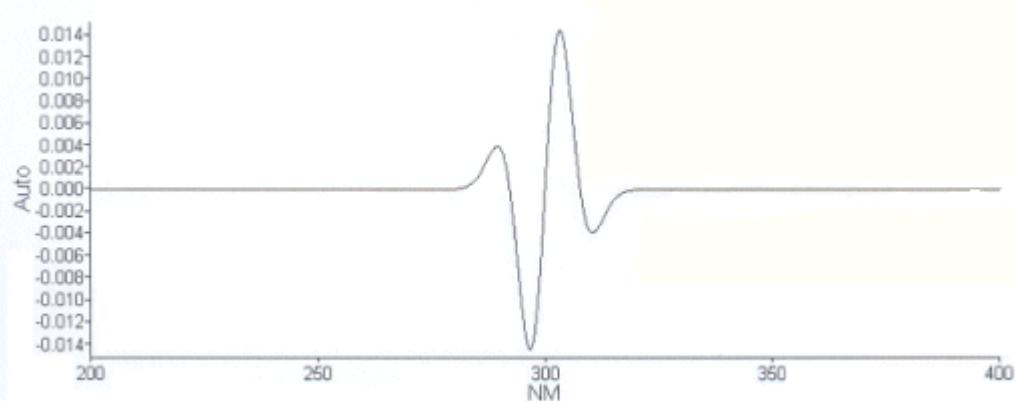
First Derivative



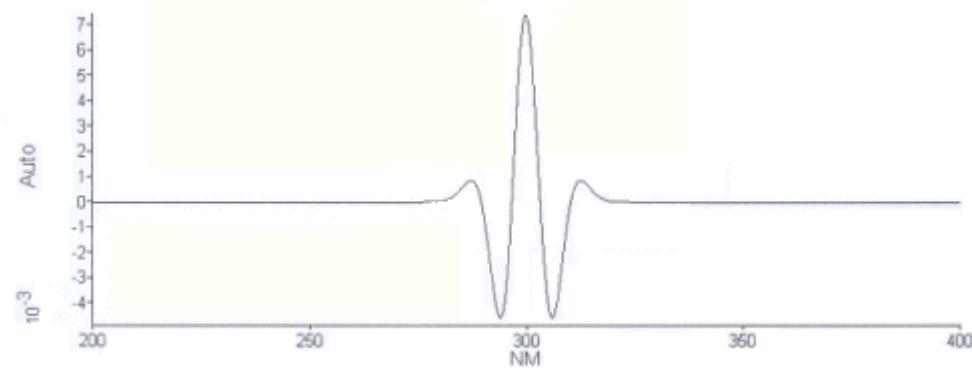
Second Derivative



Third Derivative



Fourth Derivative



When should I use Derivative?

Derivative spectra usually have sharper features than the original spectra. In quantitative analysis, they are sometimes used to reduce the effects of overlapping bands. You can use the amplitudes of features in derivative spectra in the same way as peak absorbances. The elimination of some baseline effects in derivative spectra can be useful in quantitative methods.

You can use the Derivative command to take the first, second, third or fourth order derivative of a spectrum.

The first derivative removes any baseline offset and the second derivative removes a linear slope.

Second-derivative spectra have sharp minima where there are maxima in the original spectrum and so can be used to identify band positions in complex regions.

What do the parameters do?

The **Width** sets the number of data points used to calculate the derivative. As you increase the width used in the calculation, the contributions of broader features increase relative to narrow features. The use of derivative can increase noise, so using a broader width can help to minimize this.

What are the default parameters for Derivative and how do I change them?

By default, when Derivative is selected the defaults are **First order** and **Width 9 point**.

To change the defaults:

1. Click Settings.
The Derivative tab of the Settings dialog is displayed.
2. Select the **Order** and **Width** and then click **OK**.
The dialog closes and the settings in the table are updated.

Difference

Difference is a processing command.

Difference subtracts one spectrum from another with an applied scaling factor. The factor can be selected manually or applied automatically.

Examples:

- Two samples of the same material have two different dilutions. When a difference is performed using an automatic scaling factor, the ideal result is a flat baseline. The scaling factor needed to achieve this is representative of the difference in dilution of the Absorbance.
- A sample spectrum containing a known and unknown can have a standard spectrum of a different dilution subtracted from it. Difference using Automatic attempts to remove the standard spectrum completely with the use of a scaling factor to leave just the unknown spectrum.

NOTE: A %T subtraction is performed in absorbance.

How do I set up a Difference?

1. Click **Import** to display the Import Data dialog, and select the spectrum to be subtracted.
2. Select **Automatic** or **Manual** factor.
Automatic will automatically calculate the factor to be used. Manual allows you to enter the factor to be used. The factor corrects for the difference in baseline between the two spectra.
3. If **Manual** is selected, enter the factor to be used.

NOTE: A scaling factor of 1 performs a straight forward subtraction of one spectrum from another.

How is difference calculated?

The difference between two spectra is calculated as:

$$\text{Difference} = (\text{Spectrum 1}) - (\text{Spectrum 2} * \text{Factor})$$

What is the difference between Automatic and Manual Factor?

If **Automatic** Factor is selected, the software automatically calculates the best scaling factor to apply to correct for the difference in baseline between the two spectra. If **Manual** Factor is selected, you must enter a value to apply. By default, **Automatic** is selected.

Interpolate

Interpolate is a processing command.

Interpolate is used to change the number of data points that are used to represent a spectrum. **Interpolate** can result in a spectrum with more or less data points than the raw spectrum.

What is Interpolation?

Interpolation adds points to a spectrum by adding new points between the points that already exist. Interpolation removes points from a spectrum by selecting every nth point of a spectrum and ignoring it.

The number of data points needed in a spectrum depends on the width of the features in the spectrum. If the separation of the data points is smaller than that needed to define the features in the spectrum, the number of points can be reduced without reducing the amount of information in the spectrum.

Increasing the number of data points in a spectrum cannot increase the amount of information in the spectrum. However, it can increase the definition of the spectrum.

What does Interpolate do to a spectrum?

Interpolate copies a spectrum, and changes the number of points used to plot the spectrum.

When should I use Interpolate?

You can use Interpolate to:

- give incompatible spectra the same data interval
- save part of a spectrum
- reduce the number of data points in a file to save storage space, or to reduce the time needed to process or print it.

How do I define the settings for Interpolation?

To change the values:

1. Click Settings.
The Interpolate dialog is displayed.
2. Enter the **Start** and **End** values for the interpolation range.
3. Enter the **Data Interval** and click **OK**.
The Interpolate dialog closes and the settings in the table are updated.

Normalize

Normalize is a processing command.

Normalize allows spectra to be scaled so that the absorbance of a chosen band or at a chosen position has a specified value, thus compensating for differences in concentration or pathlength. The band is selected by choosing a frequency range within which it represents the maximum absorbance. A baseline offset can be applied so that either absorbance at a chosen frequency, or the minimum absorbance in the spectrum, is set to zero.

What can I use Normalize for?

Normalize enables you to compare spectra of different ordinate amplitude.

Its main use is for setting a common peak in several spectra to the same ordinate limit. Other peaks in these spectra can then be compared. All calculations are performed in absorbance.

What does Normalize do to a spectrum?

Normalize multiplies each point in a spectrum by a factor.

What do the parameters do?

The parameters enable you to choose:

- the ordinate value to which you want the spectrum or spectra to be scaled
- the abscissa point (or range) at which this ordinate value is to be set
- whether you want the baseline to be adjusted to zero and, if so
- whether you want this adjustment to be automatic; or to be at a point you select.

How do I change the settings?

To change the settings:

1. Click Settings.
The Normalize dialog is displayed.
2. Enter the ordinate value to **Normalize to** (in A).
3. Select to normalize on Maximum Ordinate value or Selected Abscissa point.
4. If you select **Maximum Ordinate value**, enter the **Start** and **End** value for the range.

OR

If you select **Selected Abscissa point**, enter the abscissa point at which the spectrum will be normalized.

5. Select Zero point **Off**, **Autozero**, or **Manual zero at**.

Selecting Off means that the baseline is not zeroed. Autozero automatically zeroes the baseline from the lowest value and Manual zero at enables you to choose a point to zero at.

6. If you select **Manual zero at**, enter the value to zero the baseline at.

What is zero point?

Zero point defines the point to zero the baseline at. There are three options:

Off	The baseline is not zeroed.
Autozero	Autozero automatically zeroes the baseline from the lowest abscissa value at a data point in the original spectrum.
Manual zero at	Manual zero at enables you to choose a point to zero at.

- If you select **Manual zero at**, enter the ordinate position value that will be set to zero.

Peak Table

The positions and intensities of peaks above a specified threshold, and bases can be obtained using a Peak Table. For spectra in ordinate units of A or log (1/R), a peak is defined as a point of maximum intensity, with a corresponding definition for bases. For spectra in %T, %R or Energy units, peaks are defined as intensity minima. A threshold is applied to distinguish genuine peaks from local maxima associated with noise.

Positions are reported either as values obtained by interpolation or as the coordinates of the data points. By default, the values obtained by interpolation are reported.

What settings are available for the Peak Table?

Peak Table tab

Threshold	Enter a value for the threshold. Peaks above this threshold are included in the peak table. The allowed threshold is 0.0001–100 %T , 0.00001–10 A.
Start, End (nm)	Enter a Start and End value for the range over which the peaks are identified. The range must be within the scan range.
Peaks, Bases	Select whether to find Peaks and / or Bases .
All peaks / Only the top X peaks	Select whether to find all peaks or only a certain number of peaks. If only the top X peaks is selected, you must enter the number of peaks to be found. The maximum is 199 and the minimum is 1. The default is 10.
Return actual data point	The closest data point to the maxima (or minima) is reported. Interpolation is not used.

Formatting tab

Change Font	Click Change Font to display a standard Windows font selector dialog.
Abscissa	Select Significant figures or Decimal places , and then select the number from the appropriate drop-down list.
Ordinate	Select Significant figures or Decimal places , and then select the number from the appropriate drop-down list.

Over what range are the peaks calculated?

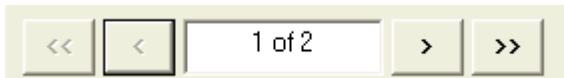
Enter a **Start** and **End** value for the range over which the peaks are calculated.

Units are in nm for collected data. If only imported data is available, the units of the imported data are used.

How are the peak table results displayed?

The results are displayed on the Peaks tab.

The buttons shown below are used to navigate through all the samples that were run. Hover the mouse over the buttons to find out what they do.



The table below shows an example of the results displayed.

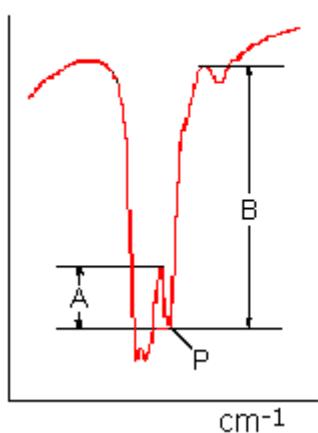
Graphs		Results		Peaks					
<<	<	1 of 2	>	>>					
		SampleID:	Holmium.Sample						
		Description:							
		Threshold:	0.1						
		Range Start nm	800						
		Range End nm	300						
		Find:	Peaks						
		Display:	List by position						
<hr/>									
	Position nm	Intensity A	Type						
1	637.64	0.1819	Peak						
2	536.4	0.3613	Peak						
3	459.92	1.101	Peak						
4	453.16	0.969	Peak						
5	445.68	2.164	Peak						
6	418.39	0.2377	Peak						
7	360.62	0.7859	Peak						

What is the threshold?

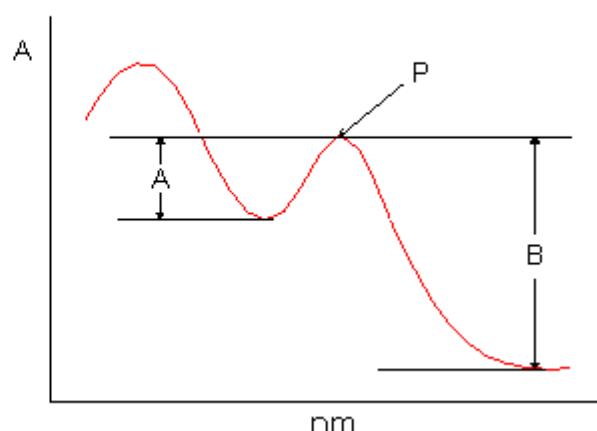
The threshold defines which peaks are included in the peak table. It is measured in units on the ordinate scale. If the peak height is less than the threshold, it is not included in the peak table.

The peak height is the smallest difference in ordinate units between the peak and the bases on either side of it. In the following diagrams, P is listed as a peak, because both A and B are greater than the threshold.

A transmission spectrum



An absorbance spectrum



Smooth

Smooth is a processing command.

Smooth applies one of three algorithms to the spectrum to make spectral features more visible by reducing noise. There are three types of Smooth available: Block Average, Triangular, and Savitsky-Golay (cubic).

What is smoothing?

Smoothing is a form of filtering.

The ordinate value of each data point is replaced by a weighted average of the ordinate values of the data points in a smoothing window around that point. Increasing the width of this smoothing window leads to greater noise reduction but broadens the bands in the spectrum.

The weighting coefficients are generally chosen to minimize this broadening. The width of the smoothing window is usually selected by visual inspection of the smoothed data. Smoothing should not change band areas or the positions of symmetrical bands.

What types of smoothing are available?

Block Average Smoothing – all points are weighted equally, giving the maximum noise reduction for a chosen window.

Savitsky-Golay Smoothing – aims for minimum distortion by least squares fitting a cubic polynomial.

Triangular Smoothing – the weighting coefficients decrease linearly from the center of the smoothing window, causing less broadening than Block Averaging but achieving less noise reduction.

NOTE: Data less than half the window width from the ends of the spectra are treated differently. The data are extended by a double reflection about the end of the spectrum and the smoothing function is applied to the real and reflected data. This allows the smoothed data to extend to the limits of the original range with no discontinuity.

NOTE: For reasons of compatibility with earlier software, the number of points used in the Triangular smoothing differs from that quoted. The relationship between these numbers is shown below. For widths of 13 and above the number used is approximately 2/3 of the number specified.

Number specified	5	9	13	19	25	37	49	149
Number used	3	7	9	13	17	25	33	99

What are the default smoothing parameters and how do I change them?

The default is a **Block Average** smooth with a **9 point** smoothing width.

To change the settings:

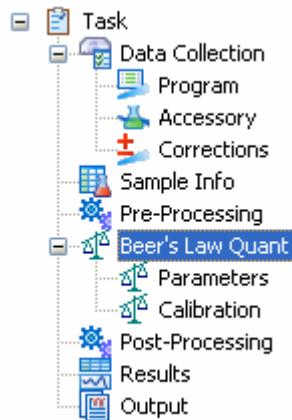
1. Click Settings.
The Smooth dialog is displayed.
2. Select the **Type of smoothing** from the drop-down list.
3. Select the Smoothing width.

What does Smooth do to a spectrum?

Smooth reduces the noise level of your spectrum, but it also degrades the resolution of your spectrum and features in the spectrum become broader.

The amount of noise in your spectrum is related to the number of scans collected. If you collect more scans, the less the noise compared with the signal.

Quant



Quant (Quantitation) is the process of determining the unknown concentration of a component in a sample. The process assumes a relationship between the absorbance of the component and the concentration. This can be based on the absorbance at a specific wavelength (Wavelength Quant) or on a peak height or area (Scanning Quant).

Data from standards or standard replicates of known concentration is collected and a calibration curve (Absorbance against Concentration) is generated. The quality of a calibration curve is normally checked by assessing the fit of the concentrations of the calibration standards or by analyzing control samples of known composition. This calibration curve is then used to determine the concentration in unknown samples.

There are three pages to Beer's Law Quant:

Beer's Law Quant – this page determines the type of calibration and the limits to be applied to the calibration, as detailed below.

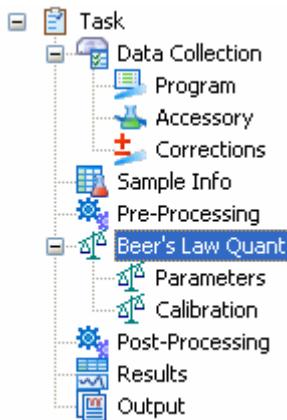
Parameters – this page is used to set up the actual calculation. It differs depending on whether the Method is a Wavelength quant or a Scanning quant.

Calibration – this page displays the Standards Table and the calibration details if the calibration type selected was Calibration curve or Single standard.

See also

[Results](#)

Beer's Law Quant



What parameters are defined on the Beer's Law Quant page?

Component	Enter a name for the column. This column will appear in the Results Table.
Units	Enter the Units that will also be displayed in the column header. The units will be displayed as Name (units).
Calibration	Select the type of Calibration to be used. Select Calibration curve , Single standard , User factor , or User defined curve from the drop-down list. NOTE: The other details on the page update depending on the type of Calibration selected.
Type of curve	Select the type of fit for the calibration curve. Select Linear , Cubic or Quadratic from the drop-down list.
Force through zero	Select Force through zero to include the origin as a point when creating the calibration curve.
Force recalibration	Select Force recalibration to force a standard (or standards) to be run either on startup or after the specified period has elapsed before the unknown samples can be run. NOTE: Force recalibration is only available when calibration curve or single standard is selected.
Correlation (r^2)	Calculates the correlation factor for the calibration and compares it to the limit set. NOTE: This is not available when Single standard calibration, User factor or User defined curve is selected.
Standard tolerance (%)	The software checks each standard against the tolerance set. NOTE: This is not available when User factor or User defined curve is selected.

Control samples tolerance (%)	The software checks each control sample against the tolerance defined.
Allow 10% extrapolation	<p>Enables you to exceed the range covered by your standards by 10%.</p> <p>NOTE: This is not available when Single Standard, User factor or User defined curve is selected.</p>
Proceed on error	<p>Select Proceed on error to display a warning when one of the tolerances is exceeded. When Proceed on error is selected, you are allowed to proceed with running the sample even if the tolerance (tolerances) is exceeded.</p> <p>NOTE: If Proceed on error is selected and the analysis is then performed, any error messages are displayed next to the predicted concentration value. If Proceed on error is not selected, then an 'unsuccessful calibration' message is displayed in the predicted concentration field (and residual if appropriate).</p>

Who can setup and run a Quant calibration?

In Enhanced Security:

- Users with Create and edit methods and IPV set-ups permission can setup Quant.
- Users with Run calibration permission and Run methods permission can run and save a calibration. Users with Edit calibration permission can modify and save calibrations.

In the Standard version of UV WinLab:

- The Analyst has permission to setup and run Quant calibrations.

Calibration

What types of calibration are available?

Calibration curve

Selecting **Calibration curve** means that the curve is calculated from the standards entered in the Standards table.

Single standard

Selecting **Single standard** means that the curve is calculated from a single standard and a fit through zero. Therefore, the Standards table only has one row.

When **Single standard** is selected, **Correlation limit** is not available.

User factor

Selecting **User factor** means that the user enters a factor instead of creating a curve. The **Factor** is the slope of the curve and it assumes an intercept through zero. There is no Standards table.

The calibration equation is Absorbance = Factor x Concentration

- Enter the required **Factor** (concentration per unit absorbance).

When **User factor** is selected, only **Control samples tolerance** and **Proceed on error** are available in the Limits section of the page.

User defined curve

Selecting **User defined curve** means that the user enters a slope and intercept instead of creating a curve from standards. There is no Standards table.

The calibration equation is Absorbance = Slope x Concentration + Intercept

- Enter values for the **Slope** and **Intercept**.

NOTE: The **Slope** and **Intercept** fields only appear when **User defined** is selected.

When User defined curve is selected, only **Control samples tolerance** and **Proceed on error** are available in the Limits section of the page.

What types of curve are available?

The options for the curve are **linear**, **quadratic** and **cubic**.

Linear $A = k_1 + k_2C$

Quadratic $A = k_1 + k_2C + k_3C^2$

Cubic $A = k_1 + k_2C + k_3C^3$

where A is the absorbance, C the concentration, and k_1 , k_2 and k_3 are constants. k_1 can be set to zero by selecting **Force through zero**.

The constants are determined by least squares fitting between the specified concentrations and measured absorbance values for a set of standards.

What is Force recalibration?

Selecting **Force recalibration** enables you to define the next time the calibration needs to be run:

On Start-up – this means that recalibration is needed as soon as the method starts.

After X days – sets the number of days after which re-calibration must be performed again.

A message will be displayed when the recalibration is due.

NOTE: **Force recalibration** is only available when calibration curve or single standard is selected.

Limits

What are the different limits for?

Correlation, Standard tolerance and Control samples tolerance allow you to define the acceptance criteria for the calibration. **Allow 10% extrapolation** allows the calculation of the unknown beyond the highest or lowest standard in the calibration by up to 10%. One or more of these options can be selected.

How do I use the Correlation limit?

When a calibration is performed the correlation factor for the calibration will be calculated and compared to the limit set here. If the calculated correlation is less than the value set here, a warning message will be displayed, and the calibration is said to have failed.

- Enter a value between 0 and 1 for the **Correlation** limit to be compared against.

NOTE: This is not available when Single standard calibration, User factor or User defined curve is selected.

How do I use Standard tolerance?

The software calculates the difference between the specified value and the calculated value for each standard and then compares that result to the **Standard tolerance** set here. If the calculated value is not within the set tolerance from the specified value then a warning is displayed.

- Enter a value between 0.1 and 10 for the **Standard tolerance (%)**.

How do I use Control samples tolerance?

The software calculates the difference between the specified value and the calculated value for each control sample and then compares that result to the **Control samples tolerance** set here. If the calculated value is not within the set tolerance from the specified value then a warning is displayed.

- Enter a value between 0.1 and 10 for the **Control samples tolerance (%)**.

NOTE: If **Proceed on error** is switched off then this warning will stop any further samples being analyzed.

How does Allow 10% extrapolation work?

When Allow 10% extrapolation is switched on, the concentration of a sample can be outside the range the calibration standards by up to 10% and still be reported. When switched off, if a sample concentration is calculated that falls below the concentration of the weakest standard, or above the concentration of the strongest standard, a warning message will be displayed.

NOTE: If **Proceed on error** is switched off then this warning will stop any further samples being analyzed.

How does Proceed on error work?

Normally, if a control limit is exceeded a warning message will be displayed and the analysis will be stopped. Selecting **Proceed on error** will enable following samples to still be run.

Proceed on error also controls the use of invalid calibrations. If the calibration is unsuccessful and **Proceed on error** is not selected, data collection of the samples will not be allowed. If the calibration is successful or the calibration is unsuccessful and **Proceed on error** is selected, data collection of the samples will proceed and the outcome of the calibration is not reported until all the samples have been run.

See also

Quant

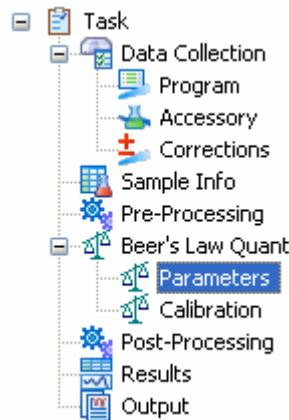
Wavelength Quant Parameters

Scanning Quant Parameters

Quant Calibration

Results

Scanning Quant Parameters



The parameters for a Scanning Quant method are used to set up whether the calculation is to be based on Height, Maximum Height (looks for the maximum height within a range), or Area. It also enables you to decide whether to include baseline correction.

What Ordinate modes are available?

- Select Height, Maximum Height or Area.
Height calculates the peak height at a specific point. Maximum Height calculates the maximum peak height within a specified range. Area calculates the peak area.

Height

- Enter the **Wavelength** at which the **Height** is to be calculated.

Maximum Height

- Enter the **Start** and **End** values for the range within which the **Maximum Height** is calculated.

Area

- Enter the **Start** and **End** values for the range over which the **Area** is calculated.

What types of Baseline correction are available?

- Select None, Single point, or Two points.

None

None uses the automatic baseline.

Single point

The absorbance at the specified wavelength is subtracted from the original values.

- Enter the wavelength in the **Base 1** field to be used for the Baseline correction.

Two points

Linear interpolation between the absorbance values at two specified wavelengths is used to determine a value to be subtracted from the original absorbance value at each wavelength.

- Enter the first wavelength to be used for the Baseline correction in the **Base 1** field, and the second wavelength in the **Base 2** field.

NOTE: If you have browsed a spectrum into the graph display you can use the markers displayed to set **Base 1** (and **Base 2**). When you move the marker(s), the value(s) in the Base field(s) update(s) accordingly.

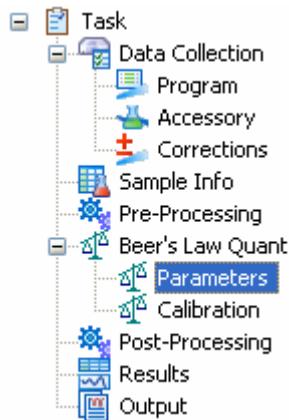
Can I display a spectrum to help me set the points for the calculation?

Yes.

1. Click **Browse**, under the graph.
A file selector dialog is displayed.
2. Select the spectrum to be displayed.
3. Click **Open**.
The spectrum is displayed on the graph.

4. Click  to switch on the cursor and use it to find the points required.

Wavelength Quant Parameters



The parameters for a Wavelength Quant method are used to set up the wavelength for the calculation and it also enables you to decide whether to include baseline correction.

The wavelength for the calculation is selected from the drop down list of wavelengths set up on the Data Collection page.

How do I use Baseline correction?

Baseline correction is used here by selecting one or two points where the absorbance is read and taken as a baseline, and the graph updated accordingly.

The points used can only be wavelengths set up on the Data Collection page.

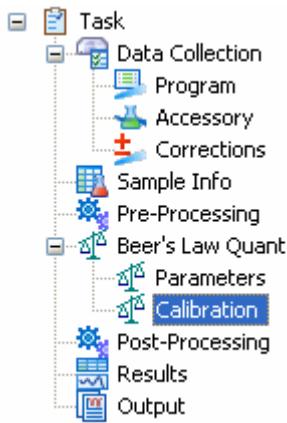
How do I export the information displayed in the table at the bottom of the page when I am working with a Wavelength Task?

The data in the table can be copied to the clipboard.

- Click the right mouse button anywhere in the table and select **Copy To Clipboard**. All the data in the table is copied to the clipboard. It can then be directly pasted into another application such as Excel.

NOTE: If you select certain cells in the table before selecting **Copy To Clipboard**, only the selected cells are copied to the clipboard.

Quant Calibration



This page is only available if the calibration type is **Calibration curve** or **Single standard**, as set up on the Beer's Law Quant page, and it enables you to set up the standards to be used and review the calibration details after the standards have been run.

The calibration curve is updated after the last standard has been run. You can choose to exclude standards (from the calibration graph) that are in error and see the results updated. Standards can be excluded in the Enhanced Security version of UV WinLab; they can be deleted in the Standard version of UV WinLab.

Setting up the Standards Table

How do I add a standard to the Standards Table?

You can not add a new standard directly to the Standard Table.

All measurements, including standards, are entered in the Sample Table. Once added to the Sample Table and the type selected as Standard, a sample will be visible in the Standards Table. A standard can be edited in the Standards Table.

Should I add new standards before or after formatting the Standards Table?

It does not matter whether you add standards to the Sample Table (which will automatically update the Standard Table) before or after formatting the Standard Table; the table will be updated accordingly. If more than one replicate is specified, the correct number of replicates are added to or removed from the table when the number of standards is changed. For example, if you specify three replicates per standard and increase the number of standards by 2, then 6 entries are added to the table.

NOTE: If you are a Developer setting up a method for use by an Analyst, you can format the table but not define the number of standards, as this can be done by the Analyst at run time (and may vary each time the method is used). However, if you wish, you can save a method with a defined number of samples and standards so that the Analyst simply has to run the method without having to enter this information.

How do I insert a standard at a particular point in the Standards Table?

After you have defined the number of standards, it is possible to add standards if you have forgotten to do so.

1. Click in the Sample Table row where you want to add the standard.

NOTE: The new row is added above the selected row.

2. Click **Insert**.

The new sample is added above the selected row.

3. Change the Type to Standard.

The standard appears in the Standards Table.

How do I import a standard into the Standards Table?

Standards can be imported into a method from another task or from another file. You need to import the standards into the Sample Table. They will then automatically appear in the Standards Table.

NOTE: When importing data, data are always opened as standards. Replicates are not applicable.



1. Click .

The Browse dialog is displayed.

2. Select the standard to import from a file or database.

3. Click **Open**.

The standard is imported into the Sample Table. A green tick appears in the left column of the table to show that the imported standard has previously been run.

How do I delete a standard from the Standards Table?

You cannot delete a standard from the Standards Table directly. You need to delete the standard from the Sample Table.

If data has not been collected:

1. Click in any cell of the row of the Sample Table that contains the standard that you wish to delete.

A dotted line appears around the cell to show that it is selected:

	Sample ID	Description	Type
1	Sample1	Supplier ABC12	Measurement 1
2	Sample1	Supplier ABC12	Measurement 2
3	Sample2	Supplier ABC12	Measurement 1
4	Sample2	Supplier ABC12	Measurement 2

2. Click **Delete**.

The standard is deleted from the Standard Table. When a replicate is deleted, the remaining replicates are renamed and ordered accordingly.

NOTE: If replicates are defined, Replicate 1 must be selected to delete the entire standard. Otherwise, only the selected replicate is deleted.

NOTE: If you are using the Enhanced Security version, replicates cannot be deleted.

If data has not been collected, the standard ID of a deleted standard can be re-used.

If data has been collected:

If you are using the Standard Security version of UV WinLab, you are informed that the calibration will be invalidated.

If you are using the Enhanced Security version of UV WinLab, you are prompted to confirm the deletion and provide a reason to be recorded with the standard. This information together with the User name and the Date/Time is recorded with the standard data. The standard is marked as Excluded (a cross is displayed in the first column of the Standard table and the row is grayed). The behaviour is the same for replicates.

How do I edit a standard description?

1. Click in the **Description** cell that you wish to edit.
A dotted line appears around the cell to show that it is selected.
2. Enter a description of the standard.
Click outside of the cell when you have finished.

How do I add a comment to a standard?

1. Select the sample to which you wish to add a comment.
2. Right-click and from the menu select **Add Comment**.
The Add Comment dialog is displayed.
3. Enter the text and then click **OK**.

NOTE: A comment can only be added to a sample via the **Add Comment** menu item that is displayed when you right-click on a sample.

The comment can be viewed in the Sample Event Log – right-click on the sample and select **View Sample Event Log**.

In the Enhanced Security version of UV WinLab, the Sample Event Log records the Event, Date and Time, User, and Reason/Comment.

It can also be viewed via the results of a Query. Select the result from the table and then select **Signatures and Comments** from the Tree. Any comments added to the sample are listed.

What menu items are available when I right-click on a row in the Standards Table?

The following menu items are available:

View Sample Event Log	Displays the Sample Event Log.
Add Comment	Allows you to add a comment to a sample.
Exclude	Allows you to exclude a standard so it is not used to build the calibration graph.
NOTE: Exclude is only available in the Enhanced Security version of UV WinLab.	

What is the Sample Event Log?

The Sample Event Log records all events connected with the standard. For example, if a description is added after the standard has been run, this event is recorded in the Event Log. The original value and new value are recorded.

The Sample Event Log records the event, time, user, and reason/comment.

NOTE: If a field has been changed from being empty to containing a value, empty quotes "" are used to show that the field was initially empty.

How do I format the Standards Table?

Formatting the Standard Table allows you to define the exact content and layout of the Standards Table for a particular task. You cannot format the table if the Method has been locked, if you do not have the necessary permission (within the Enhanced Security version of UV WinLab), or if the standards have been run. See Security Settings for further information.

1. Click Format Standards Table.
The Table Builder dialog is displayed.
2. Edit the details as required.
Each of the four tabs are explained in turn below.

Preparation

NOTE: The types of Preparation available depend on the method selected.
--

Select the type of standard preparation to be used. Columns will be added to the table depending on the type of standard preparation selected:

Concentration	Only the Concentration column is available in the Table.
Dissolved solid	Adds columns called Weight and Volume to the Table along with Concentration .
Dissolved solid and dilution	Adds columns called Weight , Volume , Initial Volume and Final Volume to the Table along with Concentration .

NOTE: For **Dissolved solid** and **Dissolved solid and dilution**, the concentration fields are automatically completed when values are entered in the other columns. The Concentration fields are non-editable in these cases.

Set preparation volume	This is only available when Dissolved solid or Dissolved solid and dilution is selected. Enter the volume used (ml). This value will appear in the Volume column of the table when the number of Standards has been specified.
Set volume taken for dilution	This is only available when Dissolved solid and dilution is selected. Enter the volume used (ml). This value will appear in the Initial Volume column of the table when the number of Standards has been specified.
Set final volume	This is only available when Dissolved solid and dilution is selected. Enter the volume used (ml). This value will appear in the Final Volume column of the table when the number of Standards has been specified.

NOTE: When a value is entered for **Set preparation volume**, **Set volume taken for dilution**, or **Set final volume**, it applies to all standards in the table. However, you can edit these values. Alternatively, you can de-select these options and manually enter any values required.

This information does not have to be entered in the table unless **Table must be completed before run** is selected, in which case this information must be entered by the Analyst before running a standard.

Design

Select whether you wish to include replicates or measurements. If you do, a column will be added to the table.

Replicates or Measurements per standard	Select Replicates or Measurements from the drop-down list. Enter the number of replicates per standard or measurements per standard . The default is one. If value >1 then each cell in the Type column has a drop down list which includes Replicate 1 to Replicate n where n is the number of replicates selected.
Sequence by standard	Replicates – Select Sequence by standard to run the standards in standard order rather than replicate order. For example, if you had 2 standards (S1 and S2) and each standard had 2 replicates (R1 and R2), the standards would be run in the order: S1R1, S1R2, S2R1, S2R2. Measurements – Select Sequence by standard to run the standards in standard order rather than measurement order. For example, if you had 2 standards (S1 and S2) and each standard had 2 measurements (M1 and M2), the standards would be run in the order: S1M1, S1M2, S2M1, S2M2.

<p>Sequence by replicate or Sequence by measurement</p>	<p>Replicates – Select Sequence by replicate to run the standards in replicate order rather than standard order. For example, if you had 2 standards (S1 and S2) and each standard had 2 replicates (R1 and R2), the standards would be run in the order: S1R1, S2R1, S1R2, S2R2.</p> <p>Measurements – Select Sequence by measurement to run the standards in measurement order rather than standard order. For example, if you had 2 standards (S1 and S2) and each standard had 2 measurements (M1 and M2), the standards would be run in the order: S1M1, S2M1, S1M2, S2M2.</p>
---	---

NOTE: The order of the standards in the table is the order in which they will be run.

Columns

Select the columns to appear in the Standard Table and the order of these columns.

Add	<p>Displays the Column type dialog which enables you to add a custom column to the table. See Custom Columns</p>
Move Up	<p>Moves the selected column one space up. This means the column is moved one space to the left in the Table.</p> <p>NOTE: Move Up is only available if a column is selected (a dotted line appears around the column name when it is selected).</p>
Move Down	<p>Moves the selected column one space down. This means the column is moved one space to the right in the Table.</p> <p>NOTE: Move Down is only available if a column is selected (a dotted line appears around the column name when it is selected).</p>
Format	<p>Displays a Format Column dialog depending upon the type of column selected. See Formatting Columns</p>

ID

User must enter	<p>When User must enter is selected, the user must fill in the Standard ID column of the Standard Table for each standard.</p> <p>NOTE: If replicates are included, when you enter the name for the first replicate the name of the other replicates for the standard are automatically filled in.</p>
Use this format	<p>When Use this format is selected, the Sample ID column of the Sample Table is automatically filled in by incrementing the prefix which you should enter in the field below. For example, if you enter the prefix Trial, the standards will be numbered Trial1 to Trialn (where n is the number of standards). The numbering is remembered across all tasks created by this method.</p>

- When you have selected all the required settings, click **OK**.

The Table Builder dialog closes and the Standard Table is populated using the chosen settings.

NOTE: The Sample Table can only be formatted using the **Format Standards Table** button. The button is not available if the Method has been locked, if the standards have been run, or if you do not have the correct permission (within the Enhanced Security version of UV WinLab).

NOTE: Permissions are defined by the UV WinLab Administrator. Please contact them for further information about your permissions.

NOTE: Setting up the Standards Table is very similar to setting up the Sample Table used on the Sample Info page. For further details, for example on adding new columns, formatting columns, and how the table can be used once it is created, see the Sample Info help.

Measurements and replicates

How do I add replicates to the Standards Table?

1. Enter the number of **Standards** to be run.
2. Click Format Standard Table.
The Organize Columns dialog is displayed.
3. Select the Design tab.
4. Select **Replicates** from the drop-down list.
5. Enter the number of replicates per standard.
6. Select whether to Sequence by standard or Sequence by replicate.
Select Sequence by standard to run the standards in standard order rather than replicate order. For example, if you had 2 standards (S1 and S2) and each standard had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the standards would be run in the order: S1R1, S1R2, S2R1, S2R2.
Select Sequence by replicate to run the standards in replicate order rather than standard order. For example, if you had 2 standards (S1 and S2) and each standard had 2 replicates (R1 and R2) (that is, S1R1, S1R2, S2R1, S2R2), the standards would be run in the order: S1R1, S2R1, S1R2, S2R2.
7. Click **OK**.
The replicates are added to the standard table.

NOTE: A maximum of 9 replicates can be added for each standard.

How do I add measurements to the Standards Table?

1. Enter the number of **Standards** to be run.
2. Click Format Standard Table.
The Organize Columns dialog is displayed.
3. Select the Design tab.
4. Select **Measurements** from the drop-down list.
5. Enter the number of measurements per standard.

6. Select whether to Sequence by standard or Sequence by measurement.

Select Sequence by standard to run the standards in standard order rather than measurement order. For example, if you had 2 standards (S1 and S2) and each standard had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the standards would be run in the order: S1M1, S1M2, S2M1, S2M2.

Select Sequence by measurement to run the standards in measurement order rather than standard order. For example, if you had 2 standards (S1 and S2) and each standard had 2 measurements (M1 and M2) (that is, S1M1, S1M2, S2M1, S2M2), the standards would be run in the order: S1R1, S2R1, S1R2, S2R2.

7. Click **OK**.

The measurements are added to the standard table.

NOTE: A maximum of 200 measurements can be added.

If I have replicates, how do I calculate the concentration per replicate and ordinate mean per replicate?

For example, within Quantitative methods you have replicate samples. UV WinLab calculates the individual analyte concentration in the Results Table. However, you may wish to calculate the analyte concentration per replicate and the ordinate mean per replicate.

To calculate the analyte concentration per replicate, set up the following equation in Post-processing:

`mean[Quant.concentration.Sampleid.Replicates]`

To calculate the ordinate mean per replicate, set up the following equation in Post-processing:

`mean[Quant.Ordinate.Sampleid.Replicates]`

Create columns (with appropriate names) and add these to the Results Table.

When would I use Replicates and when would I use Measurements?

Replicates should be used when you have identical samples. For example if you have 3 solutions of a sample and they are all the same concentration, you would enter 1 sample in the number of samples and number of replicates = 3. You cannot use varying concentrations with replicates of a sample. If concentration is specified in the sample table, all replicates of a sample must have the same concentration.

Measurements should be used when you have similar samples. For example you have three solutions that are of varying concentrations having been made up by diluting one original solution. In this case you would enter 1 in the number of samples, and the number of measurements = 3. If concentration is specified in the sample table you can define the concentration of each measurement.

What else can I view on the Calibration page?

Spectral graph – shows the data for each standard for Scanning Quant.

Data – shows the data for each standard for Quant (Wavelength Program).

Calibration graph – can be used to display the Absorbance vs concentration graph, a graph of Calculated vs specified values, or the Residuals.

Calibration details – displays a report listing correlation coefficient and coefficients of the calibration equation, and the calculated concentrations and residuals for all the standards.

Calibration event – displays an event log for the calibration. This is only available in the Enhanced Security version of UV WinLab.

Can I save a Calibration?

As well as choosing at what stage of a task to save the calibration, you can choose to save the method and on the dialog displayed, choose to **Save Calibration**.

A Quant method may be developed with or without a calibration being performed. In the case of the latter, the analyst would have to run the Method and perform the calibration first, before analyzing the standards. In the case of the former and as referenced in step 16, the Method is saved with the calibration having been performed. The analyst would then only run the Method to analyze the standards.

How do I modify a Calibration?

When the calibration graph has been displayed, you can review and if necessary, modify the calibration curve by excluding one or more standards.

NOTE: Within the Enhanced Security version of UV WinLab, you can only modify a calibration if you have the correct permissions. Permissions are defined by the Administrator. Please contact your UV WinLab Administrator if you do not have the permission you need.

To exclude a standard, right-click on the standard point on the graph, or right-click on the relevant row of the standard table.

In the ES software, the point excluded is grayed out on the graph and in the standard table.

In the Standard version of UV WinLab, the standard is deleted from the table.

NOTE: IT IS NOT POSSIBLE TO INCLUDE A STANDARD AGAIN ONCE IT HAS BEEN EXCLUDED.

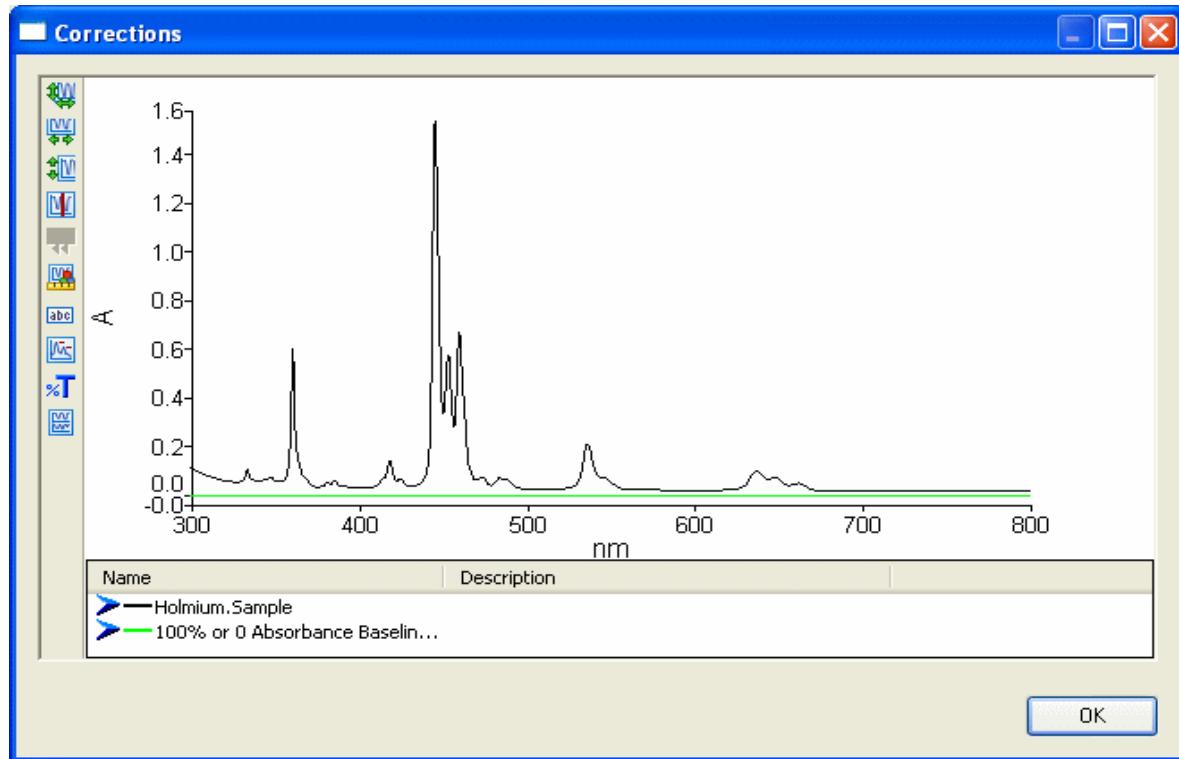
NOTE: When you modify the calibration you are warned that this will invalidate calibration. Once you elect to continue, the calibration will then be re-calculated.

How do I view the correction spectrum/spectra for a standard?

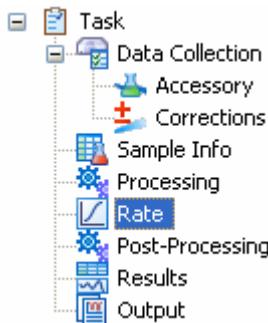
The correction spectra can be viewed within the Standards table.

- Right-click on the standard whose correction spectrum you wish to view, and select **View Corrections**.

A second window is displayed showing the standard and correction spectra:



Rate



Rate allows you to extract basic kinetic information from timedrive data (that is, measurements of absorbance as a function of time). A typical measurement involves measuring absorbance over a period of time during which a reagent, such as an enzyme, is added to the sample. Rate can be used simply to measure the absorbance or to perform additional calculations. These calculations may be based on the rate of change of absorbance or on the total change in absorbance. The rate of change is used to determine enzyme activity (Slope analysis) while the total change in absorbance provides a measure of substrate concentration (End point analysis).

What information can be determined from the rate data?

The rate input data is a plot of Absorbance versus Time. The following data can be determined from the plot:

- The total change in absorbance due to the reaction. Additionally, the change in absorbance is used to determine the substrate concentration.
- The slope of the Timedrive. This is determined by calculating the slope between two specified time points. This slope is used to calculate the enzyme activity.
- Creeping baselines. This is used to evaluate the total change in absorbance and slope of the curve. If the user decides the curve has a sloping baseline, they calculate the slope based on the different slope areas of the curves and the difference between the slopes in the resultant slope of the curve. The resultant slope is used to determine the enzyme activity.

How do I define the Time units?

The time units are defined on the Data Collection page.

What types of calculation are available?

- Select the type of calculation to be performed – **None**, **End point analysis** or **Slope analysis**.

If **None** is selected, the task is a simple Timedrive with no rate calculations.

End point analysis – use to determine the substrate concentration in the sample.

Slope analysis – use to determine enzyme activity.

What default Timedrive methods are available?

The following default Timedrive methods are provided. Further information about a method, see

- Simple Timedrive
- Timedrive with slope
- Enzyme Activity
- Substrate concentration.

Can I extend the time of a Timedrive when data collection is in progress?

Yes, simply alter the time on the Data Collection page.

NOTE: It is not possible to make the time less than was originally specified.

Can I import previously saved Timedrive data?

Yes. Data imported into the Sample Table are processed as samples – See Sample info for more information on Importing. Data imported into the graph display are for reference purposes only.

End Point Analysis

What is an end point analysis?

The change in absorbance is determined between the start of the measurement period and an end point that is either determined automatically or taken at a specified time.

What can I define on the End Point Analysis page?

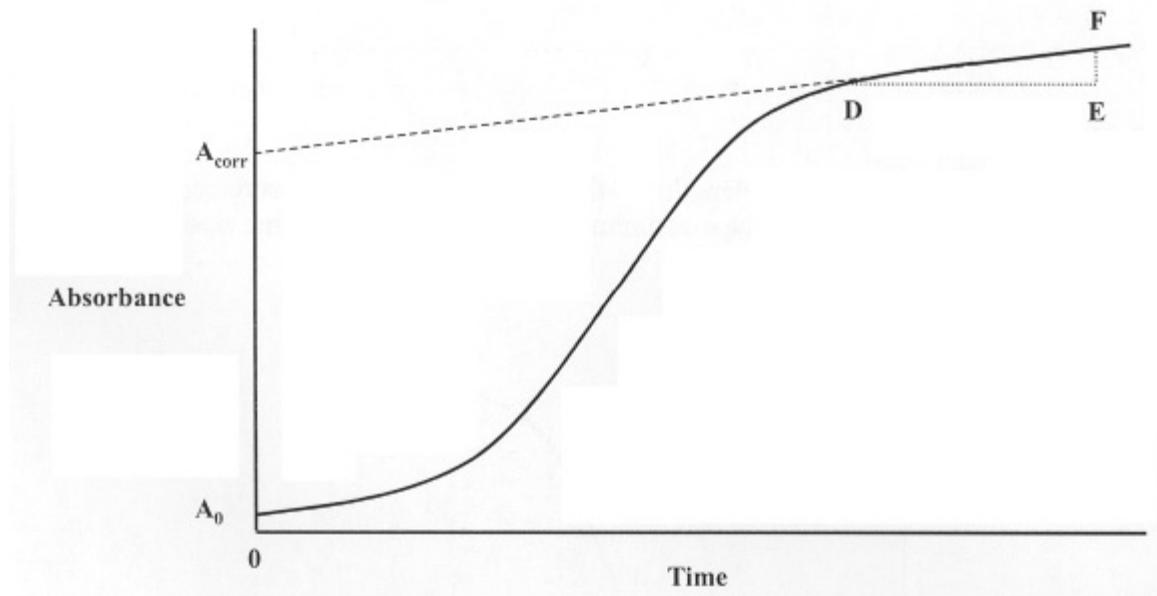
Automatic end point	The end point is automatically calculated at the end of the time set for the Timedrive.
Manual end point	Enables you to enter a value for the end point.
Baseline corrected end point	Enables you to calculate the true end point on the basis of the slope of the baseline over a section defined by two points.
Calculate substrate concentration	Enables you to calculate the concentration of the substrate.
Component	Enter the name for the column that will be added to the Results Table to display the substrate concentration results.
Units	Enter the units for the column that will be added to the Results Table to display the substrate concentration results. These units are displayed as Name (units) .

NOTE: The time units are set from the time units control on the Timedrive Data Collection page.

How do I determine the end point?

- You can select Automatic end point, Manual end point or Baseline corrected end point.

An **Automatic end point** is taken at the point where there is the maximum change in absorbance from the initial value. If the absorbance change is positive, the end point corresponds to the maximum absorbance. For a negative absorbance change, the end point corresponds to the minimum absorbance.



1. Select Automatic end point, Manual end point or Baseline corrected end point.
If you select Automatic, the end point is calculated automatically at the end of the time set for the Timedrive.
2. If you select **Manual end point**, enter the end point.
If you have imported a reference spectrum to locate the end point you can use the marker on the graph to locate the desired end point. If you use the mouse to move the marker on the graph the Manual end point field updates automatically.

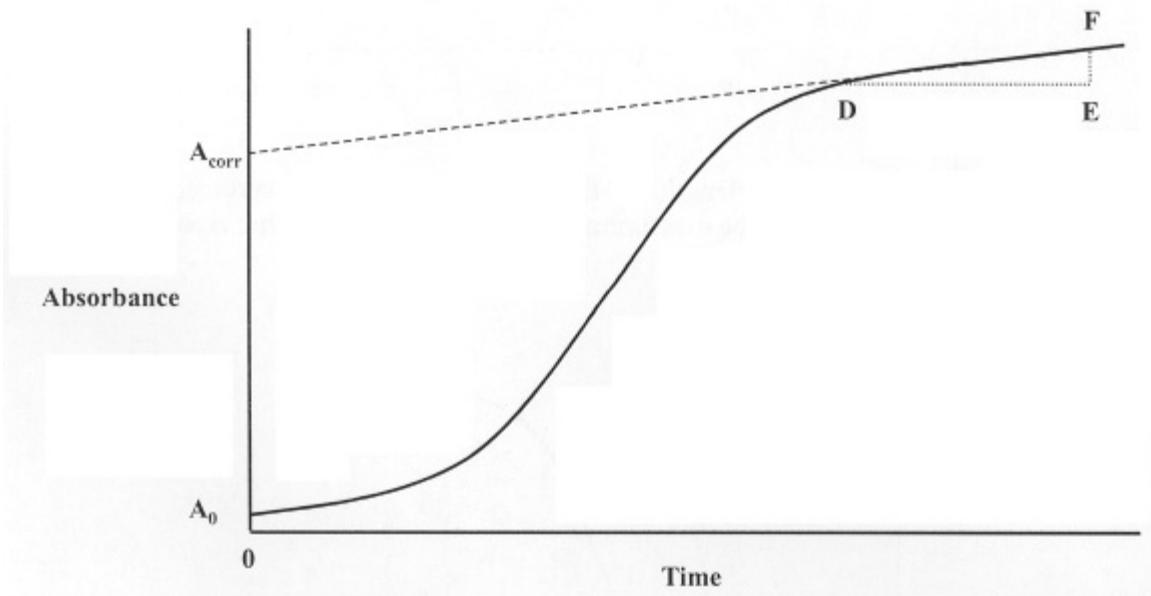
OR

If you select Baseline corrected end point, enter the Base 1 and Base 2 values.
If you have imported a spectrum to locate the base points you can use the base point markers on the graph to locate the desired points. If you use the mouse to move the base point markers on the graph the Base 1 and Base 2 fields update appropriately.

NOTE: A Manual end point cannot be calculated if the run was stopped.

What is Baseline corrected end point QA?

Baseline correction involves measuring the Slope over a period after the reaction is deemed to be complete. This is extrapolated back to the start time to give a corrected absorbance value A_{corr} .



$$\text{Baseline Slope} = EF / DE$$

$$A_{corr} = A(\text{end}) - \text{Baseline Slope} \times \text{Time (end)}$$

$$\text{Absorbance change} = A_{corr} - A_0$$

- Enter the **Base 1** and **Base 2** points to be used when calculating the slope.

OR

If you have imported a spectrum to locate the base points you can use the base point markers on the graph to locate the desired points. If you use the mouse to move the base point markers on the graph the **Base 1** and **Base 2** fields update appropriately.

How do I calculate substrate concentration?

The absorbance change can also be reported as a substrate concentration.

$$\text{Concentration} = \text{Absorbance change} \times \text{Molecular weight} / \text{Pathlength} \times \text{Extinction coefficient}$$

Selecting **Calculate substrate concentration** allows you to calculate the concentration of the substrate and include the results in the Results Table.

If you specify a dilution for a sample, the result is multiplied by the dilution factor to give the concentration of the material before dilution.

1. Select Calculate substrate concentration.
2. Click Settings.
The Substrate concentration settings dialog is displayed.
3. Enter values for the Molecular weight (g/mol), Pathlength of cuvette(cm), and the Molar extinction coefficient (mmol⁻¹ cm⁻¹).
4. Click **OK**.

5. Enter the **Component** name and **Units**.

This is the name of the column and the units that will appear as the heading for the substrate concentration column in the Results Table.

6. Select **Sample Info** from the Folder List.

The Sample Info page is displayed.

7. Click Format Sample Table.

The Table Builder dialog is displayed.

8. On the Preparation tab select **Substrate Concentration** from the drop-down list.

9. Select **Starting Volume** and/or **Added Volume** if you wish to enter the values at this stage. Otherwise, the columns are added to the table and the analyst can enter the values at run-time.

All the settings for calculating substrate concentration are now defined. When the samples are run, the Substrate concentration is reported on the Results page.

NOTE: A pre-defined substrate concentration method is available.

NOTE: A reaction may be accompanied by either an increase or a decrease in absorbance at the wavelength being monitored. In calculations of activity or concentration the absolute value of the slope or the absorbance change is used.

NOTE: If you re-process a Rate task which did not previously have substrate concentration selected, it is not possible to select it as by definition the sample table would need to be edited and this is not permitted when re-processing a task.

How do I display a previously collected time graph?

You may wish to display a previously collected graph if you select Baseline corrected end point, to help you determine the Start and End points.

1. Click Add Reference.

The Add Reference Spectrum dialog is displayed.

2. Select the graph to import.

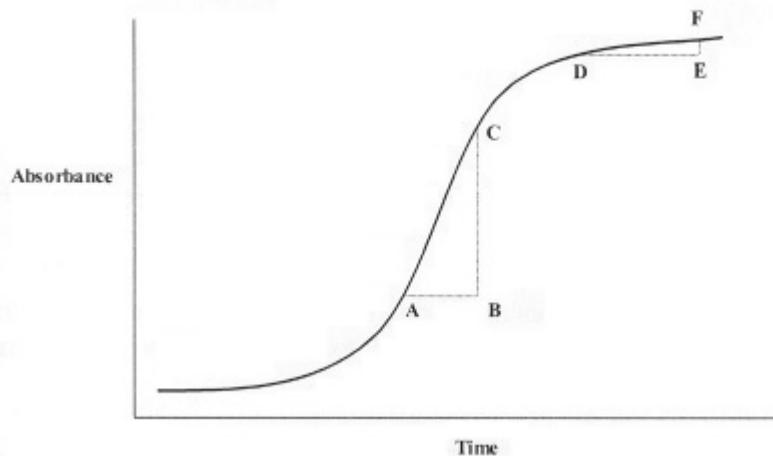
3. Click **Open**.

The graph is displayed in the graph pane.

Slope Analysis

What is slope analysis?

The slope is calculated from the change in absorbance (A) between two times – **Slope Start** and **Slope End**.



$$\text{Slope} = [\text{A(end)} - \text{A(start)}] / [\text{Time(end)} - \text{Time(start)}] = BC / AB$$

The result can be displayed in units of Enzyme Activity by specifying an **Enzyme activity factor**. The measured slope is multiplied by this factor to give the Enzyme Activity. You may specify a dilution for the sample, in which case the result is multiplied by the dilution factor to give the enzyme activity of the material before dilution.

NOTE: The slope is calculated between two data points (start and end). Fitting is not applied over the points in the selected range.

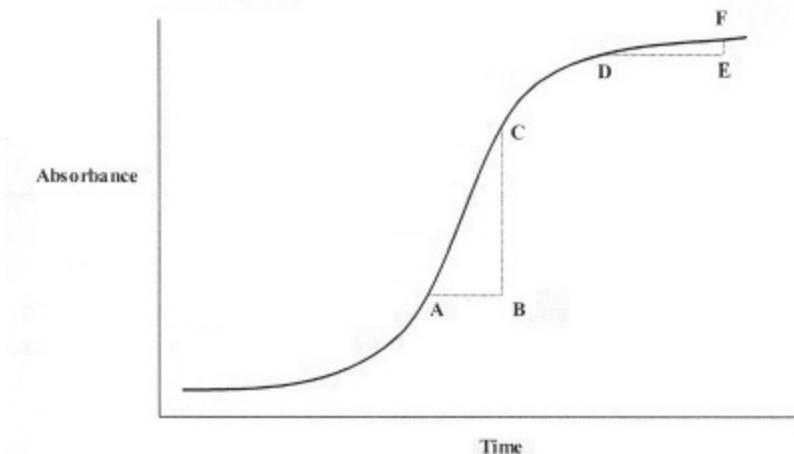
What can I define on this page?

Slope Start	Enter the start point of the slope.
Slope End	Enter the end point of the slope.
Baseline Correction	Select Baseline correction to automatically correct for a creeping baseline.
Calculate enzyme activity	Select Calculate enzyme activity to calculate the enzyme activity. An Enzyme activity factor must be specified. NOTE: If this is not selected, the calculated result is the slope.
Component	This is the name of the column added to the Results Table to display the enzyme activity.
Units	Enter the units that will also appear in the column header.

NOTE: The time units are defined on the Data Collection page .

How do I correct for a creeping baseline?

Optional baseline correction involves subtraction of a baseline slope calculated over a period for which separate Start and End times are specified.



$$\text{Baseline Slope} = EF / DE$$

1. Select Baseline correction.
2. Enter the **Base 1** value and **Base 2** value.
The baseline is checked for creep between these two points.

How do I calculate the enzyme activity?

1. Select Calculate enzyme activity.
2. Enter the **Enzyme activity factor** for the calculation.
3. In the **Component** field, enter the name of the component that will be displayed in the Results Table.
This column displays the enzyme activity results.
4. In the **Activity units** field, enter the units that will also be displayed in the enzyme activity results column header.

How do I use a previously collected time graph to define the baseline?

You may wish to display a previously collected graph if you select Baseline corrected end point, to help you determine Base 1 and Base 2.

1. Click Add Reference.
The Add Reference Spectrum dialog is displayed.
2. Select the graph to display.
3. Click **Open**.
The graph is displayed in the graph pane.

4. Move the markers displayed on the graph.
Base 1 and Base 2 are updated as appropriate.

OR

Enter values in the Base 1 and Base 2 fields.

How do I use a previously collected time graph to define the slope?

You may wish to display a previously collected graph to help you determine the Slope Start and End points.

1. Click **Browse**.
The Browse dialog is displayed.
2. Select the graph to display.
3. Click **Open**.
The graph is displayed in the graph pane.
4. Move the markers displayed on the graph.
Start Slope and End Slope are updated as appropriate.

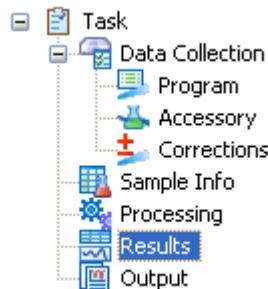
OR

Enter values in the **Start Slope** and **End Slope** fields.

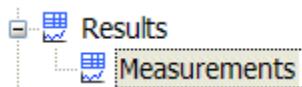
See also

Rate

Results



The Results page allows you to define the columns of results that are displayed when the samples have been run. It also shows spectra (where applicable) and peak tables (if previously defined). If measurements have been defined, a measurements sub-branch is added underneath Results:



How do I select which columns will appear in the Results Table?

1. Click **Organize Columns**.
The Columns tab of the Table Builder dialog is displayed.
2. Click in the box next to the name of a column that you want to appear in the table.
A tick indicates that the column will appear in the Results Table.

How do I reorder the columns in the Results Table?

The order of the columns listed on the Columns tab is the order in which the columns will appear in the Results Table.

1. Click on a column name to select it.
A dotted line appears around the name of the column to show that it is selected.
2. Click **Move Up** or **Move Down** to move the selected column up or down the list.
The columns in the Results Table are updated accordingly.

What menu items are available when I right-click on a row in the Results Table?

The following menu items are available:

View Sample Event Log	Displays the Sample Event Log.
Add Comment	Allows you to add a comment to a sample.
Show Corrections	Shows the correction spectra collected for the sample.

How do I export the information displayed in the table when I am working with a Wavelength Task?

The data in the table can be copied to the clipboard.

- Right-click anywhere in the table and select **Copy To Clipboard**.

All the data in the table is copied to the clipboard, It can then be directly pasted into another application such as Excel.

NOTE: If you select certain cells in the table before selecting **Copy To Clipboard**, only the selected cells are copied to the clipboard.

What is the Sample Event Log?

In the Enhanced Security version of UV WinLab:

The Sample Event Log records all events connected with the sample. For example, if a description is added after the sample has been run, this event is recorded in the Event Log. The original value and new value are recorded.

The Sample Event Log records the event, time, user, and reason/comment.

NOTE: If a field has been changed from being empty to containing a value, empty quotes "" are used to show that the field was initially empty. For example, if the description field was empty and then changed to read Batch 1, the Event Log would record: Description changed from "" to "Batch 1".

In the Standard version of UV WinLab:

The Sample Event Log only records any comments associated with the sample.

How do I add a comment to a sample?

1. Select the sample to which you wish to add a comment.
2. Right-click and from the menu select **Add Comment**.
The Add Comment dialog is displayed.
3. Enter the text and then click **OK**.

NOTE: A comment can be added to a sample via the **Add Comment** menu item that is displayed when you right-click on a sample.

The comment can be viewed in the Sample Event Log – right-click on the sample and select **View Sample Event Log**. The Sample Event Log records the Event, Date and Time, User, and Reason/Comment.

A comment can also be added and viewed via the results of a Query.

How do I view the spectrum of a sample?

- Select the sample whose spectrum you wish to view by clicking in the field in the Results Table.
The spectrum on the Graph tab below updates for the selected sample.

How do I view a peak table?

If a peak table has previously been defined as part of the processing, a peaks tab is added on the results page.

- Select the Peaks tab to display the peak table information:

The screenshot shows the 'Results' window with the 'Peaks' tab selected. The left sidebar lists 'Task', 'Data Collector', 'Instrument', 'Accessory', 'Corrections', 'Sample Info', 'Processing', 'Results' (which is selected and highlighted in blue), and 'Reporting'. The main area displays a peak table with the following data:

Sample ID	Sample3.Sample
Description	
Threshold	0.1
Range start (nm)	800
Range end (nm)	400
Find	Peaks
Display	List by position

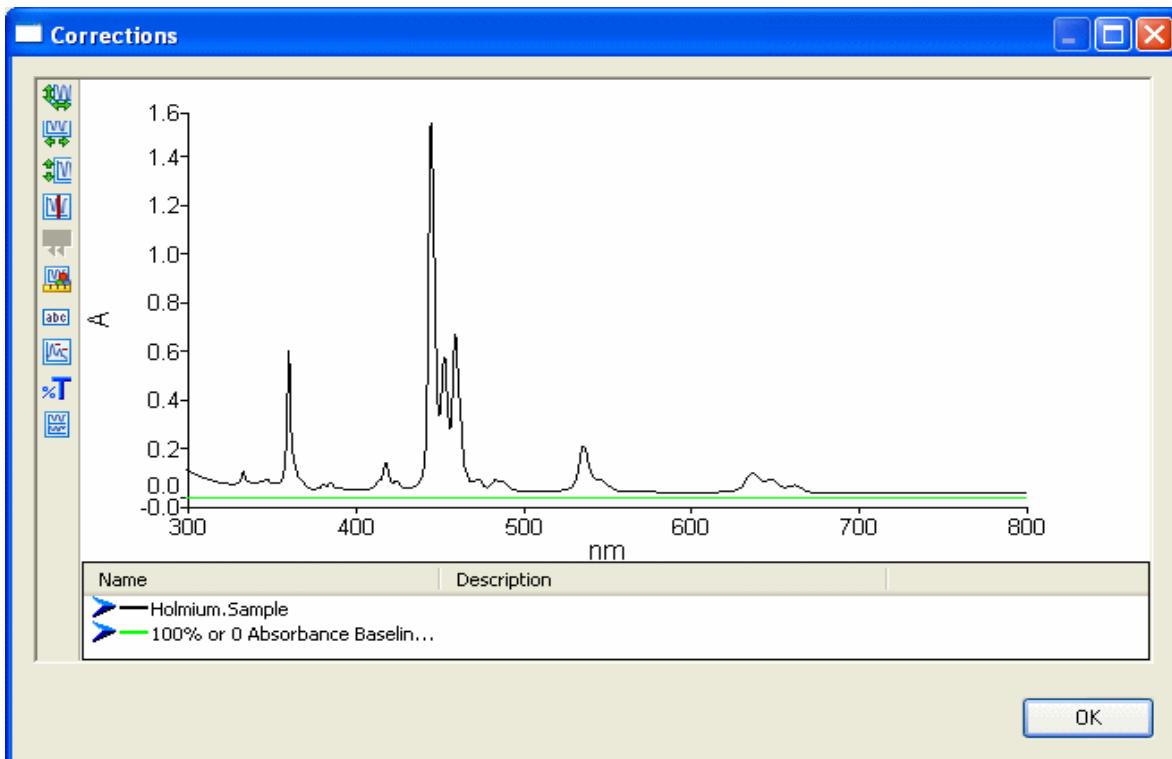
	Position (nm)	Intensity (A)	Type
1	536.49	0.2452	Peak
2	459.98	0.638	Peak
3	453.2	0.5658	Peak
4	445.87	1.344	Peak

How do I view the correction spectrum / spectra for a sample / standard?

The correction spectra can be viewed within the Sample Table, Standards Table or Results Table.

- Right-click on the sample / standard whose correction spectrum you wish to view, and select **View Corrections**.

A second window is displayed showing the sample and correction spectra:



What is the Replicates tab?

Replicates only appears if you have replicate samples and gives the average result for the set of replicates.

For example, if you have Sample10.Replicate1 and Sample10.Replicate2, on the Replicates page there is Sample10.Mean

What is the Custom tab?

Custom appears if you have set up an equation that needs to put the results in a new table because they don't apply to each sample. For example the equation discussed in Sample Tag creates a result from two samples so it used a new Custom table to display those results.

What is displayed on the Measurements page?

A table listing the measurement samples that have been run and a graph tab (if applicable) shows the spectra. The table can be formatted in the same way as the Results Table. See 'How do I select which columns will appear in the Results Table ?' above. All the information above on this page of the Help such as adding comments also applies to the measurements page.

What is a custom table?

A custom table is generated when equations are used in processing and when the result of an equation cannot be associated with a particular sample.

For example if you have 2 samples and you define an equation as **Sample 1 / Sample 2**, the result is not associated with either of these samples and so a custom table is created to report the result.

What is the Mass fraction in solid column?

The Mass fraction in solid column is added to the Results Table (by default) when a Quant method is run and the preparation type (specified on the Preparation tab of the Table Builder dialog) is set to **Dissolved solid** or **Dissolved Solid and dilution**. The column is also available on the Sample Info page but it is not enabled by default.

NOTE: There are no units associated with this column as the quantity is dimensionless.

For Dissolved solid, the values in this column are calculated using the equation

Mass Fraction = Concentration in solution (mg/ml) * solution volume (ml) / Mass of substrate (mg)

that is, Quant Predicted concentration (Analyte) column * Volume column / Weight column

For Dissolved solid and dilution, the values in the Mass fraction in solid column are calculated using the equation

Mass Fraction = [Concentration in solution (mg/ml) * solution volume (ml) / Mass of substrate (mg)] * final volume / initial volume

that is, [Quant Predicted concentration (Analyte) column * Volume column / Weight column] * final volume column / initial volume column

NOTE: If Weight = 0 or initial volume = 0, the result will be empty.

The values from this column will be available in Post-processing Equations.

Custom Columns

What is a custom column?

A custom column is a user defined column that can be added to the Results Table.

How do I add a custom column?

1. Click Format Sample Table.
The Table Builder dialog is displayed.
2. Select the Columns tab.
3. Click **Add**.
The Column Type dialog is displayed.
4. Select Data entry, Text selection or sample tag and click OK.
The appropriate Format dialog is displayed.
5. Edit the details as required.

How do I delete a custom column?

1. On the columns tab of the Table Builder dialog, select the column you wish to delete by clicking on it.
A dotted line appears around the name to show that it is selected.
2. Press the **Delete** key on your keyboard.
The column is removed.

NOTE: Sample ID is a mandatory column and cannot be deleted.

Formatting Columns

What types of column can be present in the Results Table?

There are 4 types of column:

System Column	Sample ID is the only column to appear by default in a table. Sample ID, Description, and Type are System Columns. It is not possible to define any other System Columns.
	NOTE: Sample ID cannot be deleted.
Data Entry Column	A Data Entry Column is a custom column added by the user. Numeric values that can then be used as variables in equations are expected in a Data Entry Column.
Text Selection Column	A Text Selection Column is a custom column added by the user. Each cell in a Text Selection column contains a drop-down list of options. These options are defined within the Format Text Selection dialog.
Sample Tag	A special text selection column that is used as an identifier in special equations that use variations on a sample. The sample tag becomes part of the spectrum name.
	NOTE: A sample tag column cannot be deleted once it has been created.

The type of formatting that can be performed on a column depends on the type of column.

What is a Sample Tag?

A sample tag is a special custom column that can be added to assign an 'alias' to a sample. For instance if you have a group of samples that each have their own particular identifier that you need to use as the sample name, but you also want to pick out samples as belonging to special groups, you can use a sample tag.

Sample tags are like other text selection columns in that the method developer assigns the entries available and then when samples are entered in the table there is a drop down list of options for the user to select from. Where this differs from a standard text table is that these options then become available in the Variables list in the Equation Builder so you can specifically select an equation to operate only on samples with that tag, or perform an operation that differentiates between samples with different tags.

NOTE: You should not use the words **Sample**, **Blank** or **Control** as sample tags as this will cause confusion with the sample types.

An Example use of sample tags is where you have a matched pairs of samples where you want to subtract one from the other:

1. Add the **Sample Tag** column and format it with the list reading **A** and **B**.
2. In your Sample Info table use the drop-down list in the new Sample Tag column to select which samples are A and which are B (in this example they must run in sequence one after the other and there must be an even number of each).
3. In Processing, set up an equation using these new variables, for example **Height[A, 265]–Height[B, 265]**.

When samples are run a Custom table will appear in the Results that gives the result of this equation for each pair of samples.

How do I format a Column?

1. Click Organize Columns.
The Columns tab of the Table Builder dialog is displayed.
2. From the list of columns highlight the column you want to format and click **Format**.
The appropriate formatting dialog is displayed:

System Column

NOTE: Only the font can be changed in a System Column.

The Format System Column dialog is displayed.

3. Click Change Font.
The Font dialog is displayed.
4. Select the **Font**, **Font Style**, **Size** and **Color**, and then click **OK**.
An example of the selected font is shown on the Format System Column dialog.

Data Entry Column

The Format Data Column dialog is displayed.

5. If you wish to format a Data Entry Column that already exists, select the **Name** of the Data Entry Column that you wish to format from the drop-down list of all available Data Entry Columns.

OR

To create a new Data Entry Column, enter a new **Name**.

NOTE: If you edit the **Name** of a previously saved column, a new column is created.

NOTE: It is not possible to create two columns with the same name.

- If you wish to add units to a column header, enter the Units in the field:

The screenshot shows a 'Column Details' dialog box. It has three sections: 'Name' (containing 'Initial Volume'), 'Units' (containing 'ml'), and a bottom section. The 'Initial Volume' field is highlighted with a blue border.

The units are automatically placed in brackets after the column name.

NOTE: If you edit the details of a previously saved column, a message will be displayed informing you that if you save the changed details, all methods containing the column will also be updated. You can choose to **Save** or **Rename**. Click **Save** to save the column details, overwriting the previous settings for that name. Click **Rename** to enter a new name for the column settings.

- If you want the analyst to enter information in the selected column before they can run a sample, select **Mandatory**.
- If you want the analyst to only be allowed to enter numbers in the selected column, select **Numbers only**.

NOTE: Significant figures or Decimal places are only available when Numbers only is selected.

- Click **Change Font** to change the font type, style, size and color.
- If **Numbers only** has been selected, select the number of **Significant figures** (1 to 9) or the number of **Decimal places** (0 to 9) from the drop-down list.

Text Selection Column

The Format Text Selection Column dialog is displayed.

- If you wish to format a Data Entry Column that already exists, select the **Name** of the Data Entry Column that you wish to format from the drop-down list of all available Data Entry Columns.

OR

To create a new Data Entry Column, enter a new **Name**.

NOTE: If you edit the **Name** of a previously saved column, a new column is created.

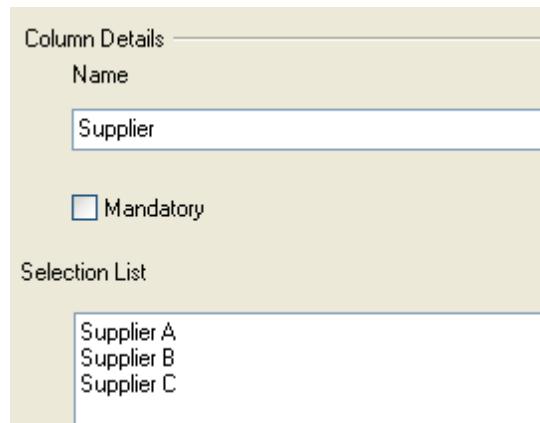
NOTE: It is not possible to create two columns with the same name.

- If you want the analyst to enter information in the selected column before they can run a sample, select **Mandatory**.

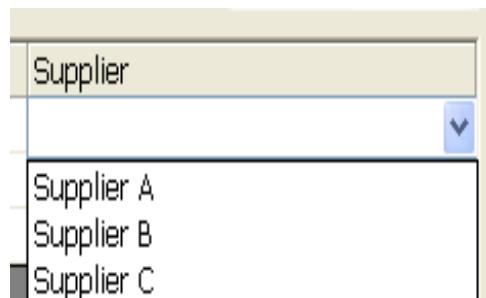
13. Enter the words to be selectable (from a drop-down list) for each cell in the column in the **Selection listfield**:

In the Supplier column you want the analyst to be able to choose from Supplier A, Supplier B and Supplier C.

Enter Supplier A, Supplier B and Supplier C in the Selection list



When the user clicks on a cell in the **Supplier** column of the table a drop-down list is displayed containing Supplier A, Supplier B and Supplier C.

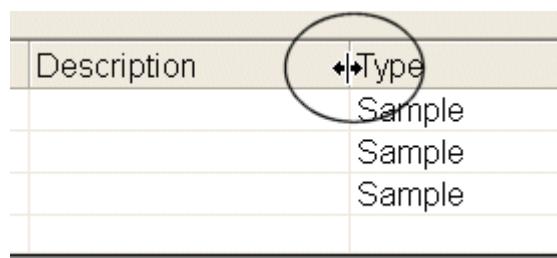


Click **Change Font** to change the font type, style, size and color.

How do I alter the width of the columns in the Results Table?

1. Position the mouse pointer between two columns.

The mouse pointer changes:



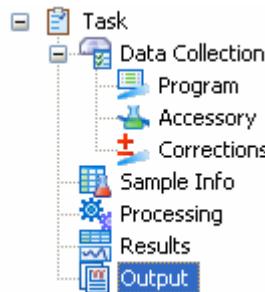
2. Drag the edge of the column in the required direction.

Can I cut or copy and then paste between cells?

1. Select the cell you wish to cut or copy.
A dotted line appears around the cell to show that it is selected.
2. From the Edit menu select **Cut** or **Copy** as appropriate.
3. Position the cursor in the cell into which the information is to be pasted and then select **Paste** from the Edit menu.
The information is pasted into the selected cell.

NOTE: It is also possible to paste information from the Windows clipboard into the Sample Table.

Output



The Output page allows you to select the template to use when creating reports, how frequently the report should be created, and how the report should be output. A template is available with each of the methods supplied as part of UV WinLab. A preview of the selected report template is shown on the Output page. A new report template can be created via the Output page. It is also possible to export data to a specified directory at the same time as the reports are generated.

NOTE: If you are using the Enhanced Security version of UV WinLab, the template assigned to a locked Method cannot be altered by the end user. If the Method is not locked, you must have permission to create and edit report templates in order to alter the assigned template. Please see your UV WinLab Administrator for further information about your permissions.

Templates

How do I select the report template to use?

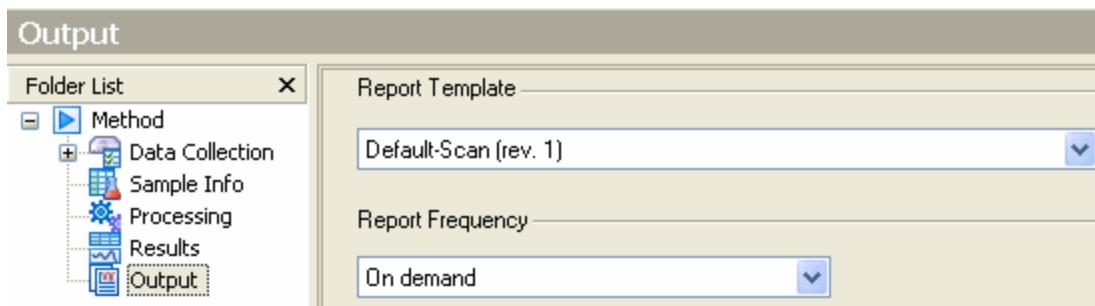
- Select the required template from the drop-down list of available templates.

How do I know which template to use?

A report template is provided with each supplied example method.

When you highlight a Report Template, the Display Pane lists Name, Description, Type, Created by, Created on, Revision, and Status.

A list of all Report Templates is also available from the Output page within the Workspace, by selecting the **Template** drop-down list.



NOTE: A report template must be approved before a report can be saved to the database using the template. See the Communiqué Report Creator section of the Help for further information on approving report templates.

Can I edit the selected template?

- Yes, select the template from the drop-down list and then click **Edit**.
Communiqué Report Creator opens enabling you to edit the selected template.

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have permission to create and edit report templates. For further information about your permissions, please contact your UV WinLab Administrator.

How do I preview the report?

- To see a preview the report, select the template from the drop-down list and click **Preview**.
The report is shown in the Communiqué Print Preview window.

How do I create a new template?

- Click **New**.
Communiqué Report Creator opens enabling you to create and save a new template.
When the template has been saved, the name appears in the Template drop-down list on the Output page.

Report frequency

How often can I obtain a report?

It is possible to obtain a report **On Demand**, **After each sample**, or **When task is completed**.

NOTE: A task is only completed once it has been saved. **On Demand** is the default.

- Select the required frequency from the **Report frequency** drop-down list.

Reports are obtained **On Demand** via the Explorer, which lists all reports saved to the database.

NOTE: Reports can only be saved to the database if an approved report template is used.

See the Communiqué Report Creator book  [Welcome to Communiqué](#) in this Help file for further information on approving report templates.

It is also possible to re-open a task and select to print the report.

Report output

What type of outputs are available?

The report can be output in the following ways: **Print hardcopy**, **Print to file** or **Print to database**. It is possible to select more than one type of output.

NOTE: If a Method is locked but the report template associated with the Method has not been approved, **Print to database** will not be available.

How do I define the settings for the selected output(s)?

1. Select the required output(s) and then click **Setup**.

The Output Setup dialog is displayed.

2. Set the options as required.

Each of the tabs is described below.

Hardcopy tab

The Hardcopy tab enables you to select the printer and the number of copies to be printed.

3. Select the **Printer** from the drop-down list of available printers.

4. Enter the **Number of copies** to be printed.

File tab

The File tab enables you to specify where the file should be saved to and the type of file to be created.

5. Select where you want the path to go; select the path from the drop-down list of previous destinations, type in a new path, or click **Browse** to select a directory for the file path.

6. Select the **File type** you want to create from the drop-down list of available file types.

Select ASCII (.txt), HTML (.htm), Microsoft Excel (.csv) or Microsoft Word (.rtf).

Sections tab

This tab allows you to switch sections created in the report template, on and off. All the sections created in the report are listed.

To turn a section on (that is, include it in the report):

- Highlight the required section in the Sections Off list and click **On**.
The section moves from the Sections Off to the Sections On list.

To turn a section off (that is, do not include it in the report):

- Highlight the required section on the Sections On list and click **Off**.
The section moves from the Sections On to the Sections Off list.

See Communiqué for more information on sections.

Data Export

Why would I use Data Export?

Data Export exports data generated when the task is created. The data can then be used by software such as Color Application Software, Protection Glass Application Software, Architectural Glass Application Software and Filter Application Software.

Tables, including the Sample Table and Replicates Table, Spectra (raw or processed), and Wavelength Program data can be exported. You can choose to export an entire table, or just selected columns from the table.

NOTE: The data is exported when the Task is saved. To export data manually, at any time the Task is open, select **Export** on the File menu to access the Export Data dialog. The settings can be different to those set in the Export Data dialog accessed via the Output page.

How do I export data?

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the correct permission to be able to export data. If you cannot see this option, please consult your UV WinLab Administrator regarding your permissions within the software.

1. On the Output page, select **Output to file**.

- ## 2. Click **Setup**.

The Export Data dialog is displayed. The folder to which the data will be exported is shown at the top of the options.

NOTE: The Output page is where you define what data you would like to export when saving a Task. You can also access the Export Data dialog via **Export** on the File menu to export your data manually, at any time a Task is open. The settings can be different to those set in the Export Data dialog accessed via the Output page.

3. To change the default export folder, click on the underlined path.

The path becomes a field and a Browse button appears:

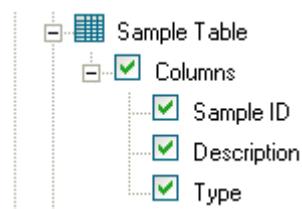


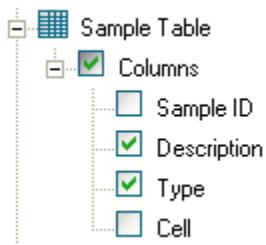
4. Click , and select the required folder or click **Make New Folder** to create a new folder.

The new path is displayed

- ## 5 Select the items to be exported

A check mark indicates that the item will be exported. When, for example, Columns (Sample Table) is automatically selected, all the items below are automatically selected: Each of these items can be selected or deselected as required. The columns check box remains selected but becomes grayed to show that only some of the options below are selected:



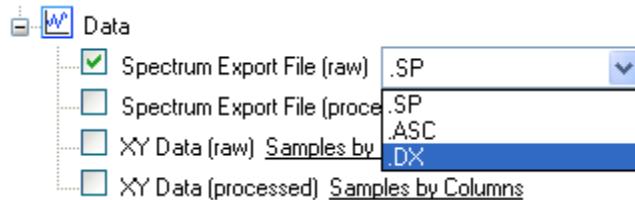


NOTE: If a branch (for example, Columns) is selected, all items underneath are automatically selected. However, if another item is later added below this branch (for example, adding another column to a table will add the column name to the list of columns) this is not automatically selected. You must deselect the branch and then reselect it so that all items are selected.

Where an item at the end of an option is underlined:, this means that a drop-down list is available and the type can be changed.



6. Click on the underlined text to display a drop-down list:, and select the required option.



The type is updated on the dialog:



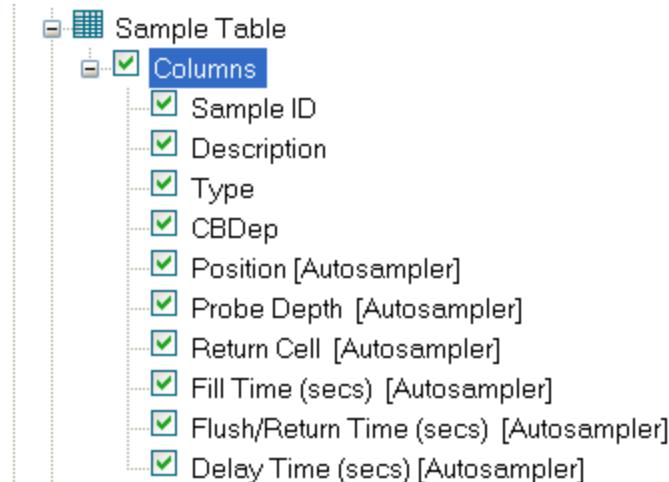
- To restore the default settings for data export, click **Restore Defaults**.
You will be asked to confirm that you wish to restore the defaults.

What data can be exported?

The Export Data dialog displays all the data that can be exported:

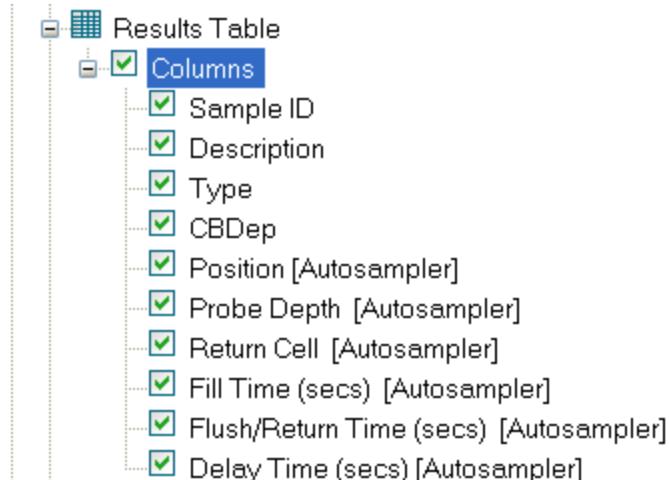
- Sample Table:

This is an EXAMPLE of the columns available within the Sample Table:



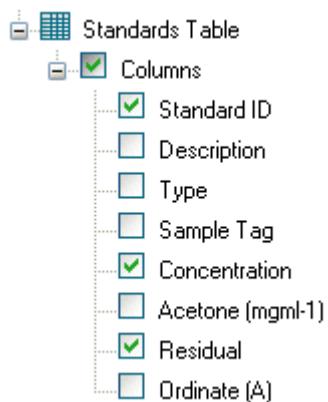
- Results Table:

This is an EXAMPLE of the columns available for export within the Results Table:



- Standards Table:

This is an example of the options available when exporting a standards table:



This is the standards table information from a scanning quant method:

	A	B	C
1	Standard ID	Concentration	Residual
2	s10std1	7.9	0.0131
3	s10std3	1.98	-0.0781
4	s10std4	0.79	0.065
5			

- Custom Table:

This is an example of the options for a custom table:



This is the page from the Workspace:

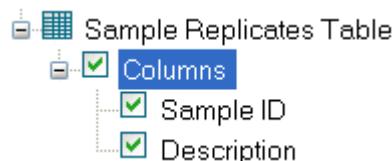
	A	B
1	Mean (nm)	656.090
2	Measured SD (nm)	0.000431
3	Repeatability Limit (nm)	0.02
4	Test Result	PASS

Below is the exported csv file:

	A	B
1	Mean (nm)	656.09
2	Measured SD (nm)	0.000431
3	Repeatability Limit (nm)	0.02
4	Test Result	PASS
5		

- Replicates Table:

This is an EXAMPLE of the columns that can be exported in the Replicates table:



- Peak Table:

This is an example of the items available when exporting a peak table:



This is the peak table from a Wavelength accuracy D2 spectrum:

	A	B	C	D
1		Position (nm)	Intensity (E2)	Type
2	WAD2-656-1	656.22	1.145	Peak
3				

- Spectral data (Spectrum export file and XY data):



The spectra and/or data can be exported.

Data: Samples by Columns:

	A	B
1	nm	A
2	365	0.14312
3	364.9	0.145429
4	364.8	0.147672
5	364.7	0.150374
6	364.6	0.153178
7	364.5	0.154722
8	364.4	0.1573
9	364.3	0.159255
10	364.2	0.162608
11	364.1	0.165575
12	364	0.169412
13	363.9	0.17167
14	363.8	0.176208
15	363.7	0.179481
16	363.6	0.184289
17	363.5	0.188599

Data: Columns by Samples:

	A	B	C	D	E	F	G	H	I	J
1	nm	365	364.9	364.8	364.7	364.6	364.5	364.4	364.3	364.2
2	A	0.14312	0.145429	0.147672	0.150374	0.153178	0.154722	0.1573	0.159255	0.162608
3										

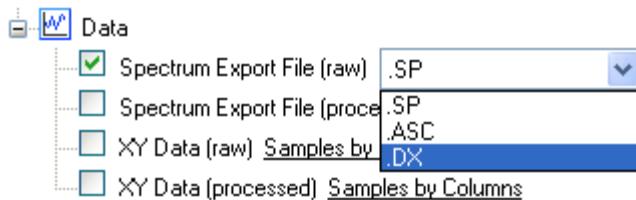
NOTE: The list of columns available for each table will be the columns selected to be displayed (see Sample Info). If a column is displayed, but does not contain any information, it is not selected for export by default.

What file formats are available for exporting spectra?

Raw and/or processed spectra can be exported as *.sp, *.asc or *.dx format.

The spectral data can also be exported as XY data (raw and/or processed data). This will create a CSV file when exported.

- Click on the underlined text to display a drop-down list:, and select the required option.



The type is updated on the dialog:



NOTE: You can also export your raw spectra as *.sp format using the **Save Spectra** option on the File menu. See Can I save my spectra as *.sp files?.

If I export XY data, how is the table created?

The table can be created as samples by columns or columns by samples.

- Click on the underlined text to display a drop-down list:, and select the required option.



The type is updated on the dialog:



Samples by columns

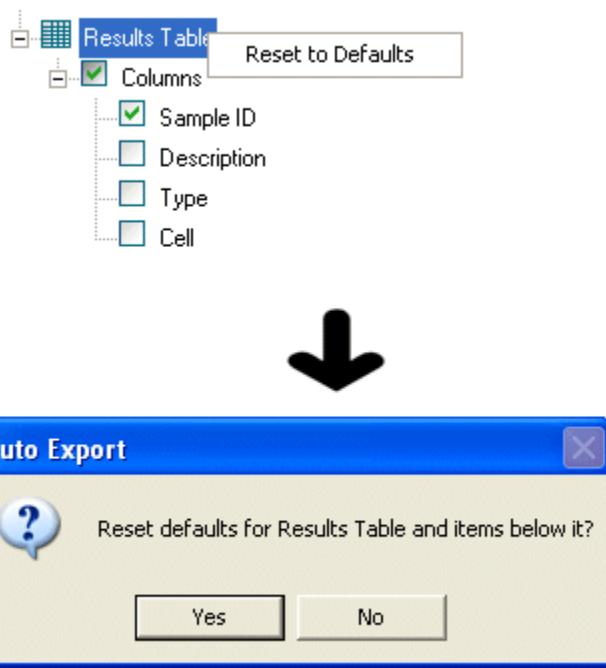
	A	B
1	nm	A
2	500	-0.00017
3	499	-0.00015
4	498	-0.00013
5	497	-9.2E-05
6	496	-5.7E-05
7	495	-5.4E-05
8	494	-6.9E-05
9	493	-7.1E-05
10	492	-8.9E-05
11	491	-8.2E-05
12	490	-0.00006
13	489	-8.8E-05
14	488	-8.3E-05
15	487	-3.3E-05
16	486	0.000003
17	485	-1.5E-05
18	484	0.000001
19	483	0.000021
20	482	0.000044
21	481	0.000078
22	480	0.000072
23	479	0.000064
24	478	0.000087
25	477	0.000115

Columns by samples

	A	B	C	D	E	F	G	H	I	J	K
1	nm	500	499	498	497	496	495	494	493	492	491
2	A	0.067507	0.06751	0.067591	0.067394	0.067397	0.067974	0.067677	0.067381	0.067821	0.068126
3											

How do I reset the defaults if I do not wish to keep the settings I have chosen?

- To restore all defaults, click **Restore Defaults**.
You will be asked to confirm that you wish to restore the Folder List to the default settings.
- To restore the defaults of just a particular node on the Folder List, right-click on the required node and select **Reset to Defaults**.
You will be asked to confirm that you wish to restore the default settings for the node and all settings below it:



Is it possible to export data to the same location each time?

Yes, If you are using the Standard version of UV WinLab, the previous data will be overwritten.

If you are using the Enhanced Security version of UV WinLab, it is not possible to save tasks with the same name. Therefore, the data cannot be overwritten.

Are hidden table columns available for export?

Yes, if a column is added to a table but is then hidden (see Sample info), it will be available for export as the software will populate all specified columns even if they are not displayed. The Export Data dialog will automatically list all table columns (displayed and hidden).

Call Application

Why would I use Call Application?

Call Application enables you to export your spectra or data directly to another software application after each sample is run or at the end of a task. For example, you can send the XY data of your processed sample to Microsoft Excel as a .csv file, after each sample is run.

How do I set up Call Application?

NOTE: If you are using the Enhanced Security version of UV WinLab, you must have the correct permission to be able to use Call Application. If you cannot see this option, please consult your UV WinLab Administrator regarding your permissions within the software.

Call Application exports the data in the formats defined in the Data Export dialog.

1. In the Data Export section, make sure **Output to file** is selected and click **Setup**.

2. Define the Data Export settings for the output as described in "How do I export data?" on this page.
3. In the Call Application sections, click **Add**.
4. Click the mouse button in the **Executable Path** field.
5. Enter the path of the application you wish to execute.
The new path is displayed in the field.

OR

click  to display the Open dialog and browse to the executable file.

The new path is displayed in the field. The path can be viewed in full in a tooltip by hovering the mouse over the field.

If the application you wish to call is one of the PerkinElmer Advanced Spectroscopy software packages, such as Color, Filter, Architectural Glass and Protection Glass, the syntax for the command line is C:\UVWINLAB\AppName where AppName is:

Application	AppName
Color	UVAPPLC.EXE
Filter	UVAPPLF.EXE
Architectural Glass	UVAPPLG.EXE
Protection Glass	UVAPPLP.EXE

6. In the **Commands** field, type the command line parameters for the external application.

Use <*> to designate the output files to be passed to the application. This is a placeholder that will be replaced with the filename of each sample.

If the application you wish to call is one of the PerkinElmer Advanced Spectroscopy software packages, such as Color, Filter, Architectural Glass and Protection Glass, the application parameter typically appears as "MethodName" <*>, where MethodName is the path to an application method file (generated using the application) and <*> is a placeholder for the output filenames.

NOTE: The path to the method file must be enclosed in quotes. For example, "C:\UVWINLAB\MyColorMethod.mcx".

NOTE: Different software applications may require different command lines. Please see the relevant software Help for details of the command line options.

7. Select the **Input** files.

The options are **None**, **Current Raw Spectra**, **Current Processed Spectra**, **Current Raw XY Data**, **Current Processed XY Data**, **All Raw Spectra**, **All Processed Spectra**, **All Raw XY Data** and **All Processed XY Data**.

If **At end of run** is selected, the current sample is the last sample run.

8. Select when the application will start from the drop-down list.
The options are **At end of run** and **After each sample**.
If **At end of run** is selected, the current sample is the last sample run.
9. Select the **Wait** option from the drop-down list.
If **Yes** is selected the system will wait until the software application has finished before continuing with the run.
If **No** is selected, the system will not wait.
For example, if you run three samples and choose to execute Excel **After each sample** and select **Yes** for the Wait option, you will need to close Excel before the second sample will be run.
10. Select **Ensure compatibility with previous versions of PerkinElmer UV applications software (ASSP)** if the application you wish to call is one of the PerkinElmer Advanced Spectroscopy Software Package applications used with UV WinLab versions 2.x/3.x. This will ensure the validity of the command line passed to these applications.
11. Select **Suppress sample prompts** if you selected After Each Sample in Step 8 and you do not want to display sample prompts. This is relevant for some custom accessories that use Call Application to move or change samples, and therefore do not require sample prompts.
The only prompts displayed if this option is selected are for Autozero and for the first sample.

How do I delete a row from the Call Application table?

- Click the mouse button in the row you wish to delete and click **Remove**.

How do I move a row up or down in the Call Application table?

- Select the row you wish to move and click **Move Up** or **Move Down** to move the row up or down respectively.

Graphs

Graphs are displayed on the Graphs tab in the Sample Info page and in the lower panes of the Processing and Results pages in the Workspace.

What is displayed on the graph?

If the Sample Table has been run and data collected, all spectra currently in memory are displayed either on one graph (Overlay Display mode) or on separate graphs (Split Display mode).

To switch between Split Display mode and Overlay Display mode and vice versa:

- Within the Workspace go to the View menu and select or deselect the **Overlay Samples** check box.

NOTE: Overlay Samples is not available for Wavelength Program or Wavelength Quant method types.

NOTE: The **Raw**, **Processed** and **Both** controls are only available when the graph is shown as part of the Processing level of the Folder List.

Select **Raw** to display the raw data as generated from the settings on the Data Collection page.

Select **Processed** to display the data after all the processes defined within the Processing page have been performed in order.

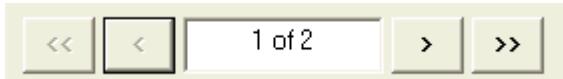
Select **Both** to display both the raw and processed data for comparison.

To switch between **Raw**, **Processed** and **Both** viewing options:

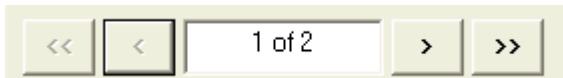
- Select the **Raw**, **Processed** or **Both** check box at the bottom of the viewing area.

What do the buttons on the top of the graph do?

Hover the mouse over the buttons to find out what they do.



Hover the mouse over the buttons to find out what they do.



What do the icons on the graph do?

	Full Range sets the graph to the maximum visible region for the graph.
	AutoRange X sets the graph X axis to the maximum for the viewable image.
	AutoRange Y sets the graph Y axis to the maximum for the viewable image.
	Vertical Cursor used to identify the X and Y co-ordinates of a point on the graph. The abscissa value is displayed in a box adjacent to the cursor and the ordinate value is displayed in the cursor column below the graph.
	Previous Range resets the graph to the previous scaling.
	Format Graph displays the Graph Properties dialog.
	Add Text Displays the Label Properties dialog which enables you to add text onto the graph.
	Peak Labels Displays the Label Properties dialog which enables you to label a peak.
	Convert to absorbance/Convert to percentage transmittance toggle button enables you to view the graph in absorbance or transmittance.
	Split Display/Overlay Display toggle button enables you to overlay multiple graphs or split them into single displays.

What menu items are available when I right-click on a spectrum name below the graph?

The following menu items are available:

Save as Binary	Displays a Save As dialog, allowing the spectrum to be saved as a .sp file to the selected directory.
Save as ASC	Displays a Save As dialog, allowing the spectrum to be saved as a .asc file to the selected directory.
Remove Curve	Clears the current curve from the view.
Status	Displays the Status dialog.
Appearance	Displays the Appearance tab of the Graph Properties dialog.

What menu items are available when I right-click on the background of the graph?

The following menu items are available:

Previous Range	Undoes a command that changes the X or Y range displayed by the graph. NOTE: To undo a Full Scale command, use the Previous Range command twice: once to undo the X component and once again to undo the Y component.
Full Scale	Sets the X range to the Start and End data points, and the Y range to the maximum and minimum, of the selected spectra.
Vertical Cursor	Places a vertical line on the graph that you can drag along the X axis. By double-clicking, you can label the position of the cursor at any point of interest.
Split Display	Switches the display between the Overlay Display and the Split Display modes.
Set Anchor Point	Places an anchor point in the graph of the selected spectrum, enabling you to drag your spectrum horizontally or vertically.
Add Text	Displays the General tab of the Label Properties dialog, which enabling you to add text to a graph.
Add Bitmap Image	Displays the Bitmap tab of the Label Properties dialog, enabling you to add an image to a graph.
Print	Prints the current display to the attached printer.
Copy To Clipboard	Copies the current display to the clipboard.
Properties	Displays the Graph properties dialog, enabling the graph properties to be defined.

Copy to Clipboard:

When you select **Copy to Clipboard**, 2 copies of the graph are placed on the clipboard a bitmap and an enhanced metafile.

A bitmap of the graph is the same dimension as the current graph window. This is an EXACT copy of what is shown on the screen. It will include the information page. The graph will have the same resolution as the screen. This means, the bigger the graph is shown in UV WinLab, the bigger the bitmap and the better the resolution. This format should be used for screen reports.

The enhanced metafile (EMF) is the same physical dimension as the screen but it is produced for the default printer rather than the screen and is therefore of much higher resolution. This is the format that should be used if you wish to print the report as the curve will appear much smoother than if the bitmap is used. There is a disadvantage with the metafile. If you stretch the image, you must keep the same aspect ratio as the original image otherwise you will get overlapping characters if the image is stretched in the vertical direction but not the horizontal direction. If you perform a non-proportional stretch with a bitmap, the characters just become stretched on one direction (rather than overlapping) but the curve starts to become 'blocky'.

You need to use the **Paste Special** command within Microsoft Word to see the different formats. Microsoft Word gives the options: **Picture (Windows Metafile)**, **Bitmap**, **Device Independent Bitmap**, and **Picture (Enhanced Metafile)**. If you use **Paste** rather than **Paste Special**, Microsoft Word will use the **Picture** format by default.

What menu items are available when I right-click on the spectral line on the graph?

The following menu items are available:

Set Anchor Point	Places an anchor mark allowing free hand scrolling of the graph
Select Only this curve	Selects the highlighted curve as the current curve
Label This Point	Displays a Label Properties dialog for adding a text label
Label Peaks	Displays a Label Peaks dialog to determine the settings and formatting of the peak labels.
Save as Binary	Displays a Save As dialog, allowing the spectrum to be saved as a .sp file to the selected directory.
Save as ASC	Displays a Save As dialog, allowing the spectrum to be saved as a .asc file to the selected directory.
Remove Curve	Clears the current curve from the view.
Status	Displays the Status dialog
Appearance	Displays the Appearance tab of the Graph Properties dialog

How do I display the status of a graph?

- Click the right mouse button on the spectral line or the name of the curve and from the menu select **Status**.

The Status dialog is displayed. Each of the tabs is described below.

Sample tab

The Sample tab allows you to select which sample (of those currently displayed) you wish to view the status of and enter a description and/or comments.

1. If you wish to view the status of a sample other than that listed by default, select the sample from the drop-down **Properties of** list.

The dialog displays the Date the spectrum was collected and the Analyst who collected it.

Both these fields are for information only and cannot be edited. This information is not provided if the spectrum was imported.
2. You can enter a **Description** of the sample and any **Comments** that you wish to associate with the sample in the appropriate fields.

The comments can be viewed via a Samples Query.

NOTE: If you add a description or any comments, you must click **OK** (the dialog closes). If you do not click **OK**, and then you choose another spectrum, the information previously entered is lost.

Scan tab

The Scan tab details the **Ordinate** used for the method, the **Interval** and **Scan Range** depending on the type of method used to generate the data. All these fields are for information only and cannot be edited.

- If you wish to view the status of a sample other than that listed by default, select the sample from the drop-down **Properties of** list.

Instrument tab

The Instrument tab lists all the available parameters used for collecting the spectra. The parameters are for information only and cannot be edited.

History tab

The History tab displays a table of operations conducted with the spectra.

- Highlight the **Operation** whose details you wish to view from the Summary list.

The History record displays the current history record number out of the total number of history records available.

- Click **Next >>** or **<<Prev** to select the next or previous record from the summary.

For each **Operation** conducted on the spectrum, the **Analyst** who performed the operation, and any **Parameters** used to perform the operation are listed. These are all for information only and cannot be edited.

How do I add a spectrum to the graph for comparison?

1. Click Add Reference.

The Add Reference Spectrum dialog is displayed.

Spectra can be imported from a database, a file or the current task.
 2. Select the spectrum or spectra and click **Open**.
- The spectrum is displayed on the graph.

NOTE: Spectra added via the **Add Reference** button are for visual comparison only. They do not become part of the task. If the task is opened again via a Query, the spectra added via **Add Reference** are not present. Also, the spectra (added via **Add Reference**) are not processed through the processing chain.

How do I alter the graph properties?

1. Click the right mouse button (right-click) on the graph background, and from the menu select **Properties**.

OR

Right-click on the spectral line on the graph, and from the menu select **Appearance**. The Graph Properties dialog is displayed.

2. Edit the settings as required.

The settings available on each tab are described below.

General tab

The General tab allows you to enter the Title and Description of the graph:

3. Enter a **Title** for the graph.
4. Enter a **Description** of the graph.

Axes tab

The Axes tab allows you to define the axes for a selected curve or all curves.

5. Select the graph (or select **All Curves**) from the **Properties of** drop-down list.
6. Enter the **Top** and **Bottom** values for the Y Axis, and select the **Units** for the Y Axis from the drop-down list.
7. Select **Autorange on new data** if you wish the software to automatically set the range to fit the graph to window, setting the top and bottom values to the required values.
8. Enter the **Left** and **Right** values for the X Axis and select the **Units** for the X Axis from the drop-down list.
9. Select **Autorange on new data** if you wish the software to automatically set the range to fit the graph to window, setting the left and right values to the required values.
10. Select **Overlay** or **Split** Display Mode.

NOTE: The **2000 cm-1 scale change** checkbox is intended for infra-red spectra only.

Appearance tab

- If you wish to display gridlines in the graph window, select **Enable Gridlines**.

Element displays a drop-down list of the text objects that are in the graph window. To change the settings for a particular element:

11. Select the **Element** from the drop-down list.
12. Select the Font **Size** from the drop-down list.
13. To change the color of the text, click **Color** and select the color from the dialog displayed.

To change the color of a curve displayed in the graph window:

14. Select the curve from the drop-down list of **Curves** currently displayed.
15. Click **Color** and select the color or create a custom color from the dialog displayed.

Advanced tab

16. Select whether to Hide X Axis Units, Hide Y Axis Units, Hide X Axis Numbers, Hide Y Axis Numbers and/or Hide Information Pane.

NOTE: If you hide a feature of a graph, such as the axis units, the information will not be included if you use the **Copy to Clipboard** function.

17. Select which items to include in the Tool Tip when the cursor is positioned on the spectral line.
One or more of the options Show Curve/Group Name, Show Co-ordinates, Show Data Point Name can be selected.
18. Select whether to display **Data Point Legends**.
This applies to Scatter Plots e.g. Calibration Graphs.
19. Select the name of the spectrum from the **Properties of:** drop-down list.
This is the spectrum whose properties you will define in the lower part of the Advanced tab.
20. For the individual data points, select **Display Never**, **Display Always**, or **Scale Dependent**.
21. Select the **Shape** of the points. Select **Dot**, **Square**, **Triangle**, **Cross** or **Diamond** from the drop-down list.
22. To change the color of the shape click **Color** and select the required color from the dialog.
23. Select the method by which the spectral line is plotted point to point. Select **None**, **Linear** or **Cubic** from the drop-down list.
24. Select the **Size (1–10)** and **Style (Solid, Dot, Dash)** of the line using the drop-down lists.

How do I edit a graph label?

The Label Properties dialog is displayed when **Add Text** or **Label This Point** is selected from the menus obtained by clicking the right mouse button (see menu items above).

The Advanced tab is only available when adding a label to a spectrum point.

General Tab

1. Enter the text of the label in the Text field.
2. Select **Horizontal** or **Vertical** orientation of the label.
3. If you wish to include the time or date in the label select **Time** or **Date** from the Insert drop-down list. (These options are available when the General Tab is accessed via **Add Text**).
 <Current Time> or <Current Date> is displayed as appropriate in the Text field, but the actual time and/or date is displayed on the graph.
 If you access the General Tab via Label This Point, you have the options: Curve Name, Description, X Value, X Units, Y value, Y Units.
4. Select the font size.
5. If you require a border around the text, select **Draw Border**.
6. To change the color of the text, click **Color**.
 A standard font color selection dialog is displayed.

Advanced Tab

7. Select **Show Tie Line** to display a line linking the position of the label to the associated data point on the spectrum.
8. Select to anchor the label Relative to tie point or Relative to screen.

NOTE: The Label Properties Dialog is also displayed if you right-click on the background of the graph and select **Add Bitmap**. The Bitmap Tab only is displayed.

How do I add a Bitmap Image to the graph?

You can place an image, such as your company logo, into the graph using the Add Bitmap Image command.

1. Right-click on the background of the graph and from the menu select **Add Bitmap Image**.
 The Label Properties dialog is displayed.
2. Click **Browse**, and from the Open dialog select the bitmap to add to the graph.
 The name of the image is displayed in the Filename field.
3. Select **Transparent Background** if you wish to set the background color of the bitmap to be transparent.

How do I add peak labels to a spectrum?

1. Right-click on the spectral line, and from the menu select **Label Peaks**.

OR



Click on the name of the spectrum and then click

The Label Peaks dialog is displayed.

2. Select the required settings on the two tabs as described below:

Settings tab

Threshold	Enter the Threshold value (in Absorbance) for determining a peak.
Start, End	Enter the Start and End values for the range over which the peaks are detected.
Peaks, Bases	Select whether to find Peaks and/or Bases .
Display All peaks, Only the top X peaks	Select whether to display All peaks , or Only the top X peaks . If Only the top X peaks is selected, enter the number of peak labels to be displayed.
Abscissa, Ordinate	Select whether to display the Abscissa and/or Ordinate values.

Formatting tab

Font size	Select the Font size of the peak labels.
Abscissa Significant figures, Decimal places	For the Abscissa value, select Significant figures or Decimal places and then select the number from the appropriate drop-down list.
Ordinate Significant figures, Decimal places	For the Ordinate value, select Significant figures or Decimal places and then select the number from the appropriate drop-down list.

3. Click **OK**.

The dialog closes and the peak labels are displayed.

NOTE: The position of a peak label on the graph can be moved. Position the mouse over the peak label. The mouse pointer changes to a 4-headed arrow. Drag the peak label to the new position.

If I delete a graph of a sample from the Sample Table is the data removed from the table?

No. The graph is removed from the display but the data is not removed from the table.

The graph can be re-displayed by double-clicking on the appropriate row in the Sample Table.

NOTE: If spectra have been added via the **Add Reference** button and subsequently removed from the graph, they must be added again via the **Add Reference** button.

What would I use the event marker for?

The event marker is only applicable for Timedrive methods. It allows you to add a label to a graph to show when you perform an action such as adding a reagent.

Options dialog

What settings are available on the Options dialog?

The settings are Ordinate precision for Instrument Status Bar and Wavelength Program table and Interpolation.

1. Within the Workspace, from the Tools menu select **Options**.

The Options dialog is displayed.

2. Select the settings required:

Ordinate precision for Instrument Status Bar and Wavelength Program table	Select Significant figures or Decimal places and then select the number from the appropriate drop-down list.
Interpolation	Select Linear interpolation between points .

NOTE: If **Linear interpolation between points** is selected, this setting will apply to all new methods created by a particular Windows user (not UV WinLab user – the option will apply to all UV WinLab users logged on to that Windows user account). Existing methods will not be changed. If **Linear interpolation between points** is not selected, linear interpolation will not be applied to all new methods created by a particular Windows user.

Audit Trail

NOTE: The method Audit Trail is only available when a method has been locked.

How do I view the Audit Trail for a method?

The Audit Trail is viewed from the Workspace.

1. Open the method whose Audit Trail you wish to view.
2. From the Tools menu select **Audit Trail**.

The Audit Trail report is displayed in the Print Preview window of Communiqué Report Creator. For further information on printing the report, see Communiqué Report Creator.

What does the Audit Trail report contain?

The report contains details on the Method Revision, Data Collection Settings, and Reporting Settings.

Event Logs

Event Logs are available for Methods, Tasks and Samples.

The Event Log for a Method or Task records each event for the method or task. For example, if the Method is created by the Method Wizard this is the first event recorded in the Event Log.

For each event, the Method Event Log records the Event, Revision Number, Date, User, Full Name, Reason and Comments.

For each event, the Task Event Log records the Event, Date, User, Full Name, Reason and Comments.

For each event in the Enhanced Security version of UV WinLab, the Sample Event Log records the Event, Date and Time, User, Reason and Comments. The Event Log starts recording after data has been collected and the Task has been saved. The first entry is the Save.

In the Standard version of UV WinLab, the Sample Event Log only records any comments associated with the sample.

Method Event Log

How do I view the Method Event Log?

The Event Log is viewed from the Workspace.

1. Open the method whose Event log you want to view.
2. From the Tools menu in the Workspace select **Event Log** and then select **Method** from the submenu.
The Method Event Log is displayed.

What events are recorded in the Method Event Log?

The Method Event Log records the Created by New Method, Saved, Locked, Unlocked, Reviewed and Approved Events.

Task Event Log

How do I view the Task Event Log?

The Event Log is viewed from the Workspace.

1. Open the task whose Event log you want to view.
Double-click on the task within the Explorer to open it in the Workspace.
2. From the Tools menu in the Workspace select **Event Log** and then select **Task** from the submenu.
The Task Event Log is displayed.

What events are recorded in the Task Event Log?

The Task Event Log records the Saved, Reviewed and Approved Events.

Sample Event Log

How do I view the Sample Event Log?

The Sample Event Log can be viewed via the Sample Table, the Results Table or the Standards Table.

1. Select the sample from any of the above tables.
2. Click the right mouse button and select **View Sample Event Log**.

The Sample Event Log is displayed.

What is the Sample Event Log?

In the Enhanced Security version of UV WinLab:

The Sample Event Log records all events connected with the sample. For example, if a description is added after the sample has been run, this event is recorded in the Event Log. The original value and new value are recorded.

The Sample Event Log records the Event, Date and time, User, and Reason/Comment.

NOTE: If a field has been changed from being empty to containing a value, empty quotes "" are used to show that the field was initially empty. For example, if the description field was empty and then changed to read Batch 1, the Event Log would record: Description changed from "" to "Batch 1".

In the Standard version of UV WinLab:

The Sample Event Log only records any comments associated with the sample.

Welcome to Communiqué

Welcome to Communiqué

Communiqué is the report designer tool supplied with your PerkinElmer application software and it enables you to create and edit your own report templates.

Communiqué is purely a design tool used by other PerkinElmer applications as a report template creator. From your main software application you choose to create a new template or edit a current template and then Communiqué opens.

Once in Communiqué, you use the layout tools and the data objects to create report templates for use with the parent application.

Communiqué is a very powerful report creator and the rest of this help file will help you get the most from it.

Tell me about the Main window

The **Title Bar** at the top of the window displays the name of the product and the standard Windows buttons - minimize, maximize and close.

The **Menu Bar** below that contains the menu commands that enable you to work with Communiqué, as described in the [Understanding the Menu Commands](#) topic.

The **Toolbar**, below the menus, contains icons for standard interactions that will be performed frequently, enabling quick access to these commands. All these commands are also available from the menus. Hover the mouse over the tools below for details:



The grey **Banner** below the toolbar informs you which template is being edited and which element of the template is currently active. For instance:

Paraben : Revision No 1 : Page Type 1 : Processed Spectrum

The **Tree View** on the left hand side enables quick access to items on the template. Selecting an item on the Tree displays the relevant view in the Main Pane. Sub-branches can be expanded and collapsed as required. Pages and items on the tree can be selected and renamed, and the page order can be changed by dragging them with the mouse. The Tree changes dynamically depending on selections made on the Template level and as objects are added to the template. Further information is in [Working with the Tree View](#).

The **Main Pane**, in the center of the window, displays views of the template enabling you to position items as required. This includes [rulers and a grid](#) that can help you place objects appropriately.

The **Objects Toolbox** on the right is split into three groups: Layout Tools, Data Objects and Custom Objects.

- [Layout Tools](#)are either tools used for laying out objects, or are visual items to be displayed on the page .
 - [Data Objects](#)represent the items from your application software to be displayed in the report.
 - [Custom Objects](#)are special objects you have created yourself, either where you want to keep re-using particular groups of objects or where you have created special equations.
- The **Status Bar** at the bottom of the window is used to provide feedback, for instance when an object is being sized or placed.

Working with Templates

The output of Communiqué is a Report Template and there are a number of things you can do with a template object:

How do I set up a template?

By selecting the name of the template in the Tree View, as shown in the picture, you access the settings for that template.

The controls on the Template page have the following functions:

The **Description** will be saved with the template, enabling you to give extra information about what the template was designed for, as information for others who may want to use the template.

The **Paper size** and the **Portrait** or **Landscape** orientation determine the size and shape of the template.

Set Margins enables you to set up the margins on the template, as described in [Working with Margins](#).

You can select whether to have a **Header** and/or **Footer**, areas that will be repeated at the top or bottom of every page, and whether to have them **Different on page one**, as described in [Working with Headers and Footers](#).

The **Grid units** and choices of whether to **Display grid** and **Snap to grid** are discussed in [Using Rulers and the Grid](#).

How do I open another template?

It is possible to work on more than one template at a time. This enables you to cut, copy and paste objects between templates. It also means that you can work on more than one template at a time.

- To start a new template, select **New** on the File menu.
A new template is added to the tree view.
- To open a template, select **Open** on the File menu. Select your template from the Open dialog.
The details of the template are added to the tree view.

NOTE: Only templates associated with the currently defined data model by name (the version does not matter) are shown.

- To close a template, select the required template and then select **Close Template** from the File menu.
If no changes have been made since the template was last saved, the template closes.

NOTE: If there are unsaved changes, you are asked whether you wish to save the changes before the template closes. Select **Yes** to save the changes before closing the template, or select **No** to close the template without saving any changes.

How do I import or export a template?

Importing

You can import a template or custom object that was exported from another copy of Communiqué.

1. Select **Import** and then **Template** or **CustomObject** from the File menu.
2. Use the file selector to locate the required template or object.

NOTE: You can select multiple files to import.

NOTE: If you import a template that uses fonts not available on your PC the template will use a different font as set by the Windows operating system. To import a template, select the required template and then select **Close Template** from the File menu. If no changes have been made since the template was last saved, the template closes.

Exporting

You can export a template or custom object to disk so that it can be used on another PC.

NOTE: Before you can export a template, you must save it.

3. Select **Export** and then **Template** or **Custom Object** from the File menu.
4. On the file selector, choose where to save the item and give it a name.

How do I lock a template in the Standard security mode?

Templates can be locked with a password using the  icon in the Print Preview. A locked template cannot be changed.

1. Open the required template in the Print Preview and click . The Lock Template dialog is displayed.
2. Enter your **Password** in the **Password** field and enter it again in the **Repeat Password** field.

NOTE: The password must be at least 8 characters.

3. Click **OK**.
The template is locked.
When the template is opened in future it will be read-only until it is unlocked.

How do I lock a template in the Enhanced security mode?

In the Enhanced Security mode templates are locked (made read-only) when the template is "approved". An approved template cannot be unapproved (unlocked).

See the procedure on [approving a template](#) for more information.

How do I unlock a template in the Standard security mode?

A template that has been locked using a password can be unlocked using the  icon in the Print Preview.

1. Open the required template in the Print Preview and click . The Unlock Template dialog is displayed.
2. Enter your **Password**.
3. Click **OK**.

Assuming the password was correct, the template is unlocked.

NOTE: In the Enhanced Security mode templates are locked (made read-only) when the template is "approved". An approved template cannot be unapproved (unlocked).

How do I save a template?

- To save a template, simply select **Save** from the File menu.
If it is a new template the Save dialog will be displayed enabling you to name the template, otherwise the template will simply be saved, overwriting the previous version.
Depending on the security settings of the parent application, you may need to electronically sign the save.
- To save a template with a new name, select **Save as** from the File menu.
The Save dialog is displayed, enabling you to give the template a new name.
Depending on the security settings of the parent application, you may need to electronically sign the save.

Working with Margins

The left and right margins are non-printing areas of the template, while the top and bottom margins can only hold a [header or footer](#).

You can not place items within the margins, and objects will not spill over into the margins when reports are created.

The margins appear on the page as red lines, as shown in the picture here. The blue shading shows the header and footer, which can only be set within the top and bottom margin, so you must create these margins before you can work with a header or footer.

The margins can be set both by using a dialog and by using the mouse.

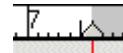
NOTE: For best results, you should set your margins before laying out a template. If margins are moved and cross over an object, the objects may not print properly.

How do I set the margins using the dialog?

1. Select the top level of the template.
2. Click Set Margins.
The Set Margins dialog is displayed.
3. Select the values required and click **OK**.
The Margins are shown as red lines on the page layout.

How do I set the margins using the mouse?

Each margin has a triangular marker that can be dragged with the mouse to change the margin size:



NOTE: These are only available if the [Rulers](#) are switched on.

Working with Headers and Footers

When items have been added to a header/footer they appear in the blue areas of the Page Type level. You can modify headers and footers on a Page Type basis. A Header and/or Footer item will appear at the Page Type level of the tree for each page type that uses a header and/or footer. Furthermore, each page type header and footer may also include a special version for the first page of that page type.

However, if an item is placed at the Page Type level and overlaps the header or footer it is not 'on' the header/footer and does not get replicated on each page.

How do I switch on the Header or Footer?

1. In the Tree View, select the required template name.
A page is displayed that includes the ability to switch on headers and footers.
2. Select **Header** and/or **Footer** to include headers and/or footers in the template.
When Header is checked a "Header" item appears in the Tree. When Footer is checked a "Footer" item appears in the Tree.

NOTE: The header and footer settings may be changed for each page type and the contents of the header and or footer may be different for each page type. See Working with Page Types for more information.

How do I have a different Header/Footer on the first page?

1. In the Tree View, right mouse click on the required Page Type and select **PageTypeProperties** from the pop up menu that appears..
The Page Type Properties dialog appears.
2. Select **Specific to this page type** to customize the header and footer for this page type only.
3. Check **Header** and/or **Footer** to include headers and/or footers in the page type.
4. Check **Different on page one** to enable you to have the Header/Footer on page one different than all the other pages in the page type.
The Page Type Header and/or Footer nodes in Tree are expanded to include a Page Type First Header and/or Footer sub node.
5. Click on the Page Type First Header and/or Footer sub nodes to customize the header and/or footer settings.

NOTE: If you create and populate a header/footer and then switch on **Different on page one** the original header/footer you set up will now only be available on page two onwards and you will need to set up the header/footer for page one as required.

How do I place objects in the Header/Footer?

1. To add items to the Header or Footer, select the appropriate **Header** or **Footer** from the Tree view.
The selected section (Header or Footer) is then displayed in white and items can be added.
2. To return to the main section of the template select the page or an item on the page from the Tree view.

What do the red lines in the Header and Footer mean?

Red lines are used to show the position of the margins.

Elements and objects added to the Header or Footer must be placed inside the margins.

Working with Page Types

While a simple report may consist of a single page type, you may want to have different types of pages set up. For instance if you wanted a multiple page report to have a title page, then the report pages and then a summary page, you would need three page types.

Page types should not be confused with actual pages of the finished report, as when a report is generated a particular page type may create more than one page of actual report, either because the objects on the page take up more room than expected and spill onto another page, or because you have set up a repeating [section](#) that gets used for reporting details of multiple samples one at a time, or a table has expanding rows.

Selecting a Page Type on the tree shows the overall design for the whole 'page', with red lines showing the position of the margins.

NOTE: Each Page Type has its own Properties dialog for setting up features such as individual header, footer, print orientation and margins.

How many Page Types can I have in my Template?

You can have a maximum of 99 page types in your template.

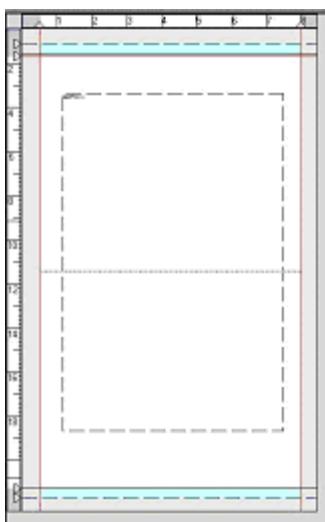
What is displayed in the Tree view?

Each page type is added to the Tree view and is numbered sequentially. For example, Page Type 1, Page Type 2.

- These can be renamed by right mouse clicking on the required page and selecting **Rename page**, then, on the dialog, enter the **New name**.

How do I work in a design page?

If you set the **Number of pages** parameter in the **Page Type Properties** dialog to a value greater than 1, the Page Type View will be updated to reflect that value. The Header and Footer display (if any) will remain unchanged but the area between will indicate the number of pages set as shown below.



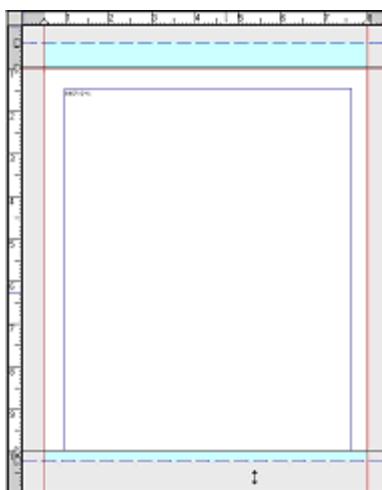
Note that the working design area displayed (the region between the header and the footer onto which you can drop objects) will be equal to twice the normal height of the template page. That is, the space occupied by the footer for the first design page and the header for the second design page will not be included. This is so that the design area accurately reflects the space that is available on the two pages.

The intervening header and footer will not be shown in design mode, to permit the manipulation of section contents across the page break, but of course these would be present in a preview or printed report. The headers and footer for multiple design pages will always be identical.

The break between the pages will be indicated with a distinctive line style. In this example a section is shown covering most of the two pages. The primary need for this new feature is to allow the generation of reports where the required contents of a section are too large to be accommodated on a single design page.

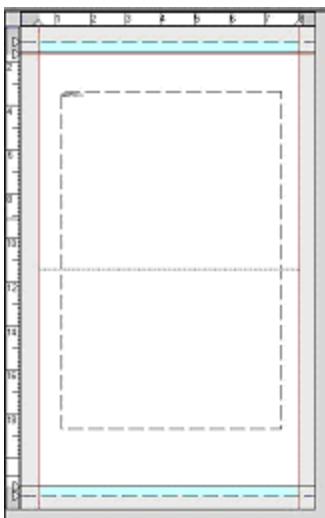
How do I add an additional design page?

1. Drag the lower edge of a section (existing or newly dropped on the page) off the bottom of the page layout (even though the section outline does not extend below the top of the footer, the drag cursor indicates the section boundary can be dropped below the page). This means below the page footer (if one exists) and hence implies that the current zoom factor allows the bottom of the page to be seen (see example below).



2. When you release the mouse button, a confirmation dialog is displayed, asking **Do you want to create an additional design page for this section?**
3. If you respond **No** then the section will be sized to the bottom of the existing page and no additional design page will be created.

If you respond **Yes** then an additional design continuation page is created and the display changes to the following format.



How do I select a Page Type in my Template?

- Select the required Page Type from the Tree view.
The selected page is highlighted in the Tree view and the contents of that page are displayed, enabling you to work on the page.

How do I Center and Zoom?

1. Select **Center and Zoom Options** from the Actions menu and set your desired parameters.

2. Click on **Center and Zoom** in the Actions menu to activate it.

When the **Center and Zoom** option is active selecting an element of a Page Type in the Tree View causes the display to update so that the selected object is at the center of the design window and displayed at the zoom level defined in the Center and Zoom Options. Center and Zoom is only used when selecting an object using the Tree View; selecting an object by clicking on it on the page does not change the current zoom level (however that was set).

The zoom factor defined in the Center and Zoom Options is completely separate from the 'global' zoom factor defined in the **Zoom** dialog displayed from the **Zoom** in the View menu command or using the toolbar control.

NOTE: The zoom behavior is an optional feature of Center and Zoom, which can be disabled separately from the basic Center and Zoom menu option or toolbar button.

Centering the object will not be possible, or necessary, at some zoom levels. For example if the whole page is displayed then the object may not be 'centered' depending on its location on the page but it will be visible. The page display will never introduce space above the header in order to center an object. True centering behavior will only be required at higher zoom factors, where only a part of the page is visible.

When the Center and Zoom option is turned off the page display will revert to the 'global' zoom factor.

Working with the Tree View

The tree view on the left hand side of the window enables quick access to both different parts of the template, and to items placed on the template. Clicking on an item 'selects' the item so you can format it.

When the tree has sub-branches they can be opened and collapsed by clicking on the + or - sign.

Why does the tree keep changing?

The tree changes dynamically depending on selections made on the Template level and on the objects added to a page:

- Selecting **Header/Footer** on the Template level adds **Header/Footer** to the tree under the Template level.
- Selecting **Different on page one** on the Template level adds **Page One** to tree under Header and/or Footer level.
- Selecting additional page types on the Template level adds extra Page items to tree (Page Type 1, Page Type 2, etc.) under Template level.
- Adding items to a page adds the objects to the tree under the relevant Page level, at multiple levels such that Sections, Tools and Objects are at different levels.
- Opening a second template will add the details for that template to the bottom of the tree.

How do I Rename items displayed on the tree?

Pages and items on the Tree can be selected and re-named using the **Rename** command on the Edit menu.

You can also right-click on a page and select **Rename page** on the context menu, or right-click on an item and select **Rename item** on the context menu.

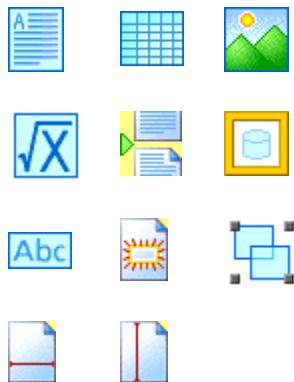
How do I add, remove, and move pages?

There is a context menu available for working with pages on the tree, displayed by right-clicking in the appropriate place:

- **Insert New First Page** inserts a new page type at the top of the tree.
- **Insert New Page Before** inserts a new page before the currently selected page.
- **Insert New Page After** inserts a new page after the currently selected page.
- **Insert New Last Page** inserts a new page type at the end of the tree.
- **Move Page Up** moves the selected page up one position on the tree.
- **Move Page Down** moves the selected page down one position on the tree.
- **Remove Page** removes the selected page.
- **PageTypeProperties** opens the Page Type Properties dialog from where you can customize margins, page orientation, headers, and footers.
- **RenamePage** opens the Rename dialog from where you can rename the Page Type.

Using the Layout Tools

The Layout Tools section of the Object Toolbox holds a set of tools used to create templates. Click on a tool below to find out how it works.



Text Block

NOTE: When typing text into a Test Block, the text may not display properly at zoom levels less than 100%. Set the zoom to 100% to see the proper text.

Text blocks can be used to create:

- a block of text you create now and place on the page
- a block to hold data objects
- a block of text with inserted data objects.

How do I add a Text Block to a Template?



1. From the Layout Tools group in the Objects Toolbox click .
The mouse pointer changes to  to show that the Text Block is selected.
2. Click on the page where you want the top left hand corner then, holding down the mouse button, drag the mouse to the bottom right corner and release the mouse button.
The text block is added to the page and the name is added to the Tree View.

A shortcut for creating text blocks:

- Right-click on a blank area of the template page and chooses the **Add Text** command from the context menu.
- A text block will be automatically created at the point of the right-click (on the left, top of the text block). The width of the text block will be such that it fills the space between the point where you double-clicked and the next object to the right (or the margin). It will initially contain no default text but the insertion point will appear at the left hand side of the block, ready for text entry. When you enter some text and move focus to another object the text block will resize to fit the text actually entered. If you move focus from the text block without entering any text then the object will be automatically deleted.

How do I use Text and/or Data Objects in a Text Block?

To enter text directly:

- Click the mouse inside the inner box of the Text Block and then type in the text.

To place a data object:

1. From the Objects Toolbox select Data Objects.
All the available Data Objects are listed.
2. Select the required Data Object.



The mouse cursor changes to  to show that a data object is selected.

3. Click the mouse in the text block.

The object is added and appears within squared brackets within the block. The actual information is inserted when the report is generated.

To insert data objects within typed text:

- Position the cursor within the text in the Text Block where the object is required and then insert the data object as above.
For example: The following results were created by [analyst name] using the task [task name].

How do I move or resize a Text Block?

The Text Block can be moved and resized using the mouse or the **Properties** command.

Using the mouse to resize the Text Block:

1. Click on the Text Block to select it.
The text block border changes to blue to show that it is selected.
2. Drag the corners or sides as required to resize.
The mouse changes to a two-headed arrow when placed over the edge of the text block.

Using the mouse to move the Text Block:

1. Click on the Text Block to select it.
The text block border changes to blue to show that it is selected.
2. Position the mouse inside the frame.
The mouse changes to a four-headed arrow.
3. Click and drag the block to the new position.

Using the Properties command to move and resize the Text Block:

- From the Format menu select **Properties**.

OR

Right-click on the Text Block and select **Properties**.

The Format Text Block dialog is displayed. In addition to resizing and moving the Text Block, the dialog enables you to determine the paragraph style and the font for the text entered in the block.

How do I format a Text Block?

- A Text Block can be formatted by selecting **Properties** from the Format menu

OR

Right-click on the Text Block and select **Properties**.

The Text Block Properties dialog is displayed. In addition to resizing and moving the block, the dialog enables you to determine the way text will look in the block.

On the Block tab:

- Use the Height and Width controls to size the block.
- Use the In from left and Down from top controls to position the block on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

- Use Oversize text to determine what happens when a text object placed in the block is too large for the block:
- Select **Truncate** if you wish any text that does not fit within the Text Block to be truncated.
- Select **Re-size block to fit** if you wish the Text Box to resize to accommodate all the text, and then select **Vertically** or **Horizontally** to decide which way the box will re-size.
- Use the **Fill** control to change the color inside the block.

On the Paragraph tab:

- Use the **Horizontal** and **Vertical** controls to determine how the object is aligned inside the block.
- Use the **Spacing** controls to determine the spacings used in the frame:
- Enter a value between **0** and **99** for the spacing **Between lines**.
- Enter a value between **0** and **99** for the spacing **Between paragraphs**.
- Use the **Preview** to see how the paragraph will look.

NOTE: If your template view is less than 100%, the text may display differently in the print preview window. This is a function of the computer font display.**On the Font tab:**

- Use the Font, Size, Style, Color, and Underline controls to determine how the text will look.
- Use the Preview to see how the text will look.

On the Border, Caption, and Notes tabs

- Set your Border properties, Captions, and any Notes.

How do I delete a Text Block?

1. In the Tree view, click on the Text Block to select it.
The border of the block turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The Text Block is deleted.

How do I delete items in a Text Block?

To delete text within the Text Block:

1. Highlight the text within the Text Block using the mouse.
2. Press DELETE.
The text is deleted.

To delete a Data Object embedded in a Text Block:

1. Highlight the Data Object within the Text Block using the mouse.
2. Press DELETE.
The object is deleted.

NOTE: The  tool and **Delete** on the Edit menu or context menu will delete the whole block.

NOTE: If you do make a mistake, use the **Undo** command on the Edit menu or click .

How do I rename a Text Block?

By default the Text Blocks are named TEXTX , where X is numerically incremented (TEXT1, etc) for each new Text Block added to the template. You can rename the Text Block if you wish.

1. Right-click on the Text Block in the Tree View and select **Rename Item**.
The Rename dialog is displayed.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the Tree view and within the Text Block is updated.

Table

Tables can be created where all the properties can be defined within Communiqué but the data comes from the Parent Application when a report is generated.

How do I add a Table to a Template?

1. From the Layout Tools group in the Objects Toolbox click .
The mouse pointer changes to  to show that the Table is selected.
2. Click on the page where you want the top left hand corner.
A default table with two rows and four columns will be placed on the template and added to the Tree View.

How do I add or delete Table rows and columns?

There are two ways you can add or delete rows and columns. The first procedure, listed below, shows you how to add or delete rows and/or columns using the Properties dialog for a selected table. The second procedure shows you how to use the Insert command to add or rows and columns in specific positions.

To add or delete rows and/or columns using the Properties dialog:

1. Select the table and click **Format > Properties**.

OR

Right mouse click on the table and select **Properties** from the pop up menu that appears.

The Table Properties dialog appears.

2. On the **Table** tab enter a new value for the **Numberofcolumns** and **Numberofrows**.

NOTE: Increasing the value will add a row to the bottom of the table or a column to the right-hand side of the table. Decreasing the value will delete the bottom row of a table or the column on the extreme right.

3. Click **OK**.

To add rows and/or columns in other positions or to delete specific rows and/or columns:

1. Select one or more rows or columns and right mouse click.

A pop up menu appears.

2. Select **Insert > Columns**, to have as many columns as are selected are inserted to the left of the selected columns. The new columns are blank.

OR

Select **Insert > Rows**, to have as many rows as are selected are inserted above the selected rows. The new rows are blank.

How do I add Text and/or Data Objects to a Table?

To enter text into a cell:

1. Right mouse click on an empty cell and select **Insert > Text** from the pop up menu that appears.
An empty text block is inserted into the cell.
2. Type in the text block.

OR

From the Layout Tools group in the Objects Toolbox click .



The mouse pointer changes to  to show that the Text Block is selected.

3. Click in the required cell.
The text block is inserted into the cell.
4. Type in the text block.

To place a data object into a cell:

1. From the Objects Toolbox select **DataObjects**.
All the available Data Objects are listed.
2. Select the required Data Object.

The mouse changes to  to show that the object is selected.
3. Click the mouse in the cell.
The object is added to the cell. The actual information is inserted when the real report is generated.

NOTE: If a table is grouped with other objects you won't be able to change the table without ungrouping it.

How do I move or resize a Table?

The Table can be moved and resized using the mouse or the **Format** command.

Using the mouse to resize the Table:

1. Click on the Table to select it.
The border changes to blue to show that it is selected.
2. Position the mouse cursor on the box at the lower left corner of the table and drag the mouse as required to resize the table.

Using the mouse to move the Table:

1. Click on the Table to select it.
The border changes to blue to show that it is selected.
2. Position the mouse cursor on the box at the top left corner of the table and drag the mouse to move the table.

Using the Format command to move and resize the Table:

- From the Format menu select **Properties**.

OR

Right-click on the Table and select **Properties**.

The Table Properties dialog is displayed. In addition to resizing and moving the Table, the dialog enables you to format the rows, columns and cells, as well as add a caption and notes to the table.

How do I format a Table?

- A Table can be formatted by selecting Properties from the Format menu

OR

Right-click on the Table and select **Properties**.

The Table Properties dialog is displayed. You can format the whole table, rows, columns, or individual cells.

On the Table tab:

- Use the **Preferred width** control to set the width of the table.
- Use the **Number of columns** and **Number of rows** controls to set the shape of the table.

NOTE: The actual number of rows will depend on whether the table is set to expand depending on data being loaded. You can define up to 50 rows here and have up to 9999 rows in the generated report.

- Use the **In from left** and **Down from top** controls to position the table on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

- Use the **Borders** controls to determine how the table edges and lines between cells will look.

On the Rows tab:

- Use the **Next** and **Previous** controls to select the required row.
- Use the **Setup** controls to determine the settings for the selected row:
- You can choose to have a **Set height** for the row by switching on the check box and then entering a value.
- For the first row, you can elect to **Repeat first row as a header**.
That row is then repeated at the top of each new page when the table splits over pages.
- Choose whether to **Allow row to break across pages**.
If not selected the whole row will be forced onto the next page if it is too big to fit on the page.
 - Choose **Enable row to expand** to enable the current row to expand.
 - Then select the way the rows will be populated from **Available expansions** by building the required order in the **Expand by** list.
 - Set the **Expansion Count** of the item in the **Expand by** list to set the source of the row count.

On the Columns tab:

- Use the **Next** and **Previous** controls to select the required column and then use the Setup controls to set the **Width** and the **Name** as required.
- Choose whether the current row can **Expand**.
Then select the way the rows will be populated by building the required order in the **Expand by** list.
 - Set the **Expansion Count** of the item in the **Expand by** list to set the source of the row count.
 - Check **Repeat column on each new page** to sets the specified column to be repeated if the table rolls over onto another page at run time.

On the Cells tab:

NOTE: This tab displays the settings for the current selection of cells. Where all selected cells share the same setting for a parameter that value will be shown; where the cells have different settings then no value will be shown.

- Use the **Horizontal** and **Vertical** controlsto set how the contents will be positioned.
- Use the **General formatting** controls to determine the font and fill color used in the cell.
- Use the **Number formatting** controls to determine how numerical data will be displayed.

- Use the **Formatting for a null result** to determine what happens if the required data for a cell is not available.
- Use the **Object fit** controls to determine what happens if the incoming data object is larger than the cell.

On the Caption tab:

See the topic titled [Edit Caption](#) for more information on adding a caption to the table.

On the Note tab:

Enter comments associated with the table. Your notes will be stored with the template and will be included in the "template details" printout but it will not be printed as part of the report.

How do I Filter, Sort, and Group Rows?

You can access the Filter, Sort, Group dialog from the Rows tab and the Columns tab of the Table Properties dialog when an item in the "Expand by list" is selected and the Filter, Sort, Group button is clicked.

On the Filter tab:

1. Check **Enabled Filter** to apply filter conditions to the specified collection.
2. Select a **Filter Type**.
3. Select a **Match** option.
4. Click a **DataItem** cell and select a data item from the drop down list.
5. Click on a **Condition** cell and select an appropriate condition.
6. Click on a **Value** cell and enter the appropriate value.

On the Sort tab:

7. Check or uncheck the **Enable sort** checkbox to determine whether or not sort conditions will be applied to the specified collection.
8. Select a data item by which to sort the collection first from the **Sort by** drop down menu.
9. From the **Type** drop down menu corresponding to the sort by field, select the mode of sorting.
10. If you wish to sort the collection further, use the **Then by** fields and the corresponding **Type** fields to select a data item and mode of sorting.

On the Group tab:

11. Check the **EnableGrouping** checkbox to activate the grouping definition.
12. From the **Group By** drop down menu, select the data item by which to sort the collection first.
13. Select the grouping **Options**.

Note: See [How do I use Statistics Rows?](#) for more information on defining statistics rows for a group.

How do I Filter and Sort Columns?

You can access the Filter and Sort dialog from the Columns tab of the Tables Properties dialog when an item in the "Expand by list" is selected and the Filter and Sort button is clicked.

The behavior of the Filter and Sort tab pages is the same as that described in [How do I Filter, Sort, and Group Rows?](#)

How do I Format Statistics?

On the Object tab:

- Use the **Horizontal** and **Vertical** controls to set the positioning of the object within its frame.
- Use the **Currency symbol**, **Decimal separator**, **Digit grouping character**, and **Negative indicator** to control how your currency is displayed.
- Use **Number of digits after the decimal** to set the number of digits to appear after the decimal symbol.
- See how your settings will look in the **Preview** area.

On the Number tab:

- Use the **Horizontal** control to set the position of the contents across the statistics row. Use the **Vertical** control to set the position of the contents vertically in the statistics row.
- Select the number format using **Significant figures**, **Decimal places**, **Absolute number**, **Scientific** (If On enables exponents), and **Force rounding**.
- Select the **Calculation** option to use raw data or data as formatted in the table for the statistics calculations.

Use raw data calculates statistics on the full original data in the data source. **Use formatted data** calculates statistics on the data as it is shown in the relevant column of the report.

On the Format tab:

- Use **Spacing/Zeros** selections to set how your currency is displayed.
- Include a space between the currency symbol and value.
- Include a 0 before the decimal separator if the value is between -1 and +1.
- Add zeros as required to make up decimal digits to the number defined by **Number of digits after decimal** on the Object tab.
- Use **Ordering selections** to set positions.
- See how your settings will look in the **Preview** area.

On the Font tab:

- Use the **Font**, **Size**, **Style**, **Color**, and **Underline** controls to determine how the text will look.
- Use the **Preview** to see how the number will look.

How do I use Statistics Rows?

You can create special rows in the table that calculate statistics:

1. Select the table and then select **Properties** from the Format menu
OR
Right-click on the Table and select **Properties**.
The Table Properties dialog is displayed.
2. Select the Rows Tab and click **Statistics Rows**.
The Statistic Rows dialog is displayed.
3. In the **Based on** column, select the grouping you want to do statistics on.
The available groups depend on your application.
4. In the **Perform on** column, select the column you want to perform statistics on.
Only columns set to display numerical data will be available.
5. In the **Statistic** column, select the type of statistic to be generated.
6. In the **Display as** column, enter any text you want to appear before the calculated number.
7. Select the Data Type as either **Numeric** or **Currency**.

NOTE: To remove a statistics row, click in the far left column to select the row and then click **Delete row**.

To format statistics rows:

1. If you wish to format a statistics row, select the row and click **FormatStatistics**.
The Format Statistics dialog appears.

2. If you selected a Numeric data type option on the Statistics Rows dialog, then define the **Alignment**, **Format**, and **Calculation** settings from the Format Statistics dialog.

OR

If you selected a Currency data type option on the Statistics Rows dialog, then define the **Alignment**, **Symbols**, and **Digits** settings from the Format Statistics dialog.

3. Click **OK** to close the Format Statistics dialog.
4. Click **OK** to close the Statistics Rows dialog.

How do I set the Expansion Count for Rows and Columns?

The **Expansion Count** button appears on both the Rows and Columns tabs. The **Expansion Count** button is enabled when 'Enables column to expand' is checked and a selection appears in the 'Expand by' drop-down list. Here you can set the index of any parent collection that contains the sub-collection you are about to iterate.

- Selecting **Properties** from the Format menu

OR

Right-click on the Table and select **Properties**.

The Table Properties dialog is displayed. You can format the whole table, rows, columns, or individual cells.

- Choose the **Expansion Count** button.
- Select the Index (Current, First, or Last) for each parent collection to define the number of expanded rows.

How do I delete a Table?

1. In the tree view, click on the Table to select it.
The border of the block turns blue to show that it is selected.
2. Click **X**, or press DELETE, or right-click and from the menu select **Delete**.
The Table is deleted.

How do I delete items in a Table?

To delete a Data Object embedded in a Table:

1. From the tree view, select the data object within the Table.
2. Click **X**, or press DELETE, or right-click and from the menu select **Delete**.
The Data Object is deleted.

To delete text from a text block within the Table:

1. Highlight the text using the mouse.
2. Press DELETE.
The text is deleted.

NOTE: The  tool and **Delete** on the Edit menu or context menu will delete the whole block.

NOTE: If you do make a mistake, use the **Undo** command on the Edit menu, or click .

How do I rename a Table?

By default the Tables are named TABLEX , where X is numerically incremented (TABLE1 etc) for each new Table added to the template. You can rename the Table if you wish.

1. Right-click on the Table in the Tree View and select **Rename Item**.
The Rename dialog is displayed.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the Tree view and within the Table is updated.

Graphic

Graphics, for example a company logo, can be imported into the Template. They can be placed in the Header, Footer or main part of the page.

How do I add a graphic to the Template?



1. From the Layout Tools group in the Objects Toolbox click

A file selector dialog is displayed.

2. Select the required graphic and click **Open**.

The possible graphic files are BMP, DIB, GIF, JPG, WMF, EMF.



The mouse pointer changes to

3. Click on the page to insert the graphic at the required position.

The graphic is added to the page and the name is added to the Tree View.

How do I move and resize the graphic in the Template?

The Graphic can be moved and resized using the mouse or by accessing the **Properties** dialog for the selected graphic.

NOTE: You can access the Properties dialog for a selected object by selecting **Properties** from Format menu, or by right clicking on the object and selecting **Properties** from the pop up menu that appears.

Using the mouse to resize the Graphic:

1. Click on the Graphic to select it.

The graphic border changes to blue to show that it is selected.

2. Drag the corners or sides as required to resize the graphic.

The mouse changes to a two-headed arrow when placed over the edge of the graphic.

Using the mouse to move the Graphic:

1. Click on the Graphic to select it.

The graphic border changes to blue to show that it is selected.

2. Position the mouse inside the Graphic.

The mouse changes to a four-headed arrow.

3. Click and drag the Graphic to the new position.

Using the Properties dialog to move and resize the Graphic:

- From the Format menu select **Properties**.

OR

Right-click on the Graphic and select **Properties**.

The Properties graphic dialog is displayed. From here you can enter the required size, or scale, and position.

How do I format a graphic?

- Select the graphic and from the Format menu select **Properties**.

OR

Right-click on the Graphic and select **Properties**.

The Properties graphic dialog is displayed.

On the Object tab:

- Use the **Size** controls to set the exact size you require.

OR

Use the **Scale** controls to set the size of the graphic as a percentage of its original size.

- Select **Keep aspect ratio** if you want to ensure that the graphic retains its aspect ratio.

When the **Height** is changed, the **Width** resets to keep the original aspect ratio, and vice-versa.

- Use the **In from left** and **Down from top** controls to position the graphic on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

On the Border tab:

See the topic titled [Edit Border](#) for more information on adding a border to a graphic.

On the Caption tab:

See the topic titled [Edit Caption](#) for more information on adding a caption to the table.

On the Note tab:

Enter comments associated with the graphic. Your notes will be stored with the template and will be included in the "template details" printout but it will not be printed as part of the report.

How do I delete a graphic?

1. In the Tree view, click on the Graphic to select it.
The border of the Graphic turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and select **Delete** from the pop up menu that appears.
The Graphic is deleted.

How do I rename a graphic?

By default the Graphics are named GRAPHICX , where X is numerically incremented (GRAPHIC1, etc) for each new Graphic added to the template. You can rename the Graphic if you wish.

1. Right click on the Graphic in the Tree view.
2. Select Rename Item.
The Rename dialog is displayed.
3. Enter a new name in the **New Name** field and click **OK**.
The name is updated.

Equation

Equations can be used to place a result on the page, or populate a cell or column, when a Report is generated.

By using conditional formatting, you can also set up special formatting or actions to occur if the result of the equation fits certain criteria. This action requires a numeric data type.

How are Equations implemented?

Equations, which are populated in the right side of the 'Custom Objects' List in the 'Communiqué Designer', are taken-up from the Template Database Table named 'Equations'. So when any of the equation is dragged and inserted on to the Template Designer the inserted equation will get housed with a unique Equation-Data-Item (a designer object, where the control-type is 'Equation').

While previewing if an equation is encountered, Communiqué will check for the immediate container where the equation is placed and depending on this container the data-item source index mentioned in the Equation Text will be evaluated. All together there are 5-different index styles, which you can specify. They are:

1. **First:** Communiqué will get the 1st index value from the data object when this index text is specified on the data-item source. This can be used in Absolute and Relative situations.
2. **Previous:** Communiqué will get the one back index value as compared to the current index from the data object when this index text is specified on the data-item source. Mainly used on Relative situations where 'Current' is dynamically referred based on the repeating iteration.
3. **Next:** Communiqué will get the one next index value as compared to the current index from the data object when this index text is specified on the data-item source. This is mainly used on Relative situations where 'Current' is dynamically referred based on the repeating iteration.
4. **Last:** Communiqué will get the index value based on the total count of a particular source collection from the data object. This can be used in Absolute and Relative situations.
5. **Current:** Communiqué will get the ongoing running index based on the iterating object (iterating object can be either a Section or a Table or Object Frame). This is specifically used in Relative situations.

Communiqué evaluates all the Equations in Phases based on whether the equation is placed on a Section or a Table and so while looping through all the inside-items of Section/Table if the item is of type 'Equation' the equation text will be evaluated based on the iterating index but if the equation has any Conditional formatting criteria specified then this will be evaluated in next phase i.e. after evaluating the Equation-text for all the Equations on a template (this is done in-order to gather all the dependent equation results for the evaluating condition.)

How do I add an Equation to a Template?

- From the Layout Tools group in the Objects Toolbox click .

The mouse pointer changes to  to show that Equation is selected.

- Click on the page where you want the result of the equation to be displayed. This can be either directly on the page, or within a table. When you release the mouse button, the equation is placed on the page and the Equation Builder dialog is displayed.

How do I create an Equation?

- Enter a **Name** and **Description** for the equation.

The **Name** is used in the **Other equations** list from now on and in the Custom Objects section of the Object Toolbox, and the **Description** becomes the tooltip in these two places.

- Create the **Equation** by typing in the field and/or selecting elements from the three lists below.

To select an item from one of the lists, position the cursor where required in the equation, then simply double-click on the required **Operator**, **Data** or **Other equation** and it appears in the Equation.

OR select an item from one of the lists, position the cursor where required in the equation, then press the space bar on the required **Operator**, **Data** or **Other equation** and it appears in the Equation.

- When a data object is inserted in the equation, the display will show how that object is set up, including indexes within square brackets, [].

By default, the indexing will be set to **Current** for all, for example an equation to create the square of a concentration value would look like:
`Sqr({Samples[Current].Replicates[Current].Conc})`

NOTE: The data object must be a numeric and/or currency object in order for equations to work. Some applications may not have numeric and/or currency objects as part of their data model.

- Clicking on one of the indexes will display the current list of possible indexes, enabling you to select the one required.

The default list for indexing is **Current**, **First**, **Last**, **Next**, **Previous**, **All**, and **New Keyword**, but you can enter your own terms using the **Organize keywords** command on the Tools menu. For example if you have a software application that identifies peaks by a name, then you can use that name for indexing. Indexing can be used for the eight functions of: Sum, Mean, Count, %RSD, RMS, SD, Min, and Max.

NOTE: In most cases the index of **Current** will suffice, other indices are used to construct complex equations or instances where you need to take for example the result of one sample from the sample before it (**previous**), or need to work with a named peak.

- Select the **ResultType**.

Numeric defines the equation result as a numeric value and Currency defines the equation result as a currency value.

6. When you have completed the equation, click **Check** to check the general construction of the equation is correct.
If the equation has been set up correctly, the message **Syntax is correct** will be displayed.
If there is a problem with the equation, the message **Syntax is incorrect** will be displayed.
7. To format the value that will result from this equation, click **Format Result**.
The Numeric Data Object Properties dialog or the Currency Data Object Properties dialog appears depending upon the Result Type you selected.
8. If you selected a **Numeric** Result Type, then select the **Alignment** and **Format** required and then click **OK**.
The formatting will be applied to the calculated result.

If you selected a **Currency** Result type, then select the **Alignment**, **Symbols**, and **Digits** required and then click **OK**.
9. Click **OK**.
The equation is created and appears on the page, in the Custom Objects section of the Object Toolbox, and in the **Other equations** list in the Equation Builder next time you use it.

How can I create conditional formatting?

Conditional formatting is used to highlight particular results from the equation. This is set up using an 'if, then, else' type logic.

1. Select the **'If' condition** required.
This will usually be an equation set up on the previous tab, but could be an equation used elsewhere or a data field already used in the template.
2. Select the **Operator** required.
3. Enter a number or select an equation for the **Criteria** to be checked against.
So, for example, these first three settings could be setting up for If the result of the equation on the previous tab is less than 0.25 then...
4. Set the **'Then'** conditional formatting to happen if the criteria is met:
 - **No formatting** - nothing changes.
 - **Replace with text** – uses the text entered in the field below instead of the number generated by the equation or data field. You can also **Change Font**.
 - **Add text** – adds the text entered in the field below after the number calculated. You can also **Change Font**.
 - **Change font** – displays the calculated number but changes the font as set by **Change Font**.
 - **Remove row** – is used when the equation is in a table and takes the relevant row out of the table.
 - **Don't display section** – stops the section selected in the field below from being displayed.

5. Set the '**Else**' conditional formatting to happen if the criteria is not met.

The options are the same as above.

How can I reuse an Equation?

Once an equation has been created it can be reused elsewhere:

- To re-use an equation elsewhere on the template, simply select it from the Custom Objects section of the Objects toolbox and click on the page where you want it.
- To re-use an equation as part of another equation, when building the new equation, select it from the **Other equations** list.

If another equation is incorporated into an existing equation, the entire new equation appears in the existing equation and is highlighted until you click in the field.

For example:

{Tasks[Current].Samples[Current].Replicates[Current].Conc}*(Sqr({Tasks[Current].Samples[Current].Replicates[Current].Conc})), where SquareConc (the new equation) =(Sqr({Tasks[Current].Samples[Current].Replicates[Current].Conc}))

How do I edit an Equation?

- Double-click on the equation on the page,

OR

Select the equation on the page and then select **Properties**, either from the Format menu or the right-click menu.

The Equation Builder is displayed with the details of the equation loaded ready to be edited.

NOTE: Because once you create an equation it can be used elsewhere, when you edit an equation you must give the edited version a new name. If you had used this equation in more than one place and need to change it in all places simply go round deleting the old equation and replacing it with the new one from the Custom Objects section of the Objects Toolbox.

How do I delete an Equation?

1. In the tree view, click on the Equation to select it.

The border of the equation turns blue to show that it is selected.

2. Click **X**, or press **DELETE**, or right-click and from the menu select **Delete**.

The Equation is deleted from the template, but still exists in the Custom Objects section of the Objects Toolbox for re-use later.

You can delete the old equation from the Custom Objects by right clicking on it in the Custom Objects section of the Objects Toolbox and selecting **Delete**.

What Errors can occur?

Below are some error messages you may encounter along with explanations of the message:

XXX - Means that something could not be calculated.

- Means that the result was too big to fit.

NOTE: All calculations done with equations are performed using the full precision of the number. The formatting (decimal places, significant figures, scientific notation, etc) is applied after the calculation is complete.

Page Break

A Page Break can be added to a page to force a break after a particular object.

How do I add a Page Break to a Template?



1. From the Layout Tools group in the Objects Toolbox click

The mouse pointer changes to to show that the Page Break is selected.

2. Click the mouse on the page at the position you require the Page Break.
A Page Break is added to the Template and the name is added to the Tree View.

How do I move a Page Break?

1. Click on the Page Break or click on the name of the Page Break in the Tree view to select it.
The border of the Page Break turns blue to show that it is selected.
2. Position the mouse over the Page Break.
The mouse pointer changes to a four-headed arrow.
3. Click and drag to the required position.

How do I delete a Page Break?

1. In the Tree view, click on the Page Break to select it.
The border of the Page Break turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The Page Break is deleted.

How do I rename a Page Break?

By default the Page Breaks are named PAGEBREAKX , where X is numerically incremented (PAGEBREAK1 etc) for each new Page Break added to the template. You can rename the Page Break if you wish.

1. Right-click on the Page Break in the Tree View and select **Rename Item**.
The Rename dialog is displayed.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the Tree view is updated.

Edit Border

You can create a border around a previously placed text block, graphic, equation, data object, group or section. The styling of the line can then be set.

How do I add a border?

NOTE: You cannot add a border to a Table, table borders are set in the formatting of the table itself. See [How do I format a table?](#)

- A border can be edited using the Layout tool  or by selecting **Format>Edit Borders** from the main menu bar. You can also right click on an object and select **Properties** from the pop up menu that appears to edit a selected object's border.

Using the tool:

1. From the Layout Tools group in the Objects Toolbox click 
The mouse changes to  to show that Edit Border is selected.
2. Click on an object that you wish to add a border to.
The border properties for the selected object are displayed.
3. Select the required options and click **OK**.
The borders are added to the object.

Using the Edit Borders command:

1. Click on the object that you wish to add a border to.
2. From the **Format** menu, select **Edit Border**.
The border properties for the selected object are displayed.
3. Select the required options and click **OK**.
The borders are added to the object.

How do I format a border?

- When you create a border you can format it immediately
OR
You can re-format it by selecting **Edit Border** again.
- Use the **Padding** tools to set the distance between the border line and the object that you wish to place a border.
Using **Keep symmetrical** makes all the padding the same.
- Use the **Style**, **Weight** and **Color** controls to define the actual line used for the border.

How do I delete a border?

1. Select the item with the border and then select **Format > Edit Border** again.

OR

Right mouse click on the item with the border, select **Properties** from the pop up menu, and then select the **Border** tab.

2. Set the **Style** to **None**.

The border is removed.

Edit Caption

You can create a caption to a previously placed tool, object, group or section. This will create a single line of text (no word-wrap) so the number of characters that can be used is basically dependent on the font. The caption is placed outside of the object.

How do I add a caption?

- A caption can be added using the layout tool  or by using the **Edit Caption** command on the Format menu.
- **Edit Caption** is also available from the menu displayed when you right-click on an object.

NOTE: Adding a caption creates a single line of text. There is no word-wrap. The caption is placed outside of the object.

Using the mouse:

1. From the Layout Tools group in the Objects Toolbox click 
The mouse changes to  to show that Edit Caption is selected.
2. Click on an object that you wish to add a caption to.
The Edit caption dialog is displayed.
3. Select the required options as explained in the table below and click **OK**.
The caption is added to the object.

Using the Edit Caption command:

1. Click on the object that you wish to add a caption to.
2. From the Format menu, or the right mouse click menu, select **Edit caption**.
The Edit caption dialog is displayed.
3. Select the required options and click **OK**.
The caption is added.

How do I Format a caption?

- When you create a caption you can format it immediately

OR

You can re-format it by selecting **Edit Caption** again.

Caption	Enter the Caption that you wish to appear with the object.
Change Font	Displays a font selector enabling you to change the font, including the size and color.
Line Style	Select the style of line for the border around the caption from the drop-down list of available line styles.
Line Color	Select the color of the border around the caption from the drop-down list of available colors.
Line Weight	Select the thickness of the line around the caption from the drop-down list of available line weights.
Fill Color	Select the fill color for the caption from the drop-down list of available colors.
Position	Select the position of the caption. Select either Above , Below , Left or Right .
Alignment	Select the alignment of the caption. Select either Left , Right or Center .
Orientation	Select the orientation of the caption.
Keep symmetrical	Select Keep symmetrical to keep all the padding the same. The Bottom , Left and Right selectors are not available if Keep symmetrical is selected.
Top, Left, Bottom, Right	Padding selectors. Select the padding - that is, the distance between the border line and the object being bordered. Select a value between 0 and 20 .

How do I delete a caption?

1. Select the item with the caption and then select **Edit Caption** again.
2. Delete the text in the caption.
The caption is removed.

Section

Sections can be used both to create a set of objects that gets repeated based on some grouping you define (for example, for each sample) or to enable you to have a part of the template that you can switch on and off when you print a report.

NOTE: You can still add items to the section once it has been created.

How do I create a section?

1. From the Layout Tools group in the Objects Toolbox click .
The mouse changes to to show that Section is selected.
2. Click on the page where you want the top left hand corner then, holding down the mouse button, drag the mouse to the bottom right corner and release the mouse button.
The idea is to surround the objects that you wish to include in the section.
A dashed line appears around the section.
When the mouse is released the Format Section dialog is displayed.
The section is added to the tree view and the items in the section become sub-branches of the section.

How do I add items to a section?

- Simply select the layout tool or data object as normal and place it within the section.

NOTE: You can easily tell what is and isn't part of a section by looking at the tree view.

How do I move or resize a section?

The section can be moved and resized using the mouse or the **Format** command.

Using the mouse to resize the Text Block:

1. Click on the section to select it.
The dashed line changes to a solid blue line to show that it is selected.
2. Drag the corners or sides as required to resize.
The mouse changes to a two-headed arrow when placed over the edge of the section.

Using the mouse to move the section:

1. Click inside the section to select it.
The section line changes to blue to show that it is selected.
2. Position the mouse inside the section, but not on any of the items in it.
The mouse changes to a four-headed arrow.
3. Click and drag the section to the new position.

Using the Format command to move and resize the Text Block:

- From the Format menu select **Format**.

OR

Right-click on the section and select **Format**.

The Format Section dialog is displayed. In addition to resizing and moving the section, the dialog enables you to determine whether the section repeats and/or can be switched off.

NOTE: Moving or re-sizing the section may make it look like items placed on the page earlier are now part of the section. This is not true, and the best way to tell what is actually part of the section is to look at the tree view as items within a section appear as sub-branches of the section.

How do I format a section?

- A section can be formatted by selecting **Format** from the Format menu

OR

Right-click on the section and select **Format**.

The Format Section dialog is displayed.

On the dialog:

- Use the **Height** and **Width** controls to size the section.
- Use the **In from left** and **Down from top** controls to position the section on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

- Use **Repeat section based on** to set whether this is a repeating section, and if so what makes the section repeat.

NOTE: You can use nested sections for example to have a set of items repeat for each replicate, and then have an enveloping set of items that repeats for each sample.

- Click **Filter and Sort** to display the dialog to filter and sort the data associated with this section.
- Use **Allow section to be switched off** to set whether the section can be turned off before a report is printed.

When this option is selected, the name of the selected section is added to the print dialog.

How do I set the Expansion Count of the Object?

The **Expansion Count** button appears on the Object tab. The **Expansion Count** button is enabled when 'Repeat section based on' is checked and a selection appears in the drop-down list. Here you can define a particular item within a collection used to determine the number of iterations to be carried out on a sub collection that is specified to expand.

- Selecting **Properties** from the Format menu

OR

Right-click on the Section and select **Properties**.

The Section Properties dialog is displayed.

- Choose the **Expansion Count** button.
- Select the Index (Current, First, or Last) for each parent collection to define the number of expanded rows.

How do I delete a section?

1. In the Tree view, click on the section to select it.
The border of the section turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The section, and all the items in it, is deleted.

How do I delete items in a section?

1. In the Tree view, click on the item to select it.
The border of the item turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The item is deleted.

How do I move items out of a section?

To move an item out of a section to somewhere else on the page, simply use the cut and paste controls on the Edit menu or the right mouse click menu:

1. In the Tree view, click on the item to select it.
The border of the item turns blue to show that it is selected.
2. Select **Cut** from the Edit menu or the right mouse click menu.
The item is removed from the section, onto the clipboard.
3. Place the cursor where you want the item moved to, and select **Paste** from the Edit menu or the right mouse click menu.
The item is placed on the page and is no longer part of the section, as can be seen from the tree view.

How do I rename a section?

By default the sections are named SECTIONX, where X is numerically incremented (SECTION1 etc) for each new section added to the template. You can rename the section if you wish.

1. Right-click on the section in the Tree View and select **Rename Item**.
The Rename dialog is displayed.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the Tree view and within the section is updated.

Create Group

Grouping items can enable you to move them easily, draw a border around a set of items, or create a custom object made up of a set of items. Once a group is created it has a Properties dialog.

NOTE: You can also group items using the menu commands, as described in [Grouping items](#).

How do I create a group?

1. From the Layout Tools group in the Objects Toolbox click .
2. Click on the page where you want the top left hand corner then, holding down the mouse button, drag the mouse to the bottom right corner and release the mouse button.
The idea is to surround the objects that you wish to include in the group.
A yellow background shows what is in the group.
The group is added to the tree view and the items in the group become sub-branches of the group.

How do I move a group?

The group cannot be resized, but it can be moved using the mouse:

1. Click on the group to select it.
The line around the group changes to blue to show that it is selected.
2. Position the mouse inside the group, but not on any of the items in it.
The mouse changes to a four-headed arrow.
3. Click and drag the group to the new position.

How do I delete a group?

1. In the Tree view, click on the group to select it.
The border of the group turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
All the items in the group are deleted.

How do I change a group or items in a group?

Simply put, you don't, you ungroup the items first.

- To make any changes, select the group and then select **Ungroup** from the Format menu or the right mouse click menu.

How do I change group properties.

The dialog may be accessed when a group object is selected by choosing the **Properties** command from the main Format menu or from the context menu. The Group Properties dialog will contain only the Border, Caption and Note tabs, since the group has no properties of its own.

How do I rename a group?

By default the groups are named GROUPX , where X is numerically incremented (GROUP1 etc) for each new group added to the template. You can rename the group if you wish.

1. Right-click on the group in the Tree View and select **Rename Item**.
The Rename dialog is displayed.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the Tree view and within the group is updated.

Insert H Line

Horizontal lines can be added to divide up different areas of the report.

How do I add a horizontal line to a Template?



1. From the Layout Tools group in the Objects Toolbox click .
The mouse pointer changes to to show that the horizontal line is selected.
2. Drag the mouse onto the page and draw a line in the position you require and of the approximate size you require.
The horizontal line is added to the page and the name is added to the Tree View.

How do I move or resize a horizontal line?

The horizontal line can be moved and resized using the mouse or the **Properties** command.

Using the mouse to resize the line:

NOTE: This only applies if the line has not been set to **Fit across page**.

1. Click on the line.
The border of the line turns blue to show that it is selected.
2. Drag the ends as required to resize the line.
The mouse changes to a two-headed arrow when placed at the ends of the line.

Using the mouse to move the line:

1. Click on the line.
The border of the line turns blue to show that it is selected.
2. Place the mouse over the line.
The mouse changes to a four-headed arrow.
3. Click and drag to the required position.

Using the Properties command to move and resize the line:

- From the Format menu select **Properties**.
The H Line Properties dialog is displayed. In addition to resizing and moving the line, the dialog enables you to determine the styling of the line.

How do I format a horizontal line?

- From the Format menu select **Format**.

The Properties HLine dialog is displayed. In addition to resizing and moving the line, the dialog enables you to determine the styling of the line.

On the dialog:

- **Fit across page** sets the line to the width of the page.
Including or excluding the margin as selected by **Inside margins**.
- **Set width** enables you to select the length of the line.
- Use the **In from left** and **Down from top** controls to position the line on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

- Use **Style**, **Weight** and **Color** to set the style of the line.

How do I delete a horizontal line?

1. In the Tree view, click on the line to select it.
The border of the line turns blue to show that it is selected.

2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The line is deleted.

How do I rename a horizontal line?

By default the Horizontal Lines are named HORIZONTALLINEX , where X is numerically incremented (HORIZONTALLINE1 etc) for each new horizontal line added to the template. You can rename the horizontal line if you wish.

1. Right-click on the line in the Tree View and select **Rename Item**.
The Rename dialog is displayed.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the tree view is updated.

Insert V Line

Vertical lines can be added to divide up different areas of the report.

How do I add a vertical line to a Template?

1. From the Layout Tools group in the Objects Toolbox click 

The mouse pointer changes to  to show that the vertical line is selected.

2. Drag the mouse onto the page and draw a line in the position you require and of the approximate size you require.

The vertical line is added to the page and the name is added to the Tree View.

How do I move or resize a vertical line?

The horizontal line can be moved and resized using the mouse or the **Properties** command.

Using the mouse to resize the line:

NOTE: This only applies if the line has not been set to **Fit Length of Page**.

1. Click on the line.
The border of the line turns blue to show that it is selected.
2. Drag the ends as required to resize the line.
The mouse changes to a two-headed arrow when placed at the ends of the line.

Using the mouse to move the line:

1. Click on the line.
The border of the line turns blue to show that it is selected.
2. Place the mouse over the line.
The mouse changes to a four-headed arrow.
3. Click and drag to the required position.

Using the Properties command to move and resize the line:

- From the Format menu select **Properties**.

The V Line Properties dialog is displayed. In addition to resizing and moving the line, the dialog enables you to determine the styling of the line.

How do I format a vertical line?

- From the Format menu select **Properties**.
The V Line Properties dialog is displayed. In addition to resizing and moving the line, the dialog enables you to determine the styling of the line.

On the dialog:

- **Fit length of page** sets the line to the height of the page.
Including or excluding the margin as selected by **Inside margins**.
- **Set height** enables you to select the length of the line.
- Use the **In from left** and **Down from top** controls to position the line on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

- Use **Style**, **Weight** and **Color** to set the style of the line.

How do I delete a vertical line?

1. In the Tree view, click on the line to select it.
The border of the line turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The line is deleted.

How do I rename a horizontal line?

By default the Vertical Lines are named VERTICALLINEX , where X is numerically incremented (VERTICALLINE1 etc) for each new vertical line added to the template. You can rename the vertical line if you wish.

1. Right-click on the line in the Tree View and select **Rename Item**.
2. Enter a new name in the **New Name** field and click **OK**.
The name in the tree view is updated.

Using Data Objects

Data Objects are the core of the template designer. Each parent application supplies Communiqué with a full list of the data objects it can create and places them in the Data Objects group within the Objects Toolbox. You then place these objects on the template as 'placeholders' and when a report is generated using the template, those data objects are 'populated'. After placing an object on the template, there are properties you can set to determine how the data will look when the object is populated.

What types of Data Objects are there?

There are four distinctly different types of Data Objects and each has different properties:

- Text objects – both individual text items and full blocks of text.
- Numeric objects – both numbers set by a user in the parent application and numbers generated when samples are run.
- Date/Time objects – both generated by the PC system and the application.
- Special objects – there are special objects generated by the parent application that are shown as  in the Data Objects list.

NOTE: Not all data object types may be present in each data source.

How do I place a Data Object on a Template?

1. Select the required Data Object from the list of available Data Objects.



The mouse pointer changes to  to show that a data object has been selected.

2. Click on the page where the data object is to be placed.

This can be just placed anywhere on the page as an individual item, or placed within a text box or table layout tool.

NOTE: Data objects marked  create a frame that other objects of the same type can also be added to. Creating one container with multiple objects, for example a spectral graph with two spectra on it.

How do I set properties of a Text Object?

- The properties of a data object can be set by selecting the object and then selecting **Properties** from the Format menu.

OR

Right-click on the object and select **Properties**.

The properties dialog is displayed.

On the Object tab:

- Use the **Horizontal** and **Vertical** controls to set the positioning of the object within its frame.
- Use the **Between lines** and **Between paragraphs** controls to set the paragraph spacings for large blocks of text.

- See how your settings will look in the **Preview** area .

On the Location tab:

- Use the **Height (inches)** and **Width (inches)** to set the size of the data object
- Use **In from left (inches)** and **Down from top (inches)** to set the position relative to the top edge of the page.
- Use the **Truncate and Resize block to fit** controls to deal with oversized blocks of text.

On the Font tab:

- Use the **Font**, **Size**, **Style**, **Color**, and **Underline** controls to determine how the text will look.
- See how the text will look in the **Preview** area.

On the Border, Caption, Note tabs:

- Make additional customized settings for this data object.

How do I set properties of a Numeric Object?

- The properties of a data object can be set by selecting the object and then selecting **Properties** from the Format menu.

OR

Right-click on the object and select **Properties**.

The properties dialog is displayed.

On the Object tab:

- Use the **Horizontal** and **Vertical** controls to set the positioning of the object within its frame.
- Use the **Format** controls to set the way the number itself will be formatted.
This enables you to set exactly how this particular number object will be displayed.

NOTE: If you want to change the default setting for numbers to be displayed, use the **Default number format** command on the Properties dialog. This will change the formatting for all new numeric data objects added to the template.

On the Location tab:

- Use the **Height (inches)** and **Width (inches)** to set the size of the data object
- Use **In from left (inches)** and **Down from top (inches)** to set the position relative to the top edge of the page.

On the Font tab:

- Use the **Font**, **Size**, **Style**, **Color**, and **Underline** controls to determine how the text will look.
- Use the **Preview** to see how the number will look.

On the Border, Caption, Note tabs:

- Make additional customized settings for this data object.

How do I set properties of a Date/Time Object?

- The properties of a data object can be set by selecting the object and then selecting **Properties** from the Format menu

OR

Right-click on the object and select **Properties**.

The properties dialog is displayed.

On the Object tab:

- Use the **Horizontal** and **Vertical** controls to set the positioning of the object within its frame.
- Use the **DateFormat** controls to determine whether the date is displayed for this particular object and set the date format.
- Use the **Time format** controls to determine whether the time is displayed for this particular object and set the time format, including:
 - **Local time** – The time zone you are in now.
 - **Original time** – The time zone the data was collected in.
 - **Selected zone** – A time preset time zone, often GMT.

NOTE: If you want to change the default setting for all times and dates to be displayed in, use the **Default date/time format** command on the Properties dialog. This will change the formatting for all new objects added to the template.

- Use the **Layout controls** to determine how the object should be laid out when both the time and date are included.
- Use the **Preview** section to see how the settings you have selected here will look.

On the Location tab:

- Use the **Height (inches)** and **Width (inches)** to set the size of the data object
- Use **In from left (inches)** and **Down from top (inches)** to set the position relative to the top edge of the page.

On the Font tab:

- Use the **Font**, **Size**, **Style**, **Color**, and **Underline** controls to determine how the text will look.
- Use the **Preview** to see how the text will look.

On the Border, Caption, Note tabs:

- Make additional customized settings for this data object.

How do I set properties of Currency Objects?

- The properties of a data object can be set by selecting the object and then selecting **Properties** from the Format menu.

OR

Right-click on the object and select **Properties**.

The properties dialog is displayed.

NOTE: If the parent application uses currency settings they will over-ride these settings.

On the Object tab:

- Use the **Horizontal** and **Vertical** controls to set the positioning of the object within its frame.
- Use the **Currency symbol**, **Decimal separator**, **Digit grouping character**, and **Negative indicator** to control how your currency is displayed.
- Use **Number of digits after the decimal** to set the number of digits to appear after the decimal symbol.
- See how your settings will look in the **Preview** area.

On the Format tab:

- Use **Spacing/Zeros** selections to set how your currency is displayed.
- Include a space between the currency symbol and value.
- Include a 0 before the decimal separator if the value is between -1 and +1.
- Add zeros as required to make up decimal digits to the number defined by **Number of digits after decimal** on the Object tab.
- Use **Ordering selections** to set positions.
- See how your settings will look in the **Preview** area.

On the Location tab:

- Use the **Height (inches)** and **Width (inches)** to set the size of the data object
- Use **In from left (inches)** and **Down from top (inches)** to set the position relative to the top edge of the page.
- Use the **Truncate and Resize block to fit** controls to deal with oversized blocks of text.

On the Font tab:

- Use the **Font**, **Size**, **Style**, **Color**, and **Underline** controls to determine how the text will look.
- See how the text will look in the **Preview** area.

On the Border, Caption, Note tabs:

- Make additional customized settings for this data object.

How do I edit properties for multiple Data Objects?

About selecting multiple Data Objects:

- Select multiple objects by holding the Shift key and clicking on each object you want to select
- Multiple objects may be selected within a single section but objects may not be selected within several sections at once. Nor may objects within a section be selected at the same time as objects outside the section.
- Multiple data objects may be selected within a table but not within several tables at once. Nor may objects within a table be selected at the same time as objects outside the table. Nor may data objects and table cells be selected at the same time.
- A group may be selected together with other objects but objects within a group cannot be selected.

The general principle for editing properties of multiple objects is that the tabs common to all the selected objects will be included in the Properties dialog, but the following tabs will never be displayed in the case of multiple objects:

- Caption – Since displaying the ‘common’ property values would not indicate which of the objects currently had captions and which did not it would be too easy to overwrite an existing caption accidentally if this tab were shown.
- Note – The same argument as for Caption applies here also.
- Location – The size and position of several objects can be set more efficiently with the ‘Align’ commands.

Selecting Different Type Objects

Because all objects have some unique properties (on the ‘Object’ or other unique tab) the maximum set of common properties that can be available when objects of different types are selected will be **Font** and **Border**. Note that a Text Block and a Text Data Object are considered different types.

NOTE: V Line, H Line and Table objects share none of the relevant tabs with other objects, so whenever one of these is included in a multiple selection the **Properties** command is disabled.

Graphic, Equation, Section, Group and ActiveX Object Frame objects only share the **Border** property with other objects, so whenever these objects are included with a multiple selection only the **Border** tab is displayed in the **Properties** dialog. When the objects are within a table then the **Properties** command is disabled.

Text Block, Text data, Numeric data, Date/Time and Currency objects share both **Font** and **Border** tabs. Therefore when the multiple selection consists only of objects from this group then both these tabs are displayed. When the selections are within a table then only the **Font** tab is displayed.

Selecting Objects of the Same Type

When a multiple selection consists of objects of a single type then (for most objects) the tab(s) specific to that type will be included in the **Properties** dialog, together with the **Font** and **Border** tabs (where appropriate). Again, Caption, Note and Location tabs will never be displayed for a multiple selection.

Exceptions to this rule are the Table, Equation and Section objects. Because of their complex nature it is inappropriate to attempt to set common properties and so these are treated as special cases. When multiple tables are selected the properties command is disabled. When multiple equations or sections are selected the **Properties** dialog contains only the **Border** tab.

How do I set properties of an Object marked ?

The properties of a  object depend on the object itself, and will differ from object to object, but can be selected by:

- Selecting the object and then selecting **Properties** from the Format menu.
- OR
- Right-click on the object and select **Properties**.
- The properties dialog is displayed.

NOTE: Not all objects will have an **Properties** dialog box.

How do I format an Object marked ?

 objects have a 'frame' that can be formatted to set the size and position of the object on the page:

- The properties of an  object can be set by selecting the object and then selecting **Properties** from the Format menu.
- OR
- Right-click on the object frame and select **Properties**.
- The Properties dialog is displayed.

On the Frame tab:

- Use the **Height** and **Width** controls to set the size of the frame.
- Use the **In from left** and **Down from top** controls to set the position of the frame on the page.

NOTE: If **snap to grid** is on, the top left corner will still snap to the grid.

- Use the **Fill** control to set the background color of the frame.

On the Object tab

- Use the **Horizontal** and **Vertical** controls to set the position of the object within the frame.
- Use the **Fit** controls to determine what happens if the object is a different size to the frame:

- Selecting **Fit to frame** will resize the incoming object to the size of the frame. Multiple objects will be tiled.

NOTE: The **Fit to frame** will occur when the report is rendered (using Preview or Print).

When the Object Frame and its associated ActiveX control are displayed in the design environment there is always a small gap between the ActiveX and its Object Frame container. This gap is there to allow selection of the Object Frame and also so that additional copies of the ActiveX control can be dropped in the same frame (for example, when creating overlaid plots). But this is purely for display purposes.

When the report is displayed or printed and the **Fit to frame** option is set then the size of the ActiveX will be the size set in the Object Frame Properties and no less.

- Selecting **Keep original size** will keep the size of the incoming object, resizing the frame to accommodate the object, and using the top left corner as the anchor point.
- Selecting **Scale on height** will rescale the incoming object so the height is the same as the frame, but it will increase the width if necessary to ensure the aspect ratio is maintained.
- Selecting **Scale on width** will rescale the incoming object so the width is the same as the frame, but it will increase the height if necessary to ensure the aspect ratio is maintained. **In from left** and **Down from top** settings are maintained.

On the Sequence tab:

NOTE: The Sequence tab appears in some parent applications.

- Use the **Repeat based on** control to set up the control to repeat for each instance of the set parameter.

For example, if the control was a graph and samples was selected for the repeat then you would get each sample on its own graph, whereas if samples was not set here, all the samples would appear on one graph.

How do I repeat an Object frame marked ?



When the Object Frame and its associated ActiveX control  are displayed in the design environment, there is always a small gap between the ActiveX and its Object Frame container. This gap is there to allow selection of the Object Frame and also so that additional copies of the ActiveX control  can be dropped in the same frame (for example, when creating overlaid plots).

1. After dropping the ActiveX controls into the small gap area, move the mouse pointer over the small gap until it turns into .
2. Right click and select Indexing from the menu.

How do I set Default Font properties?

While the properties of the fonts of individual data objects can be set (by selecting the object and then selecting **Properties** from the Format menu, followed by the Font tab) you can also set the default font for all items in the template:

1. Select **Default font** on the Format menu.
The Default font dialog is displayed.
2. Use the **Font**, **Size**, **Style**, **Color**, and **Underline** controls to determine how the text will look.
Use the **Preview** to see how the text will look.
3. Click **OK**.
From now on, all data objects and text added to the template will use this new font.

How do I use Indexing?

Where an object can potentially be one of a collection of items, you can set the specific instance using indexing.

- Having selected the object, select **Indexing** from the Format menu.

On the Indexing dialog you can set the particular index for each possible choice as listed in the first column.

If there is more than one item on the object then you will get more than one Index tab.

The default list for indexing is **Current**, **First**, **Last**, **NextPrevious**, and **All** but you can enter your own terms, for example if you have a software application that identifies peaks by a name, then you can use that name for indexing.

When a new term, or keyword, is entered it then becomes available from the drop down list the next time you start the Indexing dialog and these keywords can be organized by selecting **Organize keywords** on the Tools menu.

NOTE: Further information is given in [Indexing](#).

How do I delete a Data Object?

Deleting an object placed on a page:

1. In the Tree view, click on the object to select it.
The border of the object turns blue to show that it is selected.
2. Click , or press DELETE, or right-click and from the menu select **Delete**.
The object is deleted.

Deleting an object embedded in text:

1. Highlight the Data Object within the Text Block using the mouse.
2. Press DELETE.
The object is deleted.

NOTE: The  tool and Delete on the context menu will delete the whole text block.

How do I rename a Data Object?

The Data Object is named (Object Name) DataX where X is numerically incremented (Object Name)Data1, etc. for each new Data Object added to the template. You can rename the Data Object if you wish.

1. Click on the Data Object to select it.
The border of the block turns blue to show that it is selected.
2. Right-click inside the Data Object and select **Rename**.
The Rename dialog is displayed.
3. Enter a new name in the **New Name** field and click **OK**.
The name in the Tree view and within the Data Object is updated.

Using Custom Objects

In the Custom Objects section of the Objects Toolbox there are two groups Custom Data Objects and [Equations](#). Both are created by users of Communiqué while generating templates and can then easily be added to other templates.

How do I create a Custom Data Object?

1. Select an item or multiple items and then select **Create custom object** from the Actions menu or click .

OR

Create a [Group](#) of items and then select **Create custom object** from the Actions menu or click .

OR

Create a [Section](#) and then select **Create custom object** from the Actions menu or click .

The Name custom object dialog is displayed.

2. Enter a name for the object and click **OK**.

The object is added to the Custom Objects section of the Objects Toolbox.

How do I place a Custom Object on a template?

1. From the Custom Objects section of the Objects Toolbox, select the required object or equation.

The mouse pointer changes to  to show that the Custom Object is selected, or to  to show an equation has been selected.

2. Click on the page where you want the object.

The object is added to the page and the structure is added to the Tree View as it was when the custom object was created.

How do I reorganize the list of Custom Data Objects?

1. From the Tools menu select **Organize custom objects**.

The Organize Custom Objects dialog is displayed.

2. From the list, highlight the object that you wish to move, rename or delete.

3. Select Move Up, Move Down, Rename or Delete.

Move Up	Moves the selected item up the list.
Move Down	Moves the selected item down the list.
Rename	Enables you to rename the selected custom object.
Delete	Deletes the selected item from the list. You are asked to confirm the deletion.

How do I rename a Custom Data Object?

1. From the Tools menu select **Organize custom objects**.
The Organize Custom Objects dialog is displayed.
2. Select the custom object from the list and click **Rename**.
The Rename Custom Object dialog is displayed.
3. Enter a **New Name** for the Custom Object and click **OK**.
4. The Organize Custom Objects dialog and the Custom Objects list in the Objects Toolbox is updated with the new name.

How do I delete a Custom Data Object?

1. From the Tools menu select **Organize custom objects**.
The Organize Custom Objects dialog is displayed.
2. Select the custom object from the list and click **Delete**.
You are asked to confirm the deletion.

How do Import/Export a Custom Data Object?

Exporting a Custom Object:

You can export a custom object to disk so that it can be used on another PC.

1. Select the item to be selected in the Custom Objects section of the Objects Toolbox.
2. Select **Export** and then **CustomObject**, from the File menu.
3. On the dialog, choose where to save the custom object and give it a name.

Importing a Custom Object:

You can import a custom object created with another copy of Communiqué.

1. Select **Import** and then **CustomObject**, from the File menu.
2. Use the file selector to locate the required Template.

Indexing

Where an object can potentially be one of a collection of items, you can set the specific instance using indexing.

Indexing is used for all data objects and equation terms that could potentially be one of a collection. Each indexing term is called a keyword and the default list of keywords for indexing is **Current, First, Last, Next, Previous**, and **All** but you can enter new words as long as they will be understood by the Parent Application.

How do I use indexing for a data object?

- Having selected the object, select **Indexing** from the Format menu or the right mouse click menu.

On the Indexing dialog you can set the particular index for each possible **Collection** as listed in the first column.

If there is more than one item on the object then you will get more than one Index tab.

The default list for indexing is **Current, First, Last, Next, Previous**, and **All**, but you can enter your own terms, for example if you have a software application that identifies peaks by a name, then you can use that name for indexing.

When a new term, or keyword, is entered it then becomes available from the drop down list the next time you start the Indexing dialog and these keywords can be organized by selecting **Organize keywords** on the Tools menu.

NOTE: In most cases the index of **Current** will suffice, other indices are used to construct complex equations or instances where you need to take for example the result of one sample from the sample before it (**previous**), or need to work with a named peak.

NOTE: When using  objects, you can place multiple objects of the same type into one frame. When you then select indexing for this frame the Indexing dialog will have multiple tabs, one for each object.

How do I use indexing in equations?

1. When a data object is inserted in the equation, the display will show how that object is set up, including indexes within square brackets, [].

By default, the indexing will be set to **Current** for all, for example an equation to create the square of a concentration value would look like:

`Sqr({Tasks[Current].Samples[Current].Replicates[Current].Conc})`

2. Clicking on one of the indexes will display the current list of possible indexes, enabling you to select the one required.

The default list for indexing is **Current, First, Last, Next** and **Previous**, but you can enter your own terms using the **Organize keywords** command on the Tools menu. For example if you have a software application that identifies peaks by a name, then you can use that name for indexing.

NOTE: In most cases the index of **Current** will suffice, other indices are used to construct complex equations or instances where you need to take for example the result of one sample from the sample before it (**previous**), or need to work with a named peak.

How do I use Indexing to get the iteration count for parent collections of a sub-collection?

The **Expansion Count** button appears on both the Rows and Columns tabs. The **Expansion Count** button is enabled when 'Enables column to expand' is checked and a selection appears in the 'Expand by' drop-down list. Here you can set the index of any parent collection that contains the sub-collection you are about to iterate.

- Selecting **Properties** from the Format menu

OR

Right-click on the Table and select **Properties**.

The Table Properties dialog is displayed. You can format the whole table, rows, columns, or individual cells.

- Choose the **Expansion Count** button.
- Select the Index (Current, First, or Last) for each parent collection to define the number of expanded rows.

How do I organize my keywords?

The **Organize Keywords** command enables you to edit the list of indexing keywords that appears both in the Equations dialog and the Indexing dialog.

1. Select **Organize keywords** on the Tools menu.
The Organize keywords dialog is displayed.
2. **Add** and **Delete** keywords as required.
You cannot delete the standard keywords of **Current**, **First**, **Last**, **Next**, **Previous**, and **All**.
3. **Edit** keywords as required.
4. Use the **Move up** and **Move down** controls to determine the order you want the drop down list of keywords to appear.
5. Click **OK** when you are finished.

How do I use the All index?

The **All** index only works with certain functions. These functions are: Min, Max, Sum Of, Count of, Mean, SD, RMS, and %RSD.

The **All** index can be useful for calculations that involve more than one piece of data (i.e. an Average of data points). For example, if you wish to know the average concentration for a set of samples the following equation would calculate that for them:

Mean{Tasks[Current].Samples[ALL].Conc}]

Laying out items

Once you have placed items on the page there are some things you can do to tidy up the layout of the page:

- Group items so you can move them as one object.
- Order items so that one item is overlaid on another in the correct order.
- Align items with other items or to the grid.

How do I group items?

Grouping can be achieved in a number of ways:



- Use the Create Group Layout Tool, , and draw around the objects you want to group.

OR

Select the required items by holding down SHIFT and clicking on them, and then select **Group objects** on the Actions menu or click .

OR

Select the required items by holding down SHIFT and clicking on them, and then right-click on one of the items and select **Group** from the menu displayed.

When items are grouped a border is drawn around them and a yellow background color is displayed. A group level is also inserted on the Tree View and the items in the group are moved to sub-branches.

NOTE: Once items are placed in a group, there is nothing you can do with them until you ungroup them again.

How do I ungroup items?

To ungroup a group of items:

- Right-click on the yellow background and select **Ungroup** from the menu displayed.

OR

Select the group on the Tree View and then select **Ungroup objects** from the Actions menu or click .

How do I set the order of items?

A number of items placed on the page can overlap, so there are tools to enable you to set the layers of the overlap.

- Select the item you want to change the order of and the select the required option from the Order objects sub menu off the Actions menu.

OR

Right-click on the item required and select an option from the Order objects sub menu of the menu displayed.

OR

Select the item you want to change the order of and click the correct toolbar icon.

Menu Command	Icon	Action
Bring to front		Brings current object to the front of all others.
Send to back		Sends current object to the back of all the others.
Bring forward		Brings current object forward one level.
Send backward		Sends current object back one level.

How do I align items?

Items on a page can be aligned in two ways:

- Select the required items by holding down SHIFT and clicking on the items, then select the required alignment from the Align sub menu on the Format menu.
- Switch on the **Snap to grid** control on the Template level of the Tree View.
All objects placed on the page will move to the nearest grid lines.

How do I move/size items?

Position the mouse cursor on the edge of an item so it turns into the required arrow headed cursor and then click and drag the mouse.

A four-headed arrow cursor will move the item, a horizontal or vertical two-headed cursor will resize the selected edge, and a diagonal two-headed cursor will resize from the selected corner.

- Select the required item and use the arrow keys on the keyboard to move the item.
- Select the required item and hold down SHIFT and use the arrow keys to expand an edge of the item.

Using the Rulers and Grid

The rulers and the grid are used to help you position items on the page.

The grid is a series of lines at equal spacings, displayed on the page.
The rulers are displayed across the top of the page and down the left side.

How do I set the Grid up?

1. Select the top (template) level of the tree view.
The template page is displayed.
2. Set the size and units, for the grid spacing.
3. Select whether to **Display grid**.
4. Select whether to use **Snap to grid**.
Snap to Grid can still be used if the grid is not displayed, as the grid is still present.

How do I turn the grid on and off?

- To turn the grid on, select **Grid** from the View menu.
OR
Click  on the toolbar.
OR
Select **Display grid** on the template page.
- Selecting the same control will turn the grid off.

How do I turn the rulers on and off?

- From the View menu select **Rulers** or click .
Selecting the same thing again will turn the rulers back off.

How do I change the gradations on the ruler from inches to metric and vice-versa?

1. Select **Measurement Units** from the Format menu.
The Measurement Units dialog is displayed.
2. Select **mm, cm** or **inches** from the drop-down list and click **OK**.
The dialog closes and the rulers are updated.

NOTE: This also changes the measurement units used on dialogs.

Using Communiqué Utilities

The Communiqué Utilities are available as a separate environment run from the parent application main menu (in a similar manner to the template design environment). It enables you to archive, restore and delete both templates and stored reports. It also provides you with the ability to import and export templates, as well as preview and print stored reports.

The Status field (located on the bottom of the Communiqué window) provides feedback. This is particularly useful during multiple operations such as importing or exporting multiple templates.

What types of utilities are there?

You can:

Import... – Imports one (or more) report template(s) from a file (or files) into the Communiqué database.

Export... – Exports one or more report templates from the Communiqué database to a file (or files).

Archive... – Moves one or more selected templates or stored reports to an archive file. Any audit trail and event log records associated with the selected datasets are always included in the archive.

Open Archive... – Enables you to select an archived data file and view the contents.

Restore... – Restores selected items in the archive file to the main Communiqué database.

Print... – Prints the selected report(s) to the selected printer.

Print Preview – Displays the selected reports in the Communiqué preview window.

Folder New Folder... – Creates a new folder as a child of the selected folder and allows the user to enter a name for the folder.

What information is available on this screen?

There are three tabs under this dialog:

- **Reports** tab - Contains a list of stored Reports and associated information.
- **Templates** tab - Contains a list of Templates.
- **Event Log** tab - The Event Log list view shows ONLY those events that are not associated with either a report or a template ('general' events). The Communiqué Utilities is not intended as an environment to examine event logs because this can be done quite adequately in the Communiqué Designer. The Utilities window is primarily for management of the database. With reference to the event log this means archiving and/or deleting sections of it.

How do I locate a template in a large list?

1. Select the **Templates** tab.
2. Choose the **Define Template Filter** command from the Options menu.
3. Set up the desired filter conditions in the **Filter Templates** dialog.
4. **Apply** the defined filter settings.

How do I export a template from the Communiqué database to a separate file?

1. Select the desired report template in the list displayed on the **Templates** tab.
2. Choose the **Export** command from the File menu.
3. Select the desired path and enter a name for the exported file, in the **Save As** dialog.

How do I import a template into the Communiqué database from a previously exported file?

1. Select the **Templates** tab.
2. Choose the **Import** command from the File menu.
3. Select the desired template file within the **Open** dialog.

How do I print a report stored in the Communiqué database?

1. Select the desired report in the list displayed on the **Reports** tab.
2. Choose the **Print** command from the File menu.
3. Select the printer to be used for output in the **Print** dialog.

How do I archive stored reports or templates?

1. Select the desired report(s) or template(s) in the list on the appropriate tab.
2. Choose the **Archive** command from the File menu.
3. Specify the archive file name and location.
4. Start the archive process.

How do I archive a portion of the general event log?

1. Select the **Event Log** tab.
2. Select the range of events to be archived.
3. Choose the **Archive** command from the File menu.
4. Specify the archive file name and location.
5. Start the archive process.

How do I restore stored reports or templates from an archive?

1. Choose the **Open Archive** command from the File menu.
2. Select the required archive file.
3. On the **Reports Archive** or **Templates Archive** tab select the desired reports or templates.
4. Choose the **Restore** command from the File menu.
5. Select whether to restore the data to the original location (folder path) in the database or to select a new root folder.
6. Start the restore process.

How do I delete a stored report or a template from the Communiqué database?

1. Select the desired report(s) or template(s) in the list on the appropriate tab.
2. Choose the **Delete** command from the Edit menu.

How do I create a new folder?

1. Select the folder in the tree which is to be the parent of the new folder (to create a folder at the highest level, select the 'Folders' node).
2. Choose the **New Folder** command from the File/Folder submenu or the Tree View context menu.
3. Enter a name for the new folder.

Reports Filter

This defines the filter for displaying stored reports. The filter applies to both the normal Reports tab view and also the Reports Archive tab view. The items listed in the dialog are all properties of stored reports in the Communiqué database. The initial settings after installation of the software is no items selected. Thereafter the last used set of conditions will be remembered.

How do I define filter settings?

To define filter settings:

1. Select reports matching any or all of the filter conditions.
2. Check the items to be used to define matching reports.
3. Select the appropriate operator (the available operators depend on the item type)
4. Enter or select the value(s) associated with the filter item (if any)
5. Click **OK** to save the filter definition and close the dialog (it will be applied if **Apply Report Filter** is currently checked but not otherwise)

or

Click **Apply** to have the list updated immediately (**Apply Report Filter** is always set **On** if the **Apply** button is clicked).

Templates Filter

The filter applies to both the normal Templates tab view and also to the Templates Archive tab view. The items listed in the dialog are all properties of stored templates in the Communiqué database. One significant difference between this and the Reports Filter dialog is that the initial default settings for the templates filter will not be to have all items unchecked. The initial default is for only the latest versions of templates to be displayed.

How do I define filter settings?

To define filter settings:

1. Select to display templates matching any or all of the filter conditions.
2. Check the items to be used to define matching templates.
3. Select the appropriate operator (the available operators depend on the item type).
4. Enter or select the value(s) associated with the filter item (if any)
5. Click **OK** to save the filter definition and close the dialog (it will be applied if **Apply Template Filter** is currently checked but not otherwise)

OR

Click **Apply** to have the list updated immediately (**Apply Template Filter** is always set **On** if the **Apply** button is clicked).

Archiving Data

The archive process is fundamentally the same for all data types, although there may be minor differences in how data are selected for archiving and in what related data are archived.

How do I select data for archiving?

The Reports and Templates views allow multiple selection of items using either of the standard Windows techniques (CTRL+Click and Shift+Click). The Event Log view only allows multiple selection using the Shift+Click mechanism. The reason for this is so that only an uninterrupted time period of events can be archived. This will allow the event log entry associated with the archive (or deletion) process to be specific and complete (and 21 CFR part 11 compliant).

How do I set archiving options?

When the Archive command has been issued, the Archive (Save As) dialog is displayed. In this dialog you specify the destination archive file. Archived items are deleted from the Communiqué database after the archive process is complete. In this case 'deletion' actually means 'purge' in that the items will be removed completely from the database and not just marked for deletion.

You may select an existing archive file in the Archive dialog. The selected items will then be added to the existing archive.

NOTE: An archive may then consist of different data types (templates, reports and event logs).

To initiate the archive process:

- Click **Save** in the Archive dialog.

If the Archive process has been set as a signature point then the electronic signature dialog will then be displayed. Only one signature will be required even if there are multiple items selected for archiving.

Previewing a Template

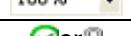
The Print Preview window has two purposes. It enables the designer of a template to see what it looks like with data and it enables the template to be reviewed, approved or locked depending on the security settings of the Parent Application.

How do I preview my template?

- Click  or select **Print Preview** from the File menu.
The Print Preview window is displayed.

What do the items on the toolbar do?

The toolbar icons perform the following actions:

	Prints the template, as discussed in Printing a template .
	Displays a full page.
	Displays a selector for showing multiple pages at once.
 100%	Zooms in or out.
	Enables you to approve (see Reviewing and Approving) or lock (see How do I lock/unlock a template?) template, depending on the security settings in your Parent Application.
	Closes the window.
	Displays this help page.

How do I move between pages within the Print Preview?

The toolbar contains controls, , to move through pages of the template. In order, the controls are: go to the first page, go back one page, current page, go to the next page, go to the last page and **GoTo** a selected page.

Reviewing and Approving

NOTE: You must first save the template before reviewing and approving.

NOTE: To minimize font size distortion when approving documents, always display the documents at 100%.

You can review or approve a template from the Print Preview dialog:

- Display the template or report required and click .

The dialog displayed enables you to see who else has signed off this template/report and enables you to select the level you want to approve the report at.

What do the columns in the table show?

Name – The user that approved the template/report.

Timestamp – The date and time of the approval in full format.

Type – The type of approval as selected in the Add approval control.

Reason – Only available if using electronic signatures. Shows the reason for the signature as selected on the e-signature dialog.

E-sig – displays a check mark if the approval point included an e-signature.

Comments – any comments entered.

What levels of approval are there?

- Select the type of approval you are creating.

Review signs to say you are happy with it, **Approve** locks it, **Hold** (reports only) rejects it.

Printing a Template

- In the Communiqué main window, click  or select **Print** from the File menu.

OR

In the Print Preview window, click .

The Print dialog is displayed.

From the Print dialog you can:

- Decide what to print:
- **Template only** - Prints the template as it is.
- **Template with details** - Prints the template and a full list of the properties of all the components on it.
- **Template with signatures** - Prints the template together with a page showing the approval signatures for the template. This option is not available if electronic signatures are not used by the parent application.
- **Template with details and signatures** - Prints the template, a full list of properties of all the components on it, and the approval signatures for the template.
- Decide how to print:
- **Print hardcopy** - Prints to the printer selected from the drop-down list of available printers. Click **Properties** to display the printer properties.

NOTE: You can print as PDF files. This feature is possible only if you purchased and installed Adobe Acrobat Writer on the target PC.

- **Print to file** - Selects for the report to be saved to a disk file.
- **Print to database** - Stores the report in the database in a **Folder** of your choice.
- **All pages, the Current page, or a range of Pages.**
 - Select the number of **Copies** to print and if you want to **Collate** them.

Displaying the Audit Trail and Event Log

As you work with Communiqué, the software tracks changes being made in Audit trails and Event logs.

As defined by 21 CFR Part 11, an Audit Trail is a secure, computer-generated mechanism used to "independently record the date and time of operator entries and actions that create, modify, or delete electronic records". As described in the GAMP Special Interest Group document "Complying with 21 CFR Part Electronic Records and Electronic Signatures", audit trails are also used to record important system functions (see Event Log below). Audit trails must be automatically timestamped in an "unambiguous" manner, i.e. by recording the time zone or offset from GMT, cannot be over-written or modified, and must be maintained at least as long as the records to which they refer.

An Event Log is related to an Audit Trail (see above), and is used to record major events, such as login, logout, password changes and system functions, in a 21 CFR Part 11 compliant system.

How do I display the Audit Trail and/or Event Log?

1. From the Tools menu select **Audit Trail**.
The Audit Trails dialog is displayed.
2. Choose whether to View the Audit Trail, Event Log, or Audit Trail andEvent Log.
Within these views you can filter what data is displayed.
3. Choose what you want to **Filter on**.
If you choose **Selected Template** or **Selected Report**, select the required Template or Report from the drop down list.
4. To order the items in the table, click on the required column heading.
To use the reverse order, click on the same column heading again.

What do the columns mean?

Report/Template	The name of the report or template then action was performed on.
Revision	The revision number of a template.
Timestamp	The date and time of the action.
User	The user that performed the action.
Event Name/Reason	The name of the event or action performed.
Event Comment/Comments	The comment saved with the event or action.

What can I do with the Audit Trail/Event Log?

As well as viewing the report, you can Print or Export it:

- Click **Print** to print the details displayed in the table.
On the Print dialog that is displayed select the required options.
- Click **Export** to save the details displayed in the table as a file.

How do I display details of the Audit Trail?

1. To get the full details of an item in the Audit trail (but not the Event Log), double click the item in the table.
A Details window is opened showing further information about that item.
2. Click **Print** to print these details.
On the Print dialog that is displayed select the required options.
3. Click **Export** to send these details to file.
On the Print dialog that is displayed select the required options.
4. Click **Done** to close this window.

What is a Report ID?

A Report ID is a serial number that is associated with every report generated, whether that be for previewing, printing, saving to a file or saving to the database. The Report ID will always be included in the event log when a report is generated, whether or not the value actually appears on the report itself.

Report IDs are unique for each instance of report generation. Thus, if all four forms of 'persistent' report (a preview is transitory) are generated at the same time, each one will have a different Report ID associated with it in the event log (and in the report itself if the report includes the Report ID object).

NOTE: A new Report ID is not generated when a report saved to the database is reprinted.
Such a reprinted report will always show the original Report ID.

Understanding the Menu Commands

The menus in Communiqué have the following functions:

What do the commands on the File menu do?

New	Starts a new template and adds it to the tree as discussed in How do I open another template .
Open	Enables you to open previously saved templates by displaying a file selector. This will only display templates that belong to the same parent application. When the template is loaded it is added to the tree as discussed in How do I open another template .
Import...	Cascading menu of Templates , which enables a template to be read in off a disk, and Custom Objects , which enables a custom object to be read in off disk. This in turn displays a standard Microsoft file selector.
Export...	Cascading menu of Templates... , which enables a template to be saved to disk, and CustomObjects... , which enables a custom object to be saved to disk. This in turn displays a standard Microsoft file selector.
Save	Saves the current template, with the current name. If it hasn't been saved before then this acts like Save As below.
Save As	Displays a file selector to enable the template to be named and saved. Once named this name replaces New Template in the Banner.
Print preview	Enables the user to see how the template looks, as discussed in Previewing a template .
Print	Displays the Print dialog discussed in Printing a template .
Close Template	Closes the currently selected template, asking whether to save any changes first, as discussed in How do I open another template .
Exit	Closes Communiqué, asking whether to save any changes first.

What do the commands on the Edit menu do?

Undo	Undoes up to the last 10 actions.
Redo	Redoes up to the last 10 actions. This is only available if Undo has been used.
Cut	Standard Windows clipboard behaviour. Cut is only available if one or more objects are selected.
Copy	Standard Windows clipboard behaviour. Copy is only available if one or more objects are selected.
Paste	Standard Windows clipboard behaviour. Paste is only available when something is on the clipboard.

Select All	Selects everything on the page.
Delete	Deletes the selected tool or object. This is only available when one or more objects are selected.
Rename...	Enables you to change the name of a page or item on the tree view.

What do the commands on the View menu do?

Grid	Turns the grid on and off, enabling you to position things properly, as discussed in How do I use the Grid?
Rulers	Turns the rulers on and off, enabling you to position things properly, as discussed in How do I use the rulers?
Object Toolbox	Enables you to switch the view in the Object Toolbox between Layout Tools, Data Objects and Custom Objects.
Toolbar	Switches the toolbar on and off.
Zoom	Enables you to switch the view in the Main Pane as discussed in How do I zoom in on the Main Pane?

What do the commands on the Format menu do?

Align	Cascading menu of Top , Middle , Bottom , Left , Center , Right , Height , and Width . Aligns selected objects in the manner chosen. Only available if more than one object is selected. See How do I align items? .
Default font	Displays a standard font selector, which becomes the default for all objects on the template that have not been manually set to something other than the original default. See How do I set Font properties?
Default number format	Sets the default format for all numeric data objects on the template that have not been manually set to something other than the original default. See How do I set the properties of a numeric object?
Default date/time format	Sets the default for all date/time data objects on the template that have not been manually set to something other than the original default. See How do I set the properties of a Date/Time object?
Measurement units	Sets the measurement units to be used on all dialogs, as discussed in How do I change Measurement Units?
Edit Border	Puts a border around a tool, object, section or group, just as the Edit Border layout tool.
Edit Caption	Puts a caption to a tool, object, section or group, just as the Edit Caption layout tool.
Properties	Displays the properties dialog for the currently selected object or objects.
Indexing...	Display the Indexing dialog as discussed in Indexing .

What do the commands on the Actions menu do?

Order objects	Cascading menu that displays the following commands: Bring to front , Send to back , Bring forward , and Send Backward . Only available when one or more objects are selected. See How do I set the order of items?
Group objects	Enables you to group a set of objects together as discussed in How do I group items?
Ungroup objects	Enables you to break a grouping created previously. Only available when a group is selected.
Center and Zoom	A toggle switch that determines whether the page display is centered (and optionally zoomed) when an object is selected in the tree view.
Center and Zoom Options	Displays the Center and Zooms Options dialog from where you can set an optional zoom factor associated with the object center feature.
Create custom object	Displays a file selector to enable the user to save a special object, populated table or group of items as a known object that will from then on appear under the Custom object selector. Only available if an object or group of objects is selected. See Using Custom Data Objects .

What do the commands on the Tools menu do?

Organize custom objects	Displays a dialog that enables you to delete or re-order custom objects in the Custom Object group as discussed in How do I reorganize the list of custom objects? . Only available if the Object Toolbox is set to the Custom Objects group and there is at least one object in it.
Organize keywords	Displays a dialog that enables the user to add and delete words on the keyword list, as discussed in How do I organize my keywords?
Audit trail	Displays the dialog used to view audit trails and event logs as discussed in Displaying the Audit Trail and Event Log .

What do the commands on the Help menu do?

Communiqué Help	Displays this on-screen help system at the opening page.
Display Tooltips	Turns the tooltips on and off.
About Communiqué	Displays a dialog that shows what version of Communiqué this is.

What do the commands on the Context menu do?

The context menu is the menu displayed when you right mouse click on an item:

Cut	Standard Windows clipboard behavior. Only available when one or more objects are selected.
Copy	Standard Windows clipboard behavior. Only available when one or more objects are selected.
Paste	Standard Windows clipboard behavior. Only available when there is something on the clipboard.
Group	Groups selected objects. Only available when one or more objects are selected.
Ungroup	Ungroups selected objects. Only available when a group is selected.
Delete	Deletes the selected tool or object. Only available when one or more objects are selected.
Edit border	Adds a border around the selected item.
Edit caption	Adds a caption to the selected item.
Font...	Displays the font dialog enabling the font to be set for the selected section of text.
Format	Displays the correct format dialog for the currently selected tool or object.
Object properties	Will display the correct properties dialog for the selected object.
Indexing	Displays the Indexing dialog as discussed in Indexing .
Order objects	Is a cascading menu that then has the commands Bring to front , Send to back , Bring forward , and Send Backward . Only available when one or more objects are selected.

Other Interactions

There are other standard interactions with the software detailed below:

How do Import and Export work?

Exporting

You can export a template or custom object to disk so that it can be used on another PC.

NOTE: Before you can export a template, you must save it.

1. Select **Export** and then **Template** or **Custom Object** from the File menu.
2. On the file selector, choose where to save the item and give it a name.

Importing

You can import a template or custom object that was exported from another copy of Communiqué.

3. Select **Import** and then **Template** or **CustomObject** from the File menu.
4. Use the file selector to locate the required template or object.

NOTE: If you import a template that uses fonts not available on your PC the template will use a different font as set by the Windows operating system.

What happens if the file name of an imported/exported item exists?

When template or item is Imported or Exported, a file of the same name may already exist in which case you will have to select what action to take. Depending on the circumstances, you can select between:

- **Change version** - Will import the file with the same name but incremented version number.
- **Change name** - Will import the file with the new name you type in.
- **Overwrite** - Will overwrite the previous file with this version.

How do I change Measurement Units on dialogs?

You can change the measurement units used on dialogs:

1. Select **Measurement Units** from the Format menu.
The Measurement Units dialog is displayed.
2. Select **mm**, **cm** or **inches** as the measurement unit to be used on all dialogs.
When you change the units, any dialog that mentions units is updated to the new unit system (except for the grids setting on the Template level) as are the graduations on the ruler.
This does not change any dimensions already set, the numbers on the dialogs currently are converted to the new measurement scale. For example, if the measurement units are changed from cm to mm, a dialog that had a setting of 1.2 cm will read 12 mm.

How do I Zoom in on the main pane?

Zooming in is particularly useful when the text on a page is too small to read:

- Select **Zoom** from the View menu and on the dialog displayed, select the required view:
- You can set a **Percentage** either from the drop down list of **400, 200, 150, 100, 75, 50, 25**, and **10**, or by typing in any required number.
- You can size the page to the **Page width** of the page.
- You can size the page so that the **Whole page** can be seen.

NOTE: View templates at 100% to get a better representation Print Preview.

UV WinLab Data Objects

UV WinLab Data Objects

The topics within the UV WinLab Data Objects book in the Help explain how to add data objects to your report template, and they also give examples of what information is shown when the objects are populated.

For information about each of the data objects within the list, see

- Global
- Task List
- Method List
- Sample List
- User List
- Instrument List
- Query Results List
- System Data.

NOTE: The IPV data objects are listed but they are only ever populated in templates that are associated with IPV tests. It is not possible to change the default templates that are associated with the IPV tests. It is possible to edit the default templates but WE STRONGLY RECOMMEND THAT YOU DO NOT DO THIS. Therefore, these data objects will not be discussed in the Help.

NOTE: There are a number of report templates that are installed when you Add your instrument. There is a default template for each base method that will produce a simple report of your results. There is also a default template for each type of method that will provide a summary of all the details of the task.

NOTE: Some UV WinLab Custom Data Objects are also provided. These include tables and graphs, such as an expanding results table, that are designed to help you create your own templates more easily. See the Communique Help for more details on using Custom Objects.

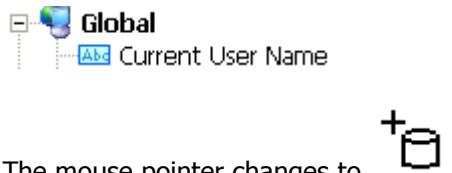
Global

This major branch has general information about the software, current user, and database.

How do I add the Current User Name to my report template?

The **Current User Name** data object will place the name of the user currently logged in into the report.

1. Click on **Current User Name** in the Data Object list to select it.



2. The mouse pointer changes to .
3. Position the mouse pointer on the template and drag to the required size.

The following object is added to the template.



The Current User Name object is added to the tree on the left hand side of the template.

To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

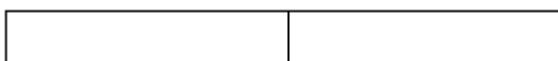


NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. This is described below.

5. Select the Table Layout tool from the Layout Tools list.

The mouse pointer changes to .

6. Position the mouse pointer on the template and drag to create a table.
Table is added to the tree
7. To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.
8. Select the Table tab and enter the number of columns and rows.
In the example below there are two columns and one row.

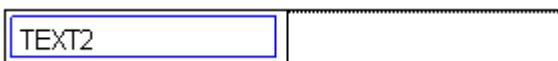


9. Select the Text Block layout tool



The mouse pointer changes to

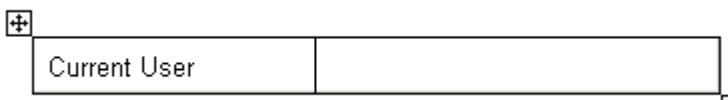
10. Click the mouse inside a field of the table.
A text object is placed in the field:



The text object is added below table on the tree:



11. Click inside the blue box in the table and edit the text as required.
In the example below '**Current User**' has been entered.



12. Click on **Current User Name** in the Data Object list to select it.

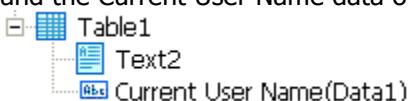


The mouse pointer changes to

13. Position the mouse pointer in the empty field in the table and drag to the required size.
The table now looks like:



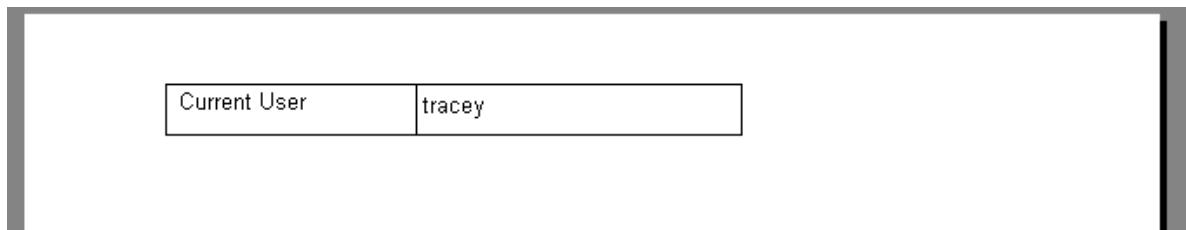
and the Current User Name data object is added to the tree



To view what will actually appear when the report is printed you need to print preview the report.

14. Click .

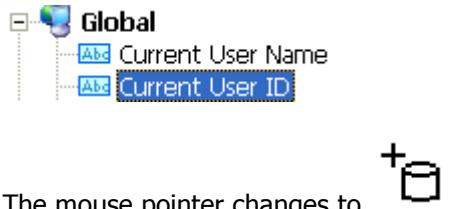
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



How do I add the Current User ID to my report template?

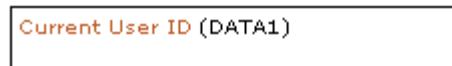
The **Current User ID** data object will place the ID of the user currently logged in into the report.

1. Click on **Current User ID** in the Data Object list to select it.



2. The mouse pointer changes to .

3. Position the mouse pointer on the template and drag to the required size.
The following object is added to the template.



The Current User ID object is added to the tree on the left hand side of the template.



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. In this example, the Current User ID is the same as the Current user Name (see above) so you would not know which data object you had placed on the page once the report was printed. You can create a table where you can enter text in one field and the data object in other field. This is described below.



5. Select the Table Layout tool



The mouse pointer changes to

6. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

7. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

8. Select the Table tab and enter the number of columns and rows.

In the example below there are two columns and one row.

--	--



9. Select the Text Block layout tool



The mouse pointer changes to

10. Click the mouse inside a field of the table.

A text object is placed in the field.

TEXT2	
-------	--



and the text object is added below table on the tree

11. Click inside the blue box in the table and edit the text as required.

In the example below '**Current User ID**' has been entered.

Current User ID	
-----------------	--

Click on **Current User ID** in the Data Object list to select it.



The mouse pointer changes to

12. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks like:

Current User ID	Current User ID (DATA2)
-----------------	-------------------------

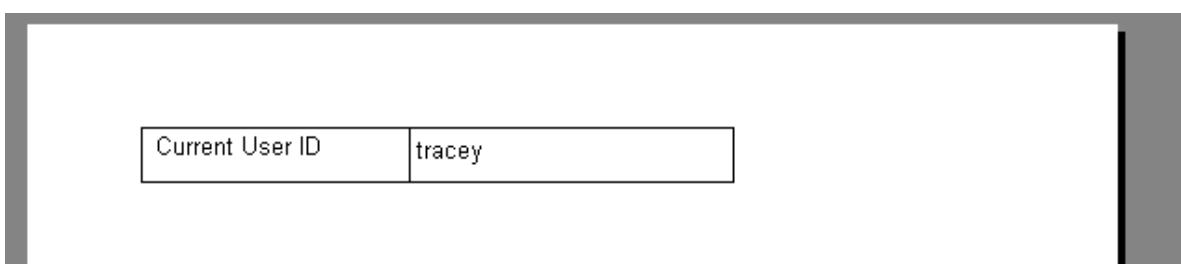


and the Current User ID data object is added to the tree.

To view what will actually appear when the report is printed you need to print preview the report.

13. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



How do I add the Software Version to my report template?

The **Software Version** data object will place the name and version of the software into the report.

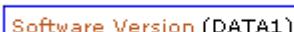
1. Click on **Software Version** in the Data Object list to select it.



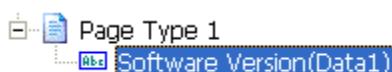
2. The mouse pointer changes to .

3. Position the mouse pointer on the template and drag to the required size.

The following object is added to the template



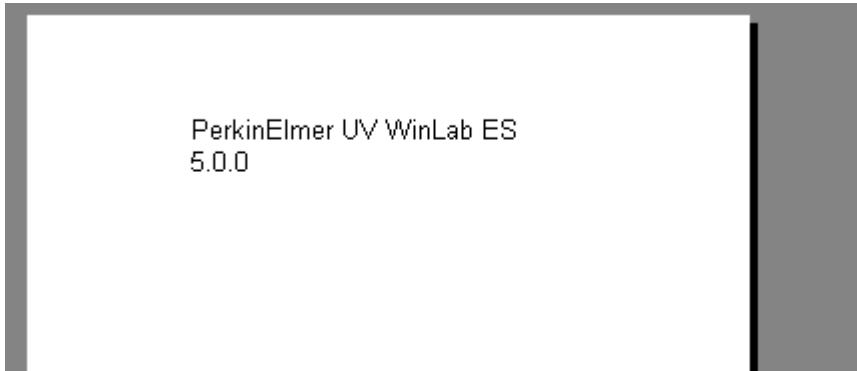
and the Software Version object is added to the tree on the left hand side of the template.



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. This is described below.



5. Select the Table Layout tool



The mouse pointer changes to

6. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

7. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

8. Select the Table tab and enter the number of columns and rows.

In the example below there are two columns and one row.



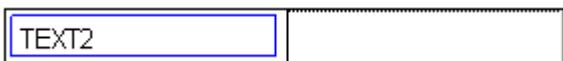
9. Select the Text Block layout tool



The mouse pointer changes to

10. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree.



11. Click inside the blue box in the table and edit the text as required.

In the example below '**Software Version**' has been entered.

Software Version	
------------------	--

12. Click on **Software Version** in the Data Object list to select it.

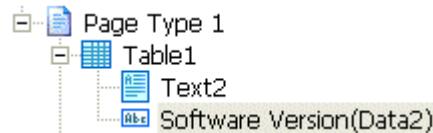


The mouse pointer changes to .

13. Position the mouse pointer in the empty field in the table and drag to the required size.
The table now looks like:

Software Version	Software Version (DATA2)
------------------	--------------------------

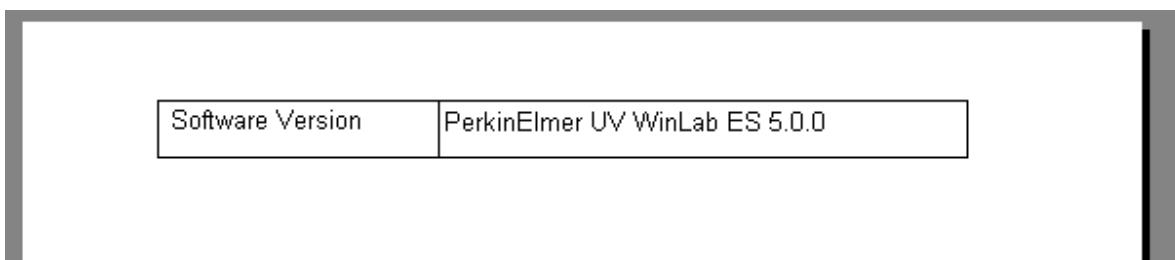
and the Software Version data object is added to the tree.



To view what will actually appear when the report is printed you need to print preview the report.

14. Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



How do I add the Database Version to my report template?

The **Database Version** data object will place the version number of the database into the report.

1. Click on **Database Version** in the Data Object list to select it.



2. The mouse pointer changes to .

3. Position the mouse pointer on the template and drag to the required size.

The following object is added to the template



and the Database Version object is added to the tree on the left hand side of the template.



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. This is described below.



5. Select the Table Layout tool .



The mouse pointer changes to .

6. Position the mouse pointer on the template and drag to create a table.
Table is added to the tree.



7. To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.
8. Select the Table tab and enter the number of columns and rows.
In the example below there are two columns and one row.





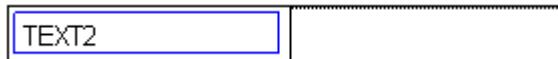
9. Select the Text Block layout tool



The mouse pointer changes to

10. Click the mouse inside a field of the table.

A text object is placed in the field.

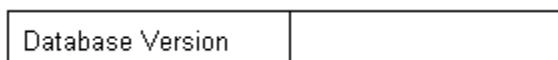


and the text object is added below table on the tree.



11. Click inside the blue box in the table and edit the text as required.

In the example below '**Database Version**' has been entered.



12. Click on **Database Version** in the Data Object list to select it.



The mouse pointer changes to

13. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks like:



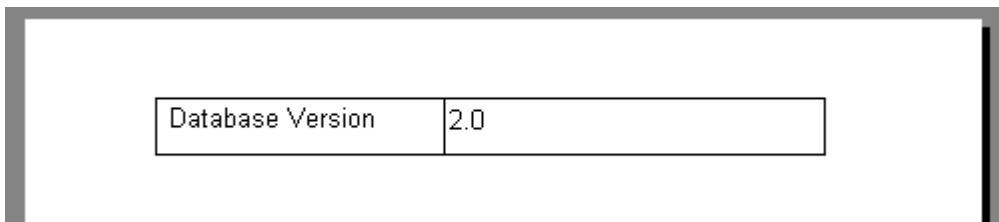
and the Database Version data object is added to the tree.



To view what will actually appear when the report is printed you need to print preview the report.

14. Click

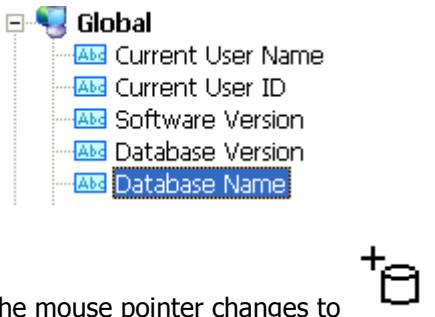
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



How do I add the Database Name to my report template?

The **Database Name** data object will place the name of the database into the report.

1. Click on **Database Name** in the Data Object list to select it.



2. The mouse pointer changes to .
3. Position the mouse pointer on the template and drag to the required size.

The following object is added to the template



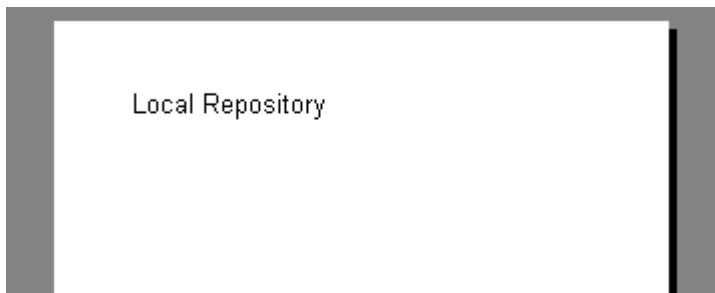
and the Database Name object is added to the tree on the left hand side of the template.



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. This is described below.



5. Select the Table Layout tool



The mouse pointer changes to

6. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree.



7. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

8. Select the Table tab and enter the number of columns and rows.

In the example below there are two columns and one row.



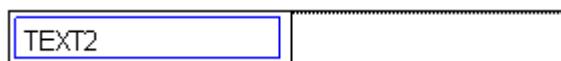
9. Select the Text Block layout tool



The mouse pointer changes to

10. Click the mouse inside a field of the table.

A text object is placed in the field.

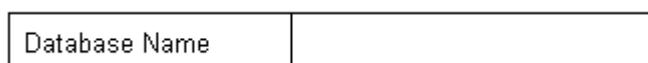


and the text object is added below table on the tree.



11. Click inside the blue box in the table and edit the text as required.

In the example below '**Database Name**' has been entered.



12. Click on **Database Name** in the Data Object list to select it.



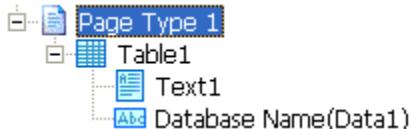
The mouse pointer changes to

13. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks like:

Database Name	Database Name (DATA2)
---------------	-----------------------

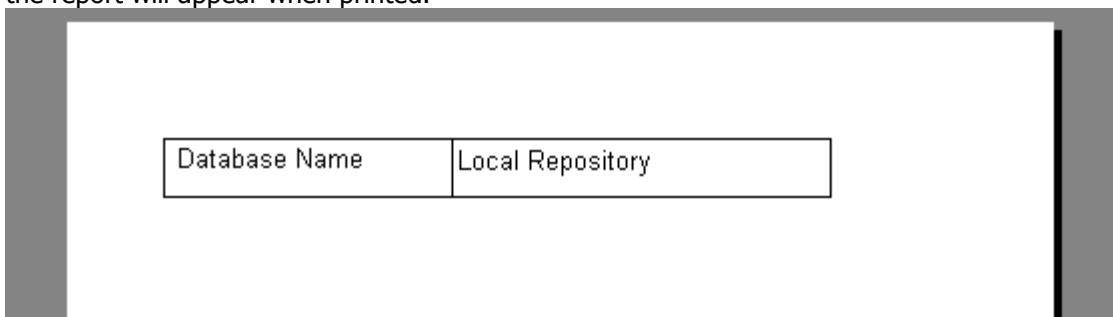
and the Database Name data object is added to the tree.



To view what will actually appear when the report is printed you need to print preview the report.

14. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



How do I add the Database Path to my report template?

The **Database Path** data object will place the location of the database into the report.

1. Click on **Database Path** in the Data Object list to select it.



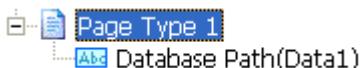
2. The mouse pointer changes to .

3. Position the mouse pointer on the template and drag to the required size.

The following object is added to the template

Database Path (DATA1)

and the Database Path object is added to the tree on the left hand side of the template.



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. This is described below.

5. Select the Table Layout tool  .

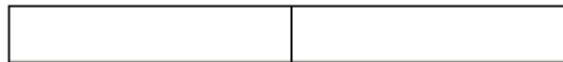


The mouse pointer changes to .

6. Position the mouse pointer on the template and drag to create a table.
Table is added to the tree.



7. To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.
8. Select the Table tab and enter the number of columns and rows.
In the example below there are two columns and one row.

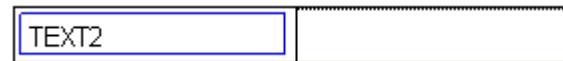


9. Select the Text Block layout tool  .



The mouse pointer changes to .

10. Click the mouse inside a field of the table.
A text object is placed in the field.



and the text object is added below table on the tree:



11. Click inside the blue box in the table and edit the text as required.
In the example below '**Database Path**' has been entered.

Database Path	
---------------	--

12. Click on **Database Path** in the Data Object list to select it.

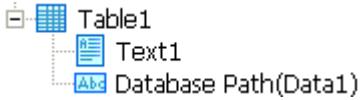


The mouse pointer changes to .

13. Position the mouse pointer in the empty field in the table and drag to the required size.
The table now looks like:

Database Path	Database Path (Data2)
---------------	-----------------------

and the Database Path data object is added to the tree.



To view what will actually appear when the report is printed you need to print preview the report.

14. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Database Path	C:\Documents and Settings\All Users\Application Data\PerkinElmer\UVWinLab\UVWinLab.mdb
---------------	--

How do I create a table in my template that contains all the Global information?



Table

1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

A table is added to the tree.

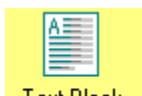


3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **2** columns and **6** rows.

There are 6 Global data objects. One column will have text names and one column will have the corresponding data objects.



Text Block

5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.

TEXT2	
-------	--

and the text object is added below table on the tree.



7. Click inside the blue box in the table and edit the text as required.

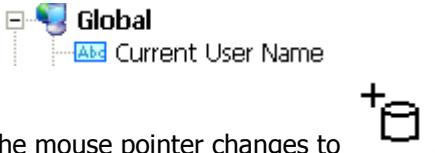
In the example below '**Current User**' has been entered first as this is the first data object listed under Global.

Current User Name	

8. Repeat steps 6 and 7 to enter text that will indicate what the data objects are.

Current User Name	
Current User ID	
Software Version	
Database Version	
Database Name	
Database Path	

9. Click on **Current User Name** in the Data Object list to select it.



10. Position the mouse pointer in the empty field adjacent to **Current User Name** in the table and drag to the required size.

The table now looks like:

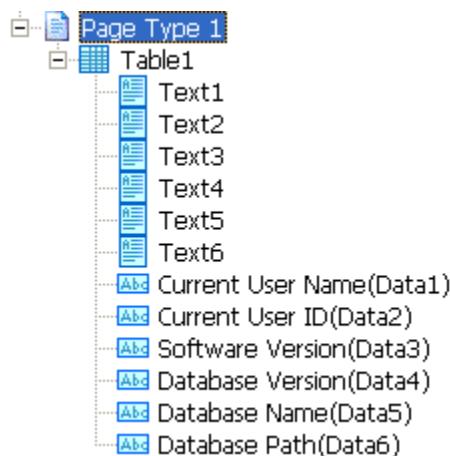
Current User Name	Current User Name (DATA1)
Current User ID	
Software Version	
Database Version	
Database Name	
Database Path	

- Repeat steps 9 and 10 for the other Global data objects.

The table will look like this:

Current User Name	Current User Name (DATA1)
Current User ID	Current User ID (DATA2)
Software Version	Software Version (DATA3)
Database Version	Database Version (DATA4)
Database Name	Database Name (DATA5)
Database Path	Database Path (DATA6)

and the tree will look like:



To view what will actually appear when the report is printed you need to print preview the report.

NOTE: The order of the objects in the tree depends on the order in which the items were added.

- Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Current User Name	tracey
Current User ID	tracey
Software Version	PerkinElmer UV WinLab ES 5.0.0
Database Version	2.0
Database Name	Local Repository
Database Path	C:\Documents and Settings\All Users\Application Data\PerkinElmer\UV\WinLab\UV\WinLab.mdb

Task List

How do I add the Name, Description, Creation Date, Modified Date and Status of the task to my report template?

This example puts this information in a table with text to explain what each of the objects are.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

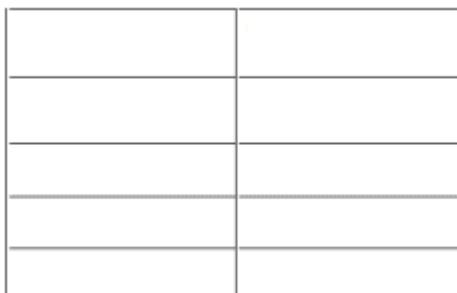
Table is added to the tree



3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter 2 columns and 5 rows.



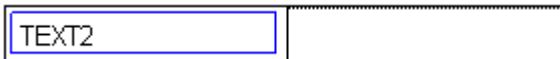
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field in the first row of the table.

A text object is placed in the field.



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Name' has been entered.

Name	

8. Repeat steps 5–7 for '**Description**', '**Date Created**', '**Date Modified**' and '**Status**' headings.

Name	
Description	
Date Created	
Date Modified	
Status	

9. Click on **Name** in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer on the template in the cell below 'Name' and drag to display.
The following object is added to the template

Name	Name (Data1)
Description	
Date Created	
Date Modified	

and the Name object is added to the tree on the left hand side of the template.

11. Repeat steps 9 and 10 for the **Description** and **Status** data objects.

The template will look like this:

Name	Name (Data1)
Description	Description (Data2)
Date Created	
Date Modified	
Status	Status (Data5)

The date data objects have not been added in the same way because these are affected by the size of the fields. If the field is too small, the date will display as #####. To avoid this, date data objects should be placed inside text blocks.



12. Select the Text Block layout tool



The mouse pointer changes to

13. Click the mouse inside the empty field for the Date Created field.



14. Remove the default text inside:

15. Select the **Date Created** data object and then click the mouse inside the text block you have just created:

Name	Name (Data1)
Description	Description (Data2)
Date Created	[D6:Date Created]
Date Modified	
Status	Status (Data5)

16. Repeat steps 12–15 for the **Date Modified** data object:

Name	Name (Data1)
Description	Description (Data2)
Date Created	[D6:Date Created]
Date Modified	[D7:Date Modified]
Status	Status (Data5)

To view what will actually appear when the report is printed you need to print preview the report.

17. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Name	Scanning quant 5 28 January 2004 14:37 GMT means
Description	3 standards
Date Created	01/28/2004 02:37:52 PM GMT
Date Modified	02/04/2004 05:35:26 PM GMT
Status	In progress

See the following for more information:

- Task Samples
- Table Objects
- Method
- Created By/Modified By
- Instrument
- Quant Calibration
- Task Event Log
- Measurements or Replicates Sample Tables
- Custom Table Samples.

Task List Task Samples

How do I include the ID, Description, Status and Comments for each of the samples in my task in the report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.



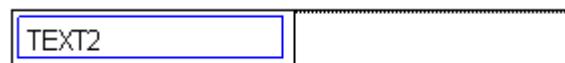
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below 'ID' has been entered.

ID			

8. Repeat steps 5–7 to enter text for the other output settings:

ID	Description	Status	Comments

NOTE: The text headings should be at the top of each column (rather than all rows in the first column) as the table will be repeated on the second row to obtain the information for all samples in the task.

- Click on the required data object in the Data Object list to select it.

	ID
	Full ID
	Description
	Type



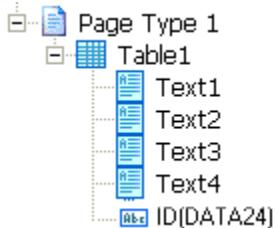
The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

ID	Description	Status	Comments
ID (DATA24)			

and the data object is added to the tree:



- Repeat steps 9 and 10 for the other output settings data objects to complete your table

ID	Description	Status	Comments
ID (DATA24)	Description (DATA25)	Status (DATA26)	Comments (DATA27)

To obtain the ID, description, status and comments for all the samples in the task, you need to repeat the second row of the table.

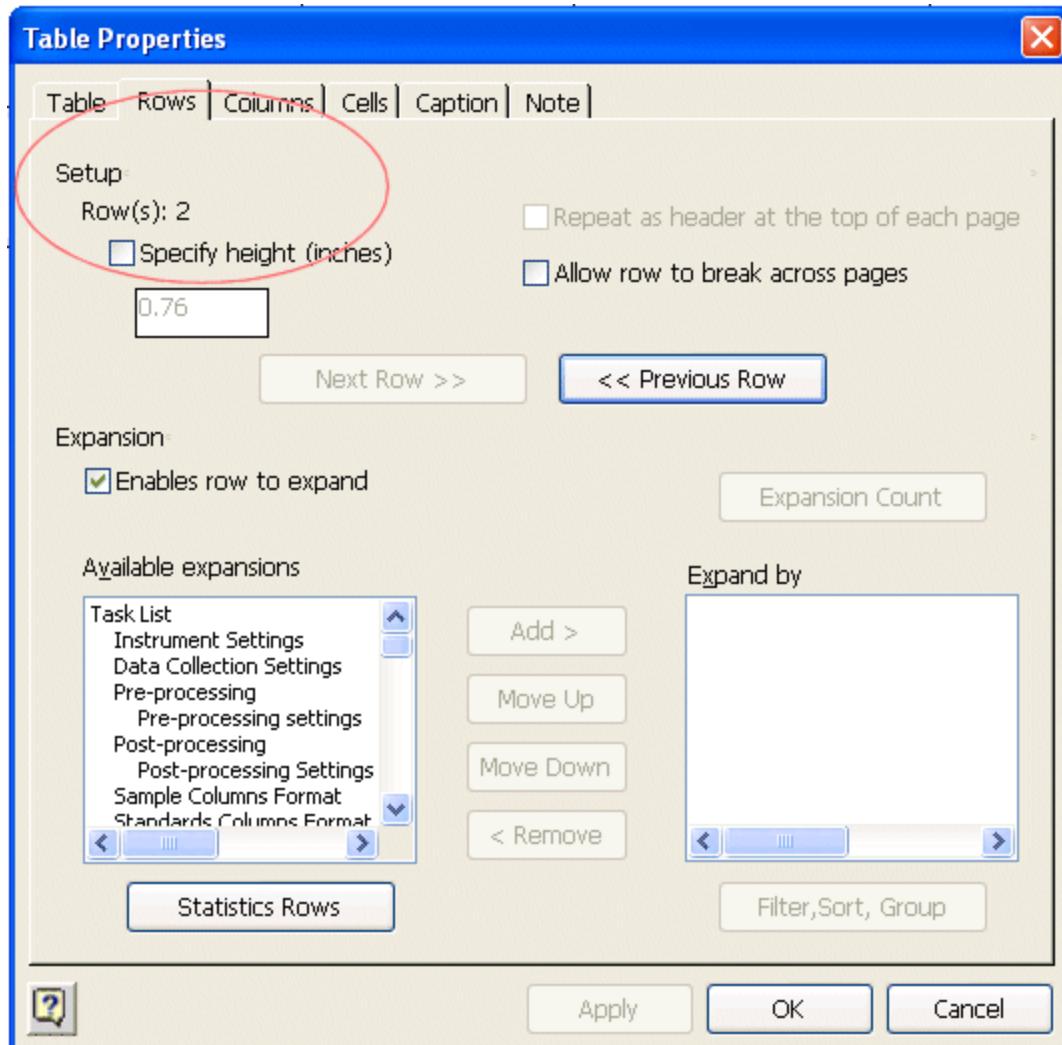
- Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

- Select the Rows tab.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.



15. Ensure Enables row to expand is selected.

16. From the list of Available expansions, select **Task Samples**, and then click **Add**. Samples moves from the Available expansions list to the Expand by list.

17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

ID	Description	Status	Comments
s10std1	Batch 10	Imported	
s10std2	Batch 10	Imported	
s10std3	Batch 10	Imported	
s10std4	Batch 10	Imported	

NOTE: In this example there are no comments associated with any of the samples and so the Comments fields are empty.

What is the difference between ID and Full ID?

The ID object will report the Sample ID as seen in the Sample Table. This is sufficient if you have a fairly simple Sample Table of, for example, just 5 samples.

ID
Sample38
Sample37
Sample39
Sample40
Sample41

However, this does not report the full sample name, which includes the extension (as seen in the Results Table). The **Full ID** is particularly useful if you are using Replicates or Measurements. If, for example, you have 1 sample with 2 replicates, and you create a table in the report template using **ID**, you will see the Sample ID twice but it will not show that these are replicates.

ID
Sample1
Sample1

If you use the **Full ID** data object, the full name including the extension will be shown.

Full ID
Sample1.Replicate1
Sample1.Replicate2

What is the Type data object?

The Type data object displays the information from the Type field in the Sample Table that is, what type of sample is to be run. For example, it could be a Sample, Blank, Control, Replicate or Measurement. If you are using the **ID** data object, you may wish to use the **Type** data object as well so that you can tell what type each sample is. However, if you use the **Full ID** data object, the extension (for example Sample1.Replicate) explains what type of sample it is and so the Type field is not necessary.

How do I create a table of all the sample names and types in my report template?

This example explains how to use the ID and Type data objects together. This information can be also be displayed using the Full ID data object (see "What is the difference between ID and Full ID ?" above).



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.



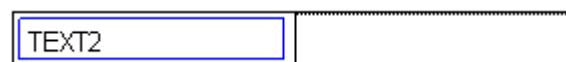
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below 'ID' has been entered.

ID	

- Repeat steps 5 to 7 to enter text for 'Type'.

ID	Type

NOTE: The text headings should be at the top of each column (rather than all rows in the first column) as the table will be repeated on the second row to obtain the information for all samples in the task.

- Click on the ID data object beneath Task Samples in the Task List.



The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

ID	Type
ID (Data1)	

- Repeat steps 9 and 10 for the Type data object to complete your table.

ID	Type
ID (Data1)	Type (Data2)

To obtain the ID and Type for all the samples in the task, you need to repeat the second row of the table.

- Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

- Select the Rows tab.

- Click Next Row.

Row 2 is now specified at the top of the dialog.

- Ensure Enables row to expand is selected.

- From the list of Available expansions, select **Task Samples**, and then click **Add**.

Task Samples moves from the Available expansions list to the Expand by list.

- Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

- Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

ID	Type
Sample1	Replicate1
Sample1	Replicate2
Sample2	Blank
Sample4	Control
Sample5	Replicate1
Sample5	Replicate2

How do I create a table that contains all the information displayed in the Results Table?

The simplest way to do this is to use the Results Table data object (See "How do I add the Results Table to my report template ?"). However, you can also create your own table (known as an expanding table). The example below shows how to create an expanding table for all the information in a Results Table. The table will be created with 2 columns. The first is for the sample ID. The second will expand so that all columns in the Results Table are reported.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--

and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below 'SampleID' has been entered.

Sample ID	

8. Select the **Title** data object beneath the Results Table Columns (within Task Samples in the Task List).

9. Click in the top cell of the second column:

Sample ID	Title (Data3)

10. Select the **ID** data object beneath Task Samples.

11. Click in the field beneath the Sample ID:

Sample ID	Title (Data3)
ID (Data1)	

The data object to be used in the final cell of the table depends on the data in the Results Table. If there is text as well as numbers, you should use the Text data object. If there is only numerical data, use the Number data object.

12. Select the **Number** data object beneath the Result Table Columns.

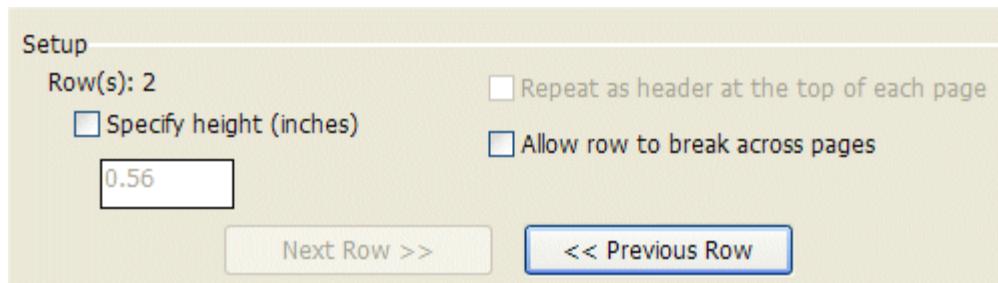
13. Click in the field beneath the Title data object.

14. Right-click on the table and select **Properties** from the menu.

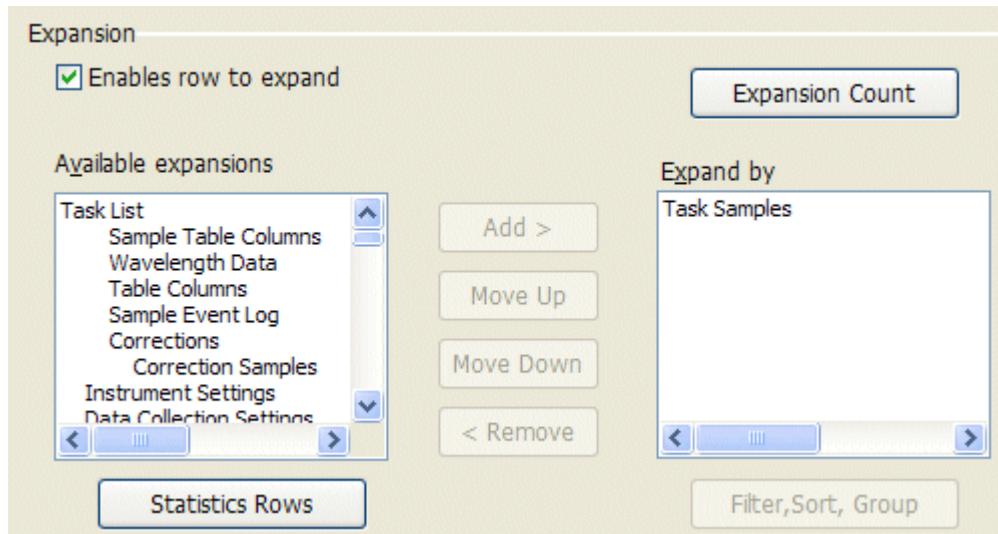
15. Select the Rows tab.

16. Click Next row.

Row 2 should be specified at the top of the Setup:



17. Select **Enables row to expand**, and then select **Task Samples** from the drop-down list.



18. Select the Columns tab.
19. Click **Next Column** so that Column 2 is specified at the top of the setup.
20. Select Enables column to expand, and then select Result Table Column.
21. Click **OK** to close the Properties dialog.

To view what will actually appear when the report is printed you need to print preview the report.

22. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

NOTE: If the Sample ID contains replicates or measurements extension, e.g. Sample1.replicate1, the extension is not reported in the table.

Sample ID	c x	Ordinate
Sample12	30.0927	0.4109
Sample13	19.9054	0.2587
Sample14	10.0675	0.1118
Sample12	10.0748	0.1119
Sample13	10.0622	0.1117
Sample14	10.0707	0.1118

NOTE: You must repeat on the same type that is specified in the table. In the example above, the text data object from the Result Table Column is used in the table, and then the column is expanded by Result Table Column. If you were to repeat by Sample Table Column in this example, the table would not expand and populate correctly.

If you wish to create an expanding table for the sample table, use the procedure described above, but change the following:

- In step 8, use the **Title** data object within the Sample Table Columns.
- In step 12 use the **Text or Number** data object within Sample Table Columns.
- In step 20, select Enables column to expand by Sample Table Columns.

How do I include the analyst name and ID and the date the samples were analyzed in my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.
The size will depend on how many options you wish to include.

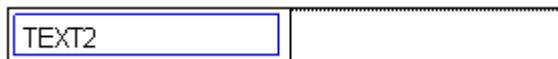


5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field of the table.
A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.
In the example below 'Analyst Name' has been entered.

Analyst Name	

8. Repeat steps 5–7 to enter text for Analyst ID and Date Analyzed.

Analyst Name	
Analyst ID	
Analysis Date	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

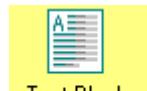
Analyst Name	Name (Data1)
Analyst ID	
Analysis Date	

and the data object is added to the tree.

- Repeat steps 9 and 10 for the analyst ID data object.

Analyst Name	Name (Data1)
Analyst ID	ID (Data2)
Analysis Date	

When using a date data object, we recommend that the data object is placed inside a text block. This ensures that all the date is displayed. If a text block is not used, and the field is not large enough to display the date, you will see ##### in the field.



- Select the Text Block from the layout tools:



The mouse pointer changes to

- Drag the mouse on the report template to create a Text Block:



- Click inside the Text Block and remove the default text.

- Select the Date analyzed data object.

16. Click inside the Text Block.

The object is placed inside the Text Block:

[D1:Date Created]

Analyst Name	Name (Data1)
Analyst ID	ID (Data2)
Analysis Date	[D3:Date Analyzed]

To view what will actually appear when the report is printed you need to print preview the report.

17. Click .

Analyst Name	tracey
Analyst ID	tracey
Analysis Date	04/26/2004 04:23:53 PM BST

Raw Spectrum/Processed Spectrum

How do I display the raw spectrum of each sample on a separate graph?

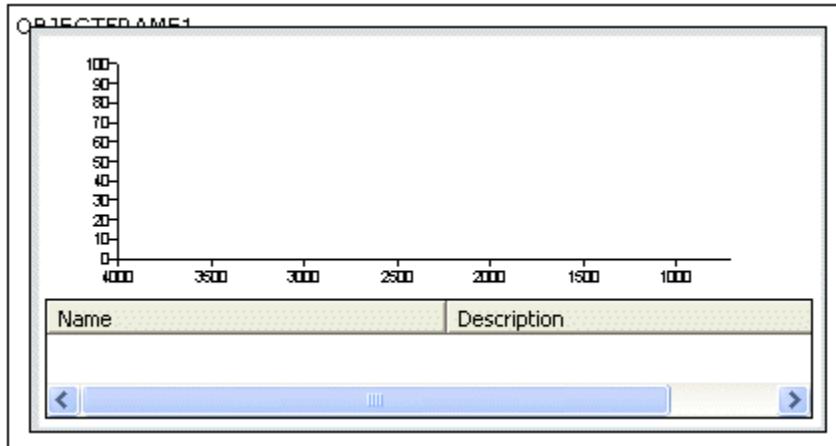
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



To display the raw spectrum for each sample on a separate graph, a section must be created.

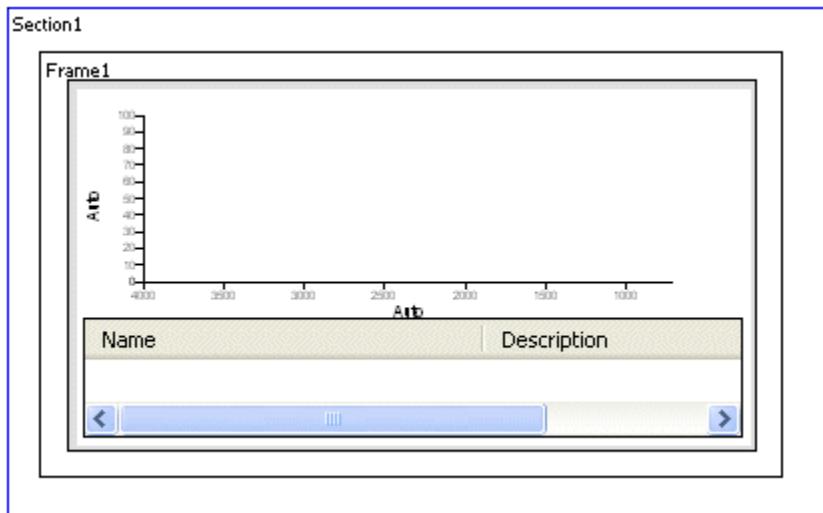


3. From the Layout Tools list, select .



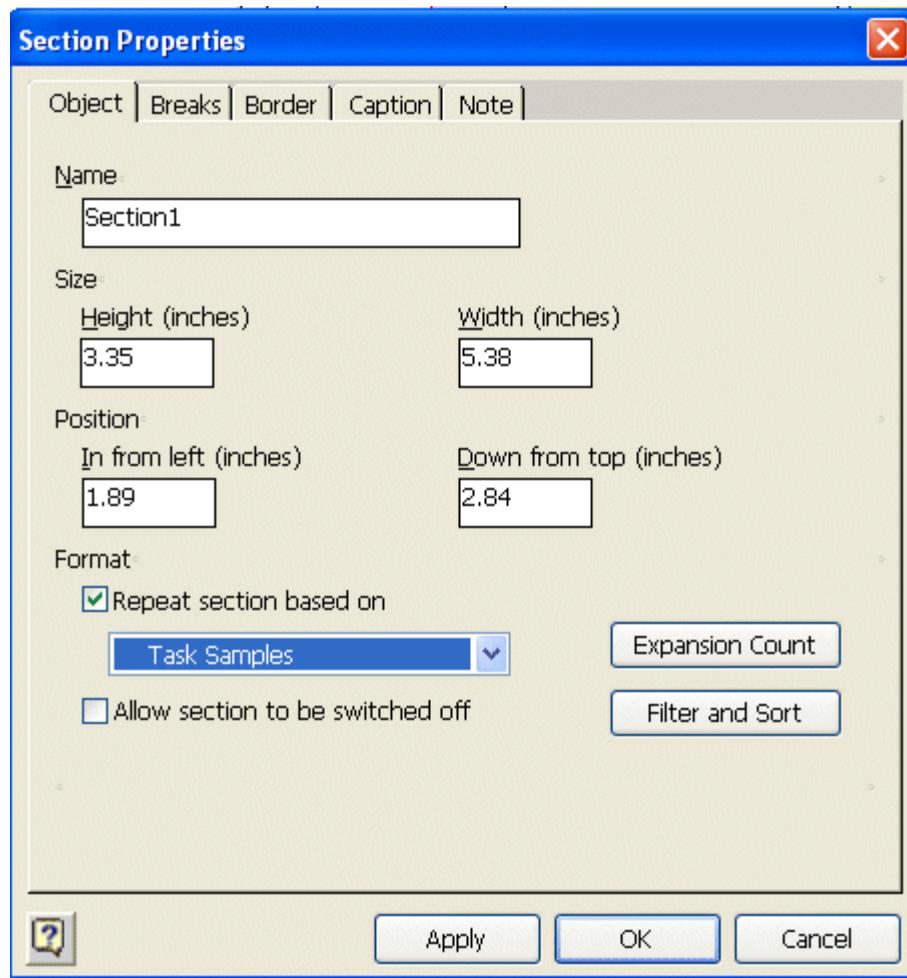
The mouse pointer changes to .

4. Drag the mouse around the object frame to create a section containing the spectrum:



The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.



6. Click **OK**.

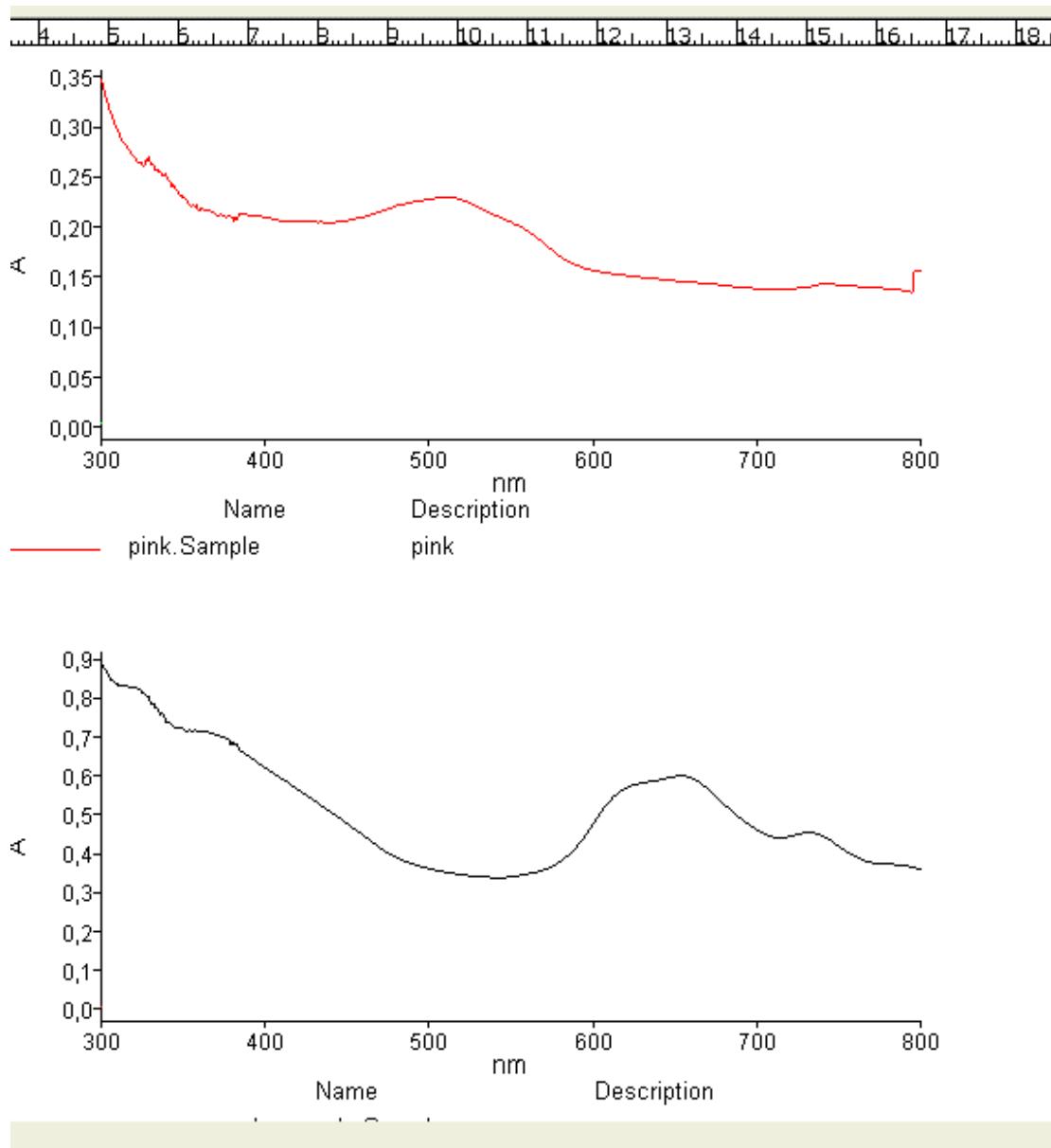
The section is added to the tree:



To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

An example is shown below:



How do I display the processed spectrum of each sample?

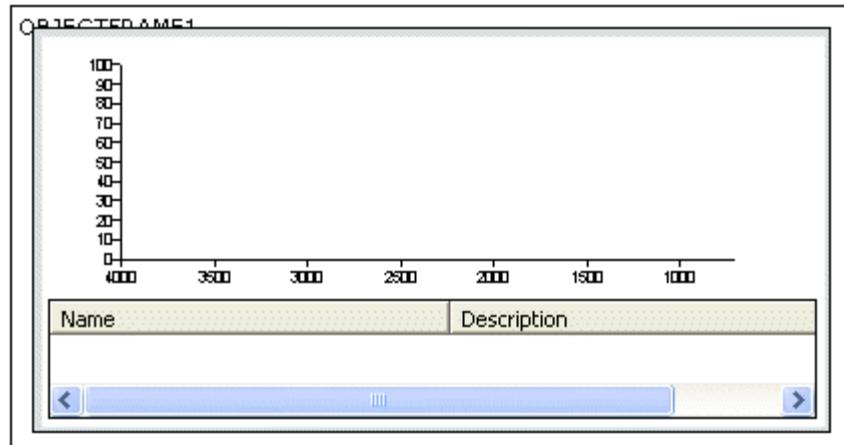
1. Select the **Processed Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



To display the processed spectrum for each sample, a section must be created.

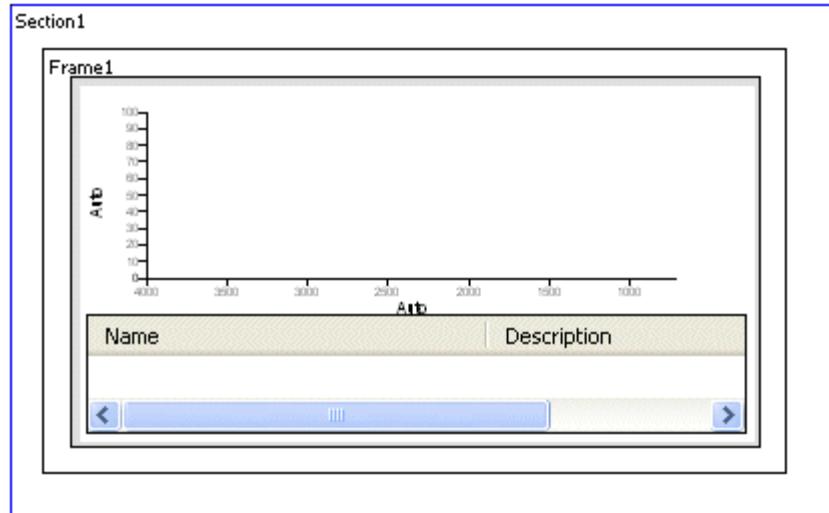


3. From the Layout Tools list, select **Section**.



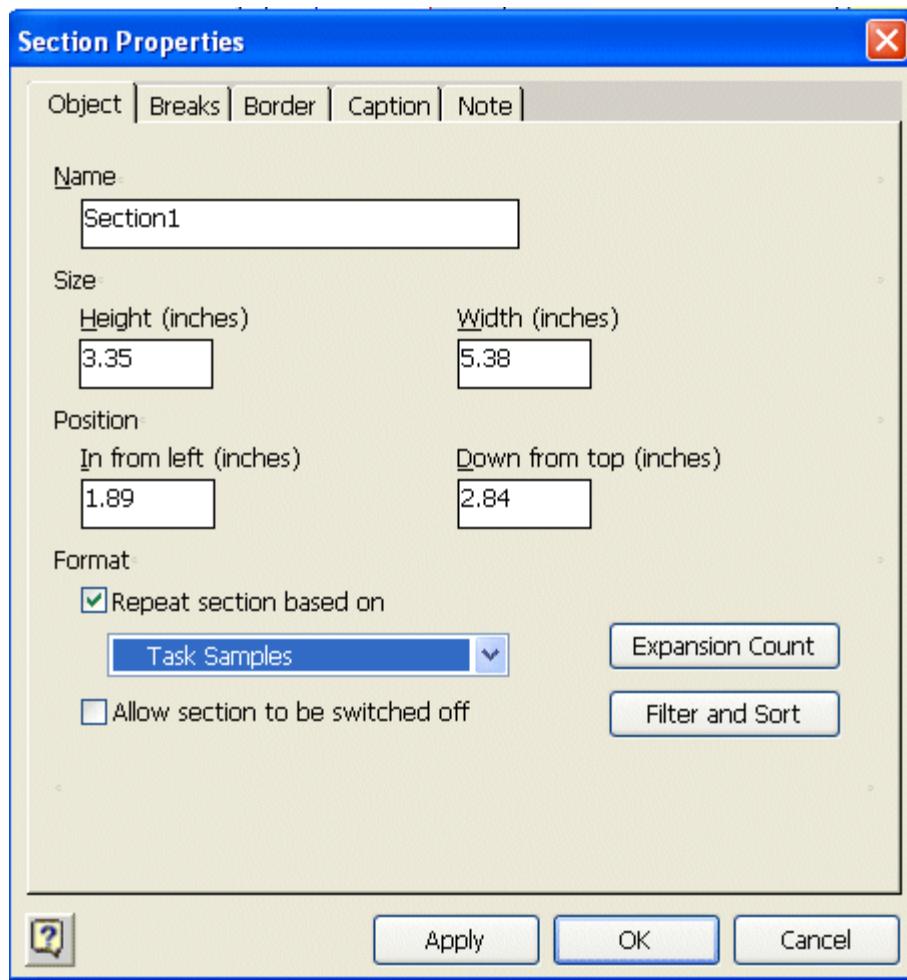
The mouse pointer changes to **+... .**

4. Drag the mouse around the object frame to create a section containing the spectrum:



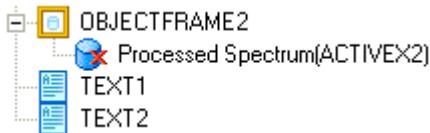
The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.



6. Click **OK**.

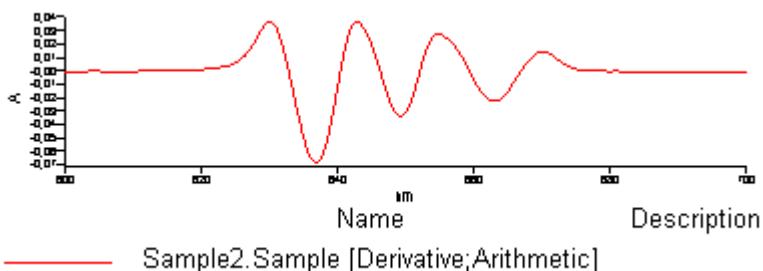
The section is added to the tree:



To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

Example:



How do I overlay the raw spectra of each sample on the same graph or How do I overlay the processed spectra of each sample on the same graph?

The example below uses the Raw Spectrum data object. To overlay all processed spectra on one graph, follow the instructions below but use the Processed Spectrum data object.

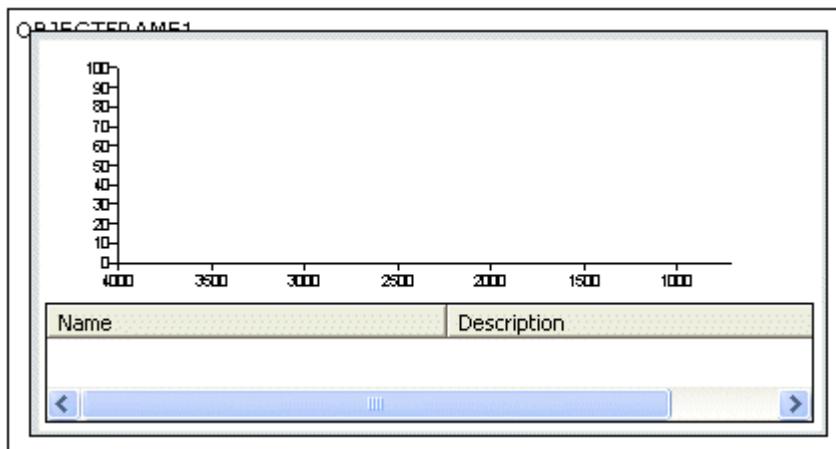
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



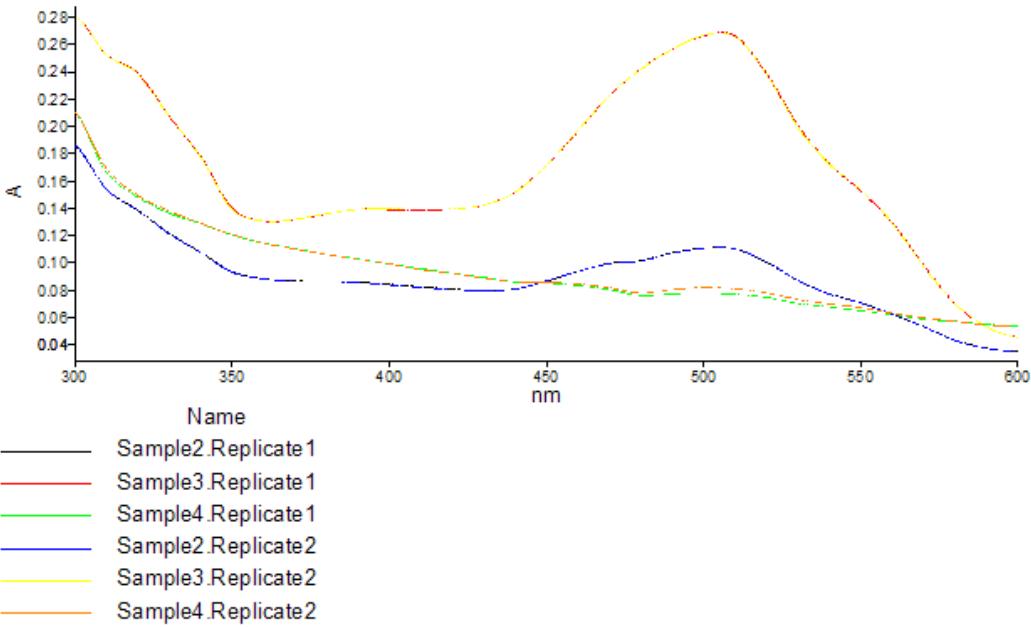
3. Right-click on the frame and select **Properties**.

4. Select the Sequence tab.

5. Select **Repeat based on**, and then select **Task Samples** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

6. Click



How do I overlay the raw and processed spectra of each sample?

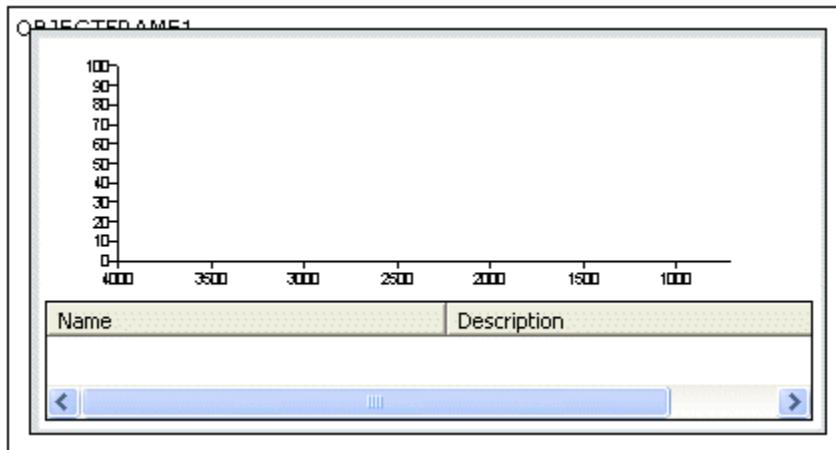
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



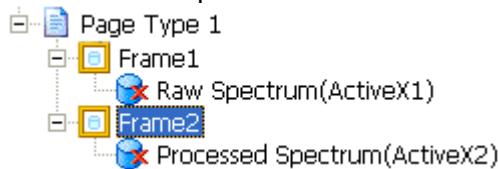
3. Select the **Processed Spectrum** data object.



The mouse pointer changes to .

4. Drag the mouse exactly over the raw spectrum object.

The Processed Spectrum frame is added to the tree:



To display the overlaid spectra for each sample (each sample on a separate graph), a section must be created.

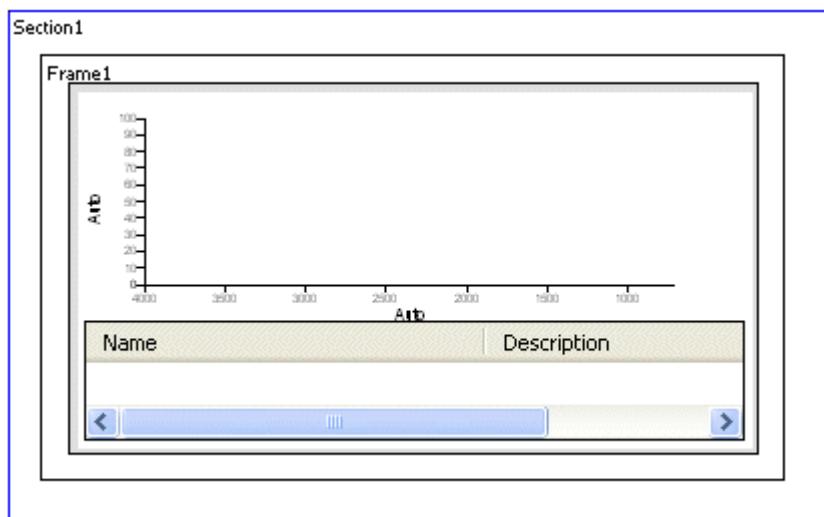


- From the Layout Tools list, select **Section**.



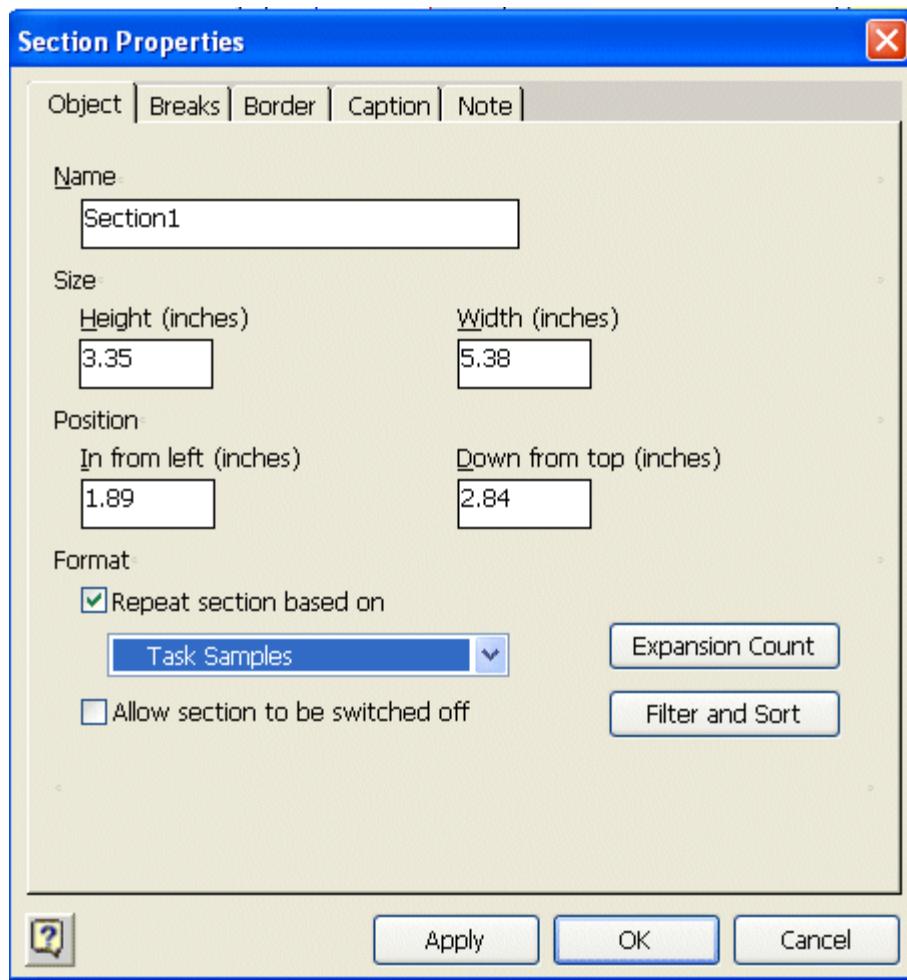
The mouse pointer changes to .

- Drag the mouse around the object frame to create a section containing the spectrum:



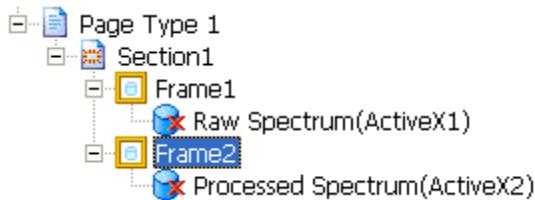
The Section Properties dialog is displayed.

- Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.



8. Click **OK**.

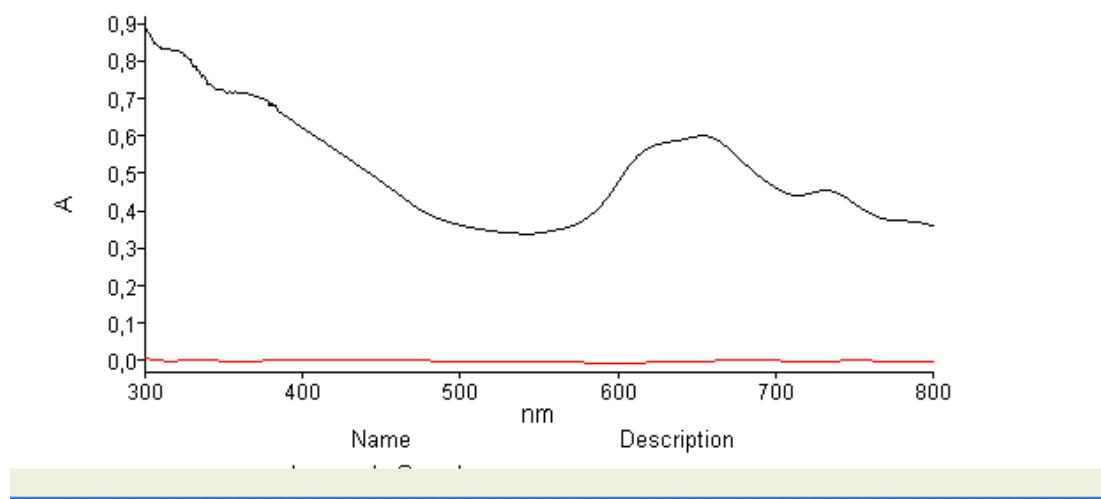
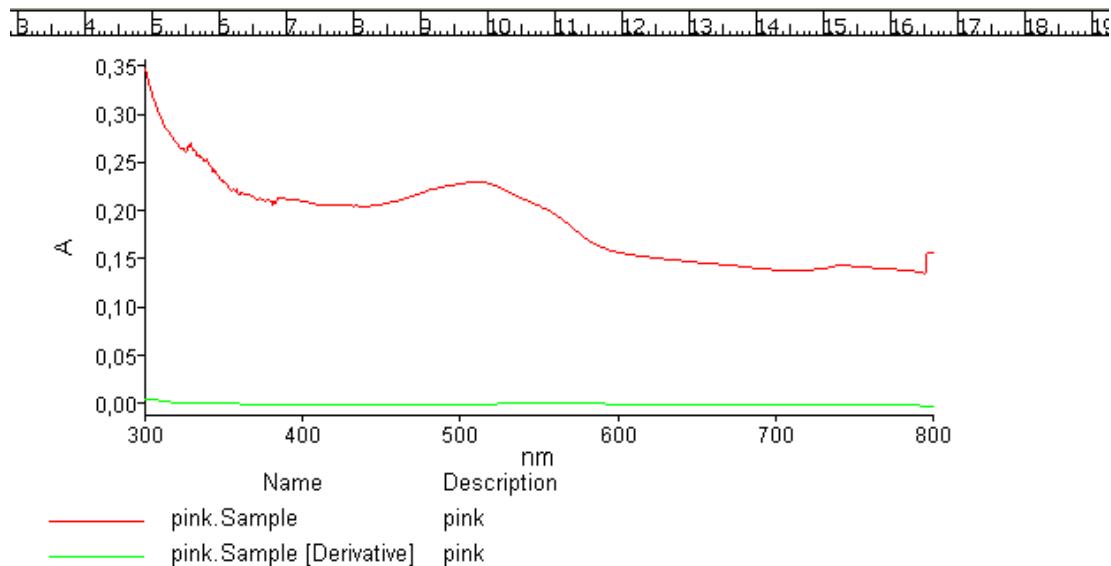
The section is added to the tree:



To view what will actually appear when the report is printed you need to print preview the report.

9. Click .

An example is shown below:



Wavelength Data

How do I add a Wavelength Table to my report template?

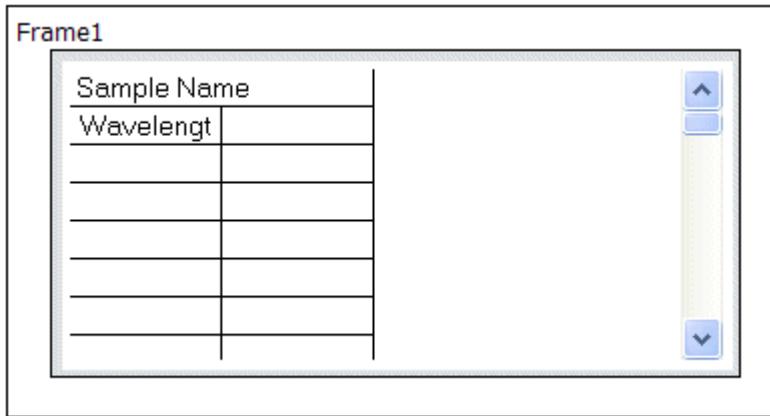
This data object creates a table that lists the absorbances for each of the wavelengths specified in the task. By default only the data relating to the first samples is displayed. You must use a section to display the information for all samples.

1. Select the **Wavelength Table** data object.



The mouse pointer changes to

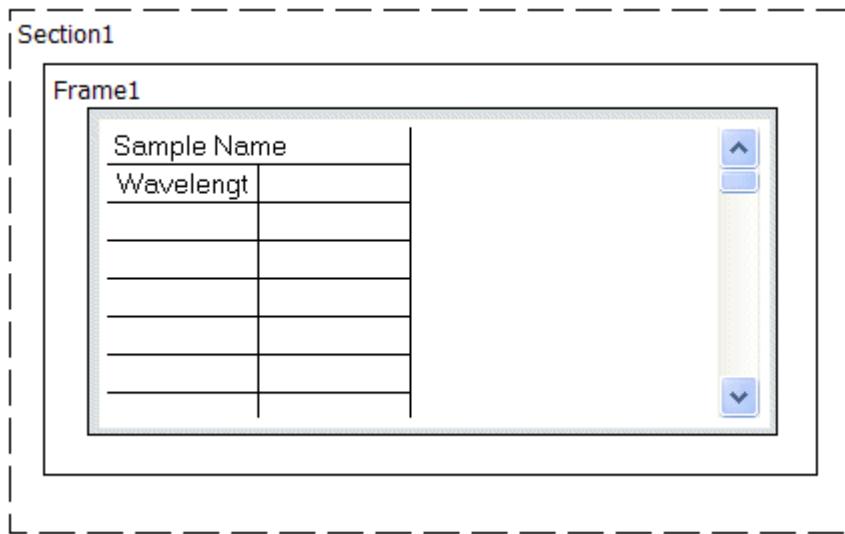
2. Click and drag the mouse to create a frame:



The size of the Frame determines the size of the table in the printed report.



3. Select the Section layout tool
4. Create a section around the frame:



The Section Properties dialog is displayed.

5. Select **Repeat section based on**, and then select **Wavelength Data** from the drop-down list.
6. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

The report is displayed. This is how the report will appear when printed.

Sample1.Sample	
Wavelength (nm)	Value (A)
175	0.94224
865	0.720284
3300	10

Sample2.Sample	
Wavelength (nm)	Value (A)
175	0.231817
865	1.35717
3300	10

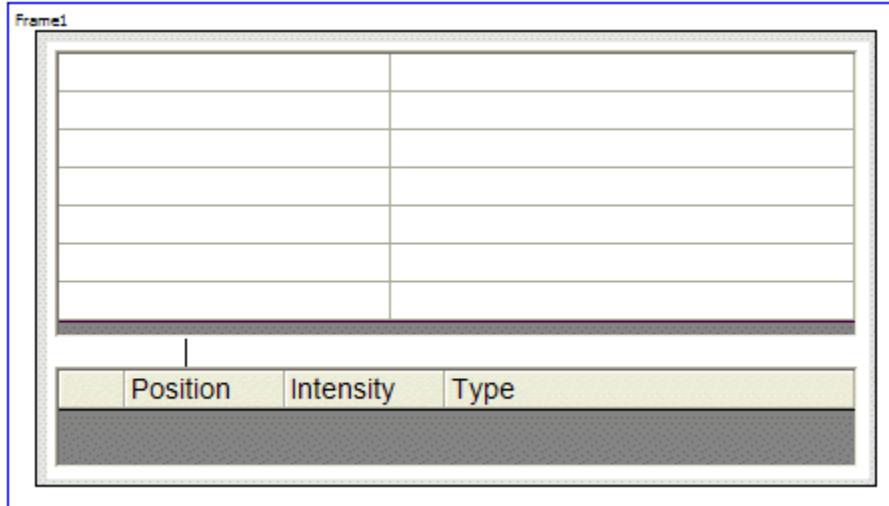
Sample3.Sample	
Wavelength (nm)	Value (A)
175	-0.00440505
865	1.35774
3300	10

Peak Table

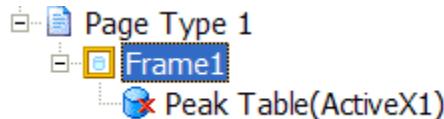
How do I include a peak table in my report template?

A peak table must have been setup as part of the processing for it to be available within the report template.

1. Select the **Peak Table** data object within the Task List.
2. Click the mouse on the report template and drag to create the object:



The size of the frame will determine the size of the peak table in the report. If the frame is too small, not all of the results will be displayed.



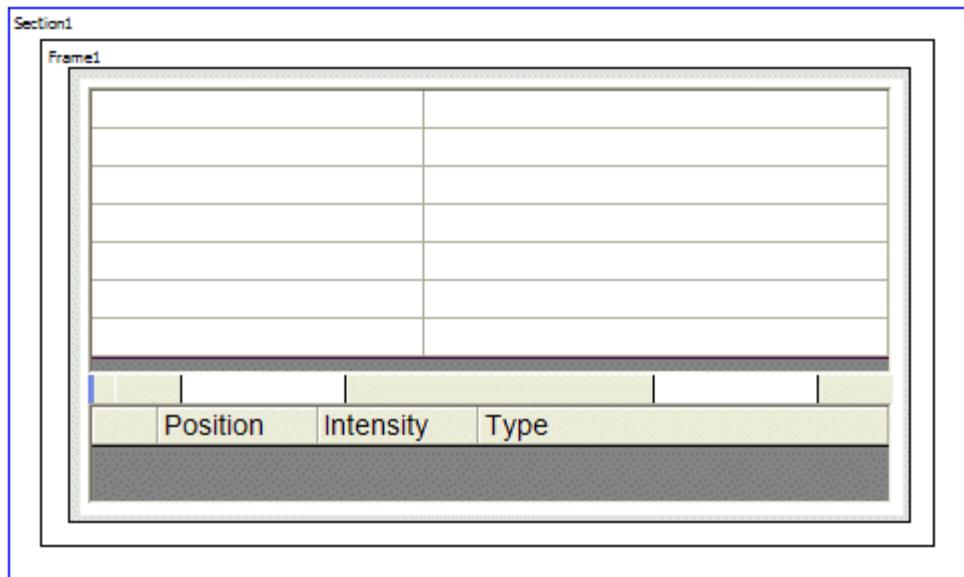
The tree is updated:

To display a peak table for each sample you need to create a section.



3. Select the Section Layout Tool **Section**.

4. Click and drag the mouse over the Peak table frame:



The Section Properties dialog is displayed.

5. Select **Repeat section based on**, and then select **Task Samples** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

6. Click .

This is how the report will appear when printed:

Sample ID	Sample5.Sample		
Description			
Threshold	0.001		
Range start (nm)	800		
Range end (nm)	300		
Find	Peaks		
Display	List by position		
	Position (nm)	Intensity (A)	Type
1	798.3	0.2448	Peak
2	796.02	0.245	Peak
3	793.96	0.2417	Peak
4	791.1	0.2393	Peak
5	789	0.2383	Peak
6	786.8	0.2352	Peak
7	783.99	0.2361	Peak
8	782.03	0.237	Peak
9	779.07	0.2337	Peak
10	777.06	0.2342	Peak
11	774.95	0.2398	Peak

Sample ID	Sample6.Sample		
Description			
Threshold	0.001		
Range start (nm)	800		
Range end (nm)	300		
Find	Peaks		
Display	List by position		
	Position (nm)	Intensity (A)	Type
1	776.25	0.03531	Peak
2	571.56	0.03371	Peak

NOTE: Peak Tables cannot wrap over pages. If you have created the object the size of a page and not all the information is displayed, we recommend that you adjust the settings of the Peak Table within the Processing page of the Task (for example, increasing the Threshold).

Table Columns

How do I create a table that contains all the information displayed in the Sample Table and Results Table (excluding mandatory columns)?

You can also create a table (known as an expanding table) using the Table columns data objects. This will display all columns from the sample table and results table excluding mandatory columns (such as Type).

The example below shows how to create an expanding table. The table will be created with 2 columns. The first is for the sample ID. The second will expand so that all columns in the both the Sample and Results Table are reported.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:

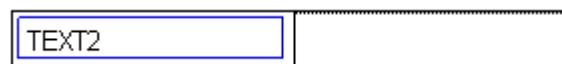


Table1

and the text object is added below table on the tree:

Text2

7. Click inside the blue box in the table and edit the text as required.

In the example below '**SampleID**' has been entered.

Sample ID	

8. Select the **Title** data object beneath the Table Columns (within Task Samples in the Task List).
9. Click in the top cell of the second column:

Sample ID	Title (Data3)

10. Select the ID or Full ID data object beneath Task Samples.

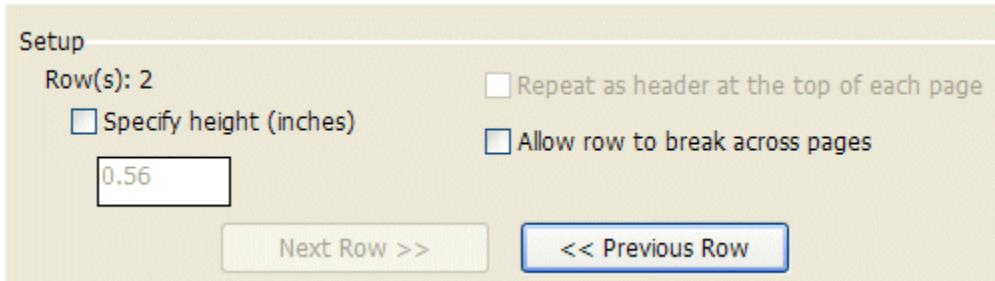
11. Click in the field beneath the Sample ID:

Sample ID	Title (Data3)
ID (Data1)	

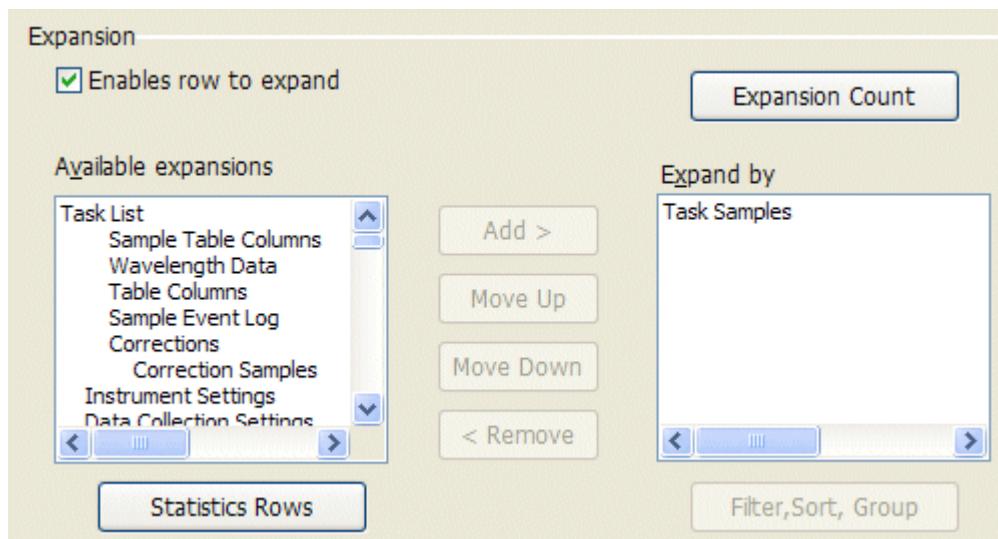
The data object to be used in the final cell of the table depends on the data in the Sample and Results Table. If there is text as well as numbers, you should use the Text data object. If there is only numerical data, use the Number data object.

12. Select the **Number** data object beneath the Table Columns
13. Click in the field beneath the Title data object.
14. Right-click on the table and select **Properties** from the menu.
15. Select the Rows tab.
16. Click Next row.

Row 2 should be specified at the top of the Setup:



17. Select **Enables row to expand**, and then select **Task Samples** from the drop-down list.



18. Select the Columns tab.
19. Click **Next Column** so that Column 2 is specified at the top of the setup.
20. Select Enables column to expand, and then select Table Columns.
21. Click **OK** to close the Properties dialog.

To view what will actually appear when the report is printed you need to print preview the report.

22. Click .

The report is displayed and the data objects are populated with the information.

NOTE: If the Sample ID contains a replicates or measurements extension, for example Sample1.replicate1, the extension is not reported in the table if ID is used. Use Full ID to include the extension.

Sample Event Log

How do I include a list of all sample events for all samples in my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.



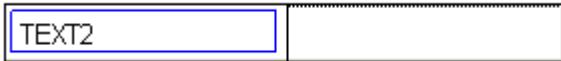
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Sample' has been entered.

Sample	

8. Repeat steps 5–7 to enter text for the Sample Event:

Sample	Event

9. Select the **Full ID** data object (from below Task Samples in the Task List).



The mouse pointer changes to .

10. Click in the field below 'Sample'.

The table now looks something like:

Sample	Event
Full ID (Data1)	

11. Repeat steps 9 and 10 for the **Sample Event Log Event** data object:

Sample	Event
Full ID (Data1)	Event (Data2)

To obtain all sample event log information for all samples, you need to repeat on the rows (to get all samples), and on the second column (to get all events for each sample).

12. Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

13. Select the Rows tab.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.

15. Ensure Enables row to expand is selected.

16. From the list of Available expansions, select **Task Samples**, and then click **Add**.

Task Samples moves from the Available expansions list to the Expand by list.

17. Select the Columns tab.

18. Click Next Column.

Column 2 is now specified at the top of the dialog.

19. Ensure Enables column to expand is selected.

20. From the list of Available expansions, select **Sample Event Log**, and then click **Add**.

21. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

22. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Sample	Event	Event
Sample7.Sample.Cycle1	Saved	Comment Added
Sample7.Sample.Cycle2	Saved	
Sample7.Sample.Cycle3	Saved	
Sample13.Sample.Cycle1	Saved	
Sample13.Sample.Cycle2	Saved	
Sample13.Sample.Cycle3	Saved	
Sample14.Sample.Cycle1	Saved	Excluded

NOTE: Where cycles are used, any comment added to a sample in the Sample Table only appears against the first cycle in the report table (as shown above).

How do I include the Sample Event Log information (event, time, reason/comment, name and ID) for each sample in my report template?

This example shows how to create a table of sample event log information. Each sample has a separate table.



1. Select the Table Layout tool .



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The columns will be set to expand for all the events recorded for a particular sample.



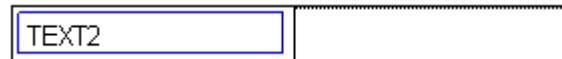
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Sample' has been entered.

Sample	

8. Repeat steps 5–7 to enter text for the other sample event log settings:

Sample	
Event	
Time	
Reason / Comment	
Name	
ID	

9. Click on the **Full ID** data object (Task Samples) in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Sample	Full ID (Data6)
Event	
Time	
Reason / Comment	
Name	
ID	

11. Repeat steps 9 and 10 for the other Sample Event Log data objects except **Time**.

12. In the Time field add a text block.

13. Delete the default text.

14. Select the **Time** data object and then click inside the text block.

The Time data object is placed inside the text block.

This enables the time to text wrap within the text block. Otherwise the cell would need to be very long to display all of the date.

Sample	Full ID (Data6)
Event	Event (Data1)
Time	[D5:Time]
Reason / Comment	Reason/Comment (Data2)
Name	Name (Data3)
ID	ID (Data4)

To include all sample event log information for the sample, the second column must be set to repeat on the Sample Event Log.

15. Select the table and right-click.
16. From the menu select **Properties**.
17. Select the columns tab.
18. Click Next Column.
Column 2 is displayed at the top of the dialog.
19. Ensure Enables column to expand is selected.
20. Select **Sample Event Log** from the drop-down list.
21. Click **OK**.

To display a table for every sample, a section must be created around the table.



22. Select the Section Layout tool



The mouse pointer changes to

23. Drag the mouse around the table.

A section is created around the table:

Sample	Full ID (Data6)
Event	Event (Data1)
Time	[D5:Time]
Reason / Comment	Reason/Comment (Data2)
Name	Name (Data3)
ID	ID (Data4)

and the Section Properties dialog is displayed.

24. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.
25. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

26. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Sample	Sample5.Sample	Sample5.Sample
Event	Comment Added	Comment Added
Time	04/29/2004 03:11:22 PM GMT Daylight Time	04/29/2004 03:11:40 PM GMT Daylight Time
Reason / Comment		first sample of new batch
Name	pksidhu	pksidhu
ID	pks1	pks1

Sample	Sample6.Sample
Event	Excluded
Time	04/29/2004 03:11:58 PM GMT Daylight Time
Reason / Comment	incorrect sample
Name	pksidhu
ID	pks1

In this example, there is an event log for sample 5 and an event log for sample 6. Sample 5 has 2 entries in the sample event log so the table has expanded to show both of these. For sample 6 there was only one entry.

Corrections

How do I include correction information for each sample in my report template?

In this example a table of sample name, correction name and corrections samples ID will be created for a sample. In addition a graph containing all the correction spectra for the sample will be created. All this information will be put within a section which will then be repeated on to get all the information for all samples in the task.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.

The correction samples ID columns will be set to expand for all correction IDs for a particular name. For example, the 100% and 0% baseline corrections (Corrections Name) has 2 Corrections IDs (spectra) associated with it – 100% or 0 Absorbance Baseline and 0% or Blocked Beam Baseline.



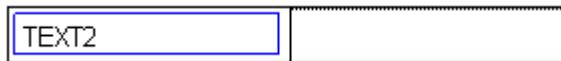
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.
In the example below '**Correction Name**' has been entered.

Correction Name		

8. Repeat steps 5–7 to enter text for the Correction ID and Full Sample ID:

Correction Name	Correction ID	Full Sample ID

9. Click on the Corrections **Name** in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Correction Name	Correction ID	Full Sample ID
Name (Data1)		

11. Repeat steps 9 and 10 for the Correction Samples **ID** and the **Full ID**.

The Full ID data object is further up the Task List beneath Task Samples.

The table now looks like:

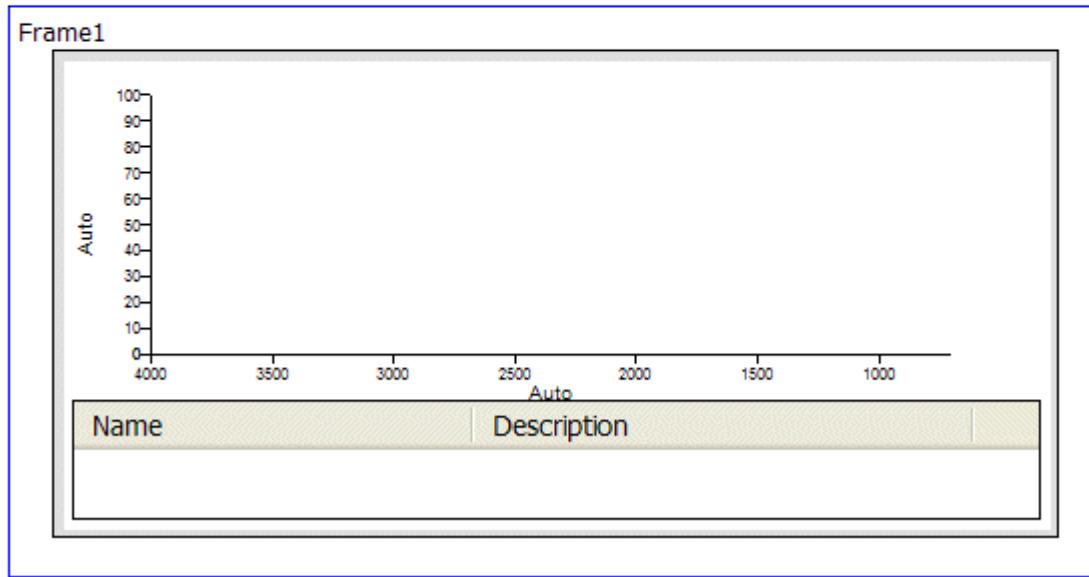
Correction Name	Correction ID	Full Sample ID
Name (Data1)	ID (Data2)	Full ID (Data3)

To show all the Correction IDs for a particular correction you need to expand column 2.

12. Select the Correction ID data object in the table (Data 2 in this example) and right-click.
13. From the menu select **Properties**.
14. Select the columns tab.

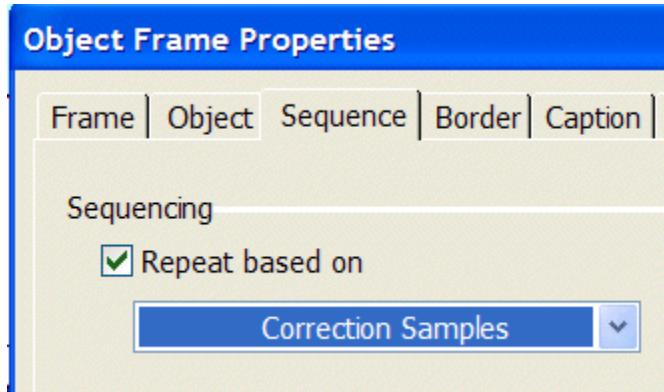
15. Click Next Column.
Column 2 is displayed at the top of the dialog.
16. Ensure Enables column to expand is selected.
17. Select **Correction Samples** from the drop-down list.
18. Click **OK**.
19. Select the Corrections Samples Raw Spectrum data object.
20. Click and drag to there required size.
The size of the object determines the size of the graph in the report.

Correction Name	Correction ID	Full Sample ID
Name (Data1)	ID (Data2)	Full ID (Data3)



To display all the correction spectra for a particular correction on one graph, you must repeat on the object frame based on correction samples.

21. Right-click on the object frame and select **Properties**.
The Object Frame Properties dialog is displayed.
22. Select the Sequence tab.
23. Ensure **Repeat based on** is selected, and then select **Correction Samples** from the drop-down list.



24. Click **OK**.

To display a table and graph for every sample, a section must be created around the table and graph.

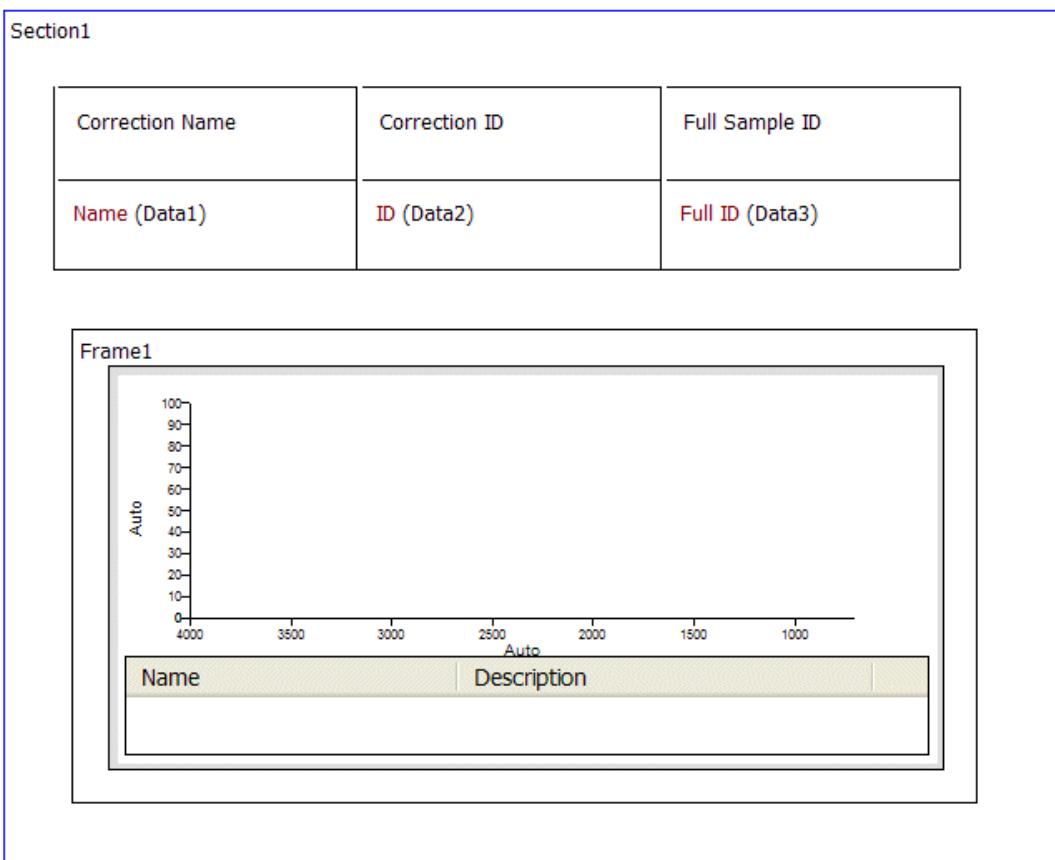


25. Select the Section Layout tool



The mouse pointer changes to

26. Drag the mouse around the table and graph.



and the Section Properties dialog is displayed.

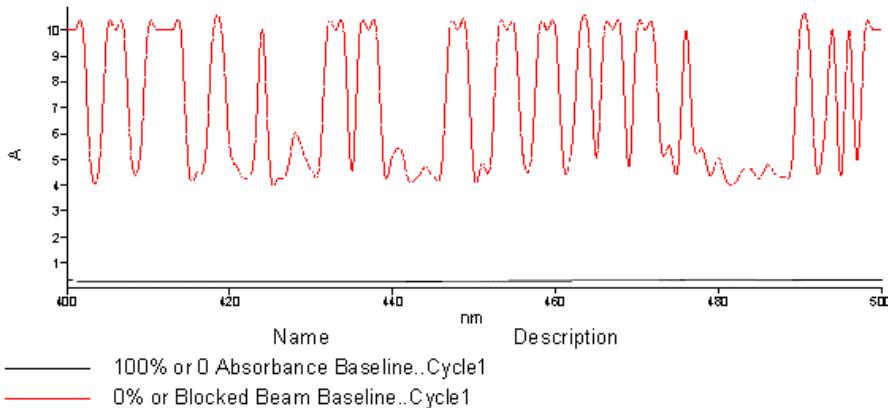
27. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.
28. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

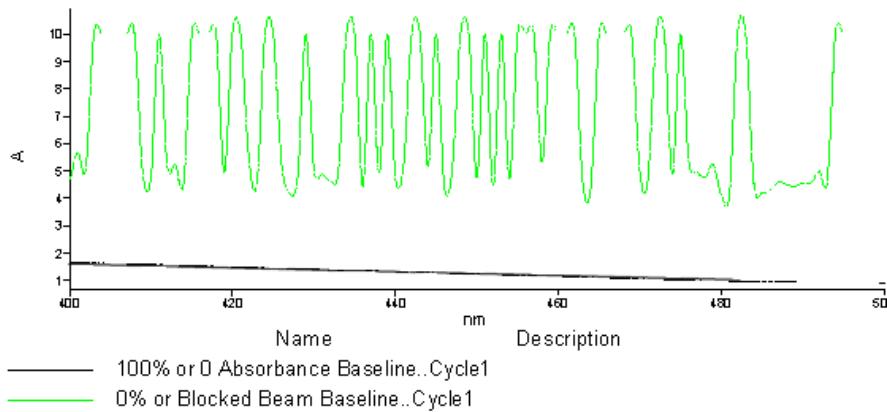
29. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Correction Name	Correction ID	Correction ID	Full Sample ID
100% and 0% baseline corrections	100% or 0 Absorbance Baseline	0% or Blocked Beam Baseline	Sample44.Sample



Correction Name	Correction ID	Correction ID	Full Sample ID
100% and 0% baseline corrections	100% or 0 Absorbance Baseline	0% or Blocked Beam Baseline	Sample45.Sample

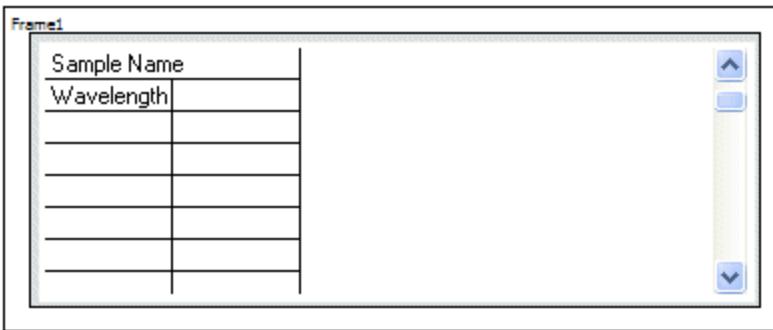


The table could also include a column for the Date Analyzed data object if required. Follow the steps above but create a table with an additional column. Ensure that the Correction ID column is still set to expand.

How do I include a wavelength table for all corrections for each sample in my report template?

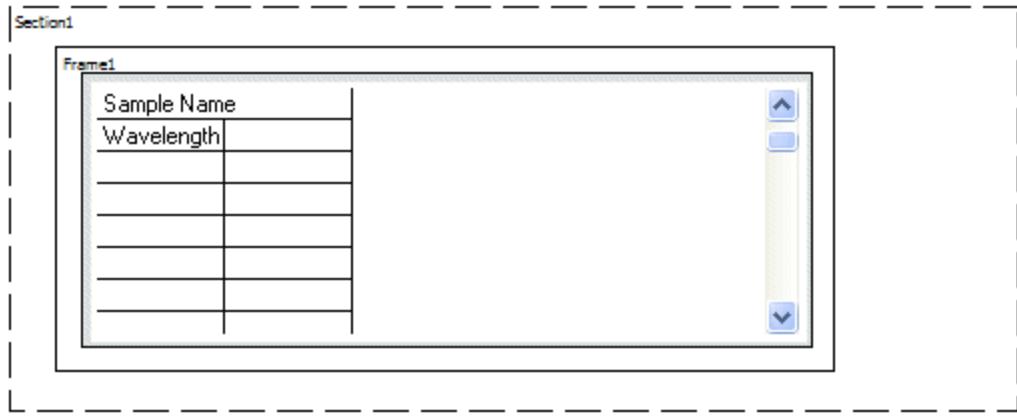
In this example the report template will include the name of the sample, the name of the correction and the wavelength table for the correction (for all samples).

1. Select the **Wavelength Table** data object (beneath Corrections) within the Task List.
2. Click the mouse on the report template and drag to create the object:



To display a table for all correction samples for a particular correction (for example 100% or 0 Absorbance Baseline and 0% or Blocked beam baseline for 100% and 0% baseline corrections), you need to create a section around the Wavelength Table.

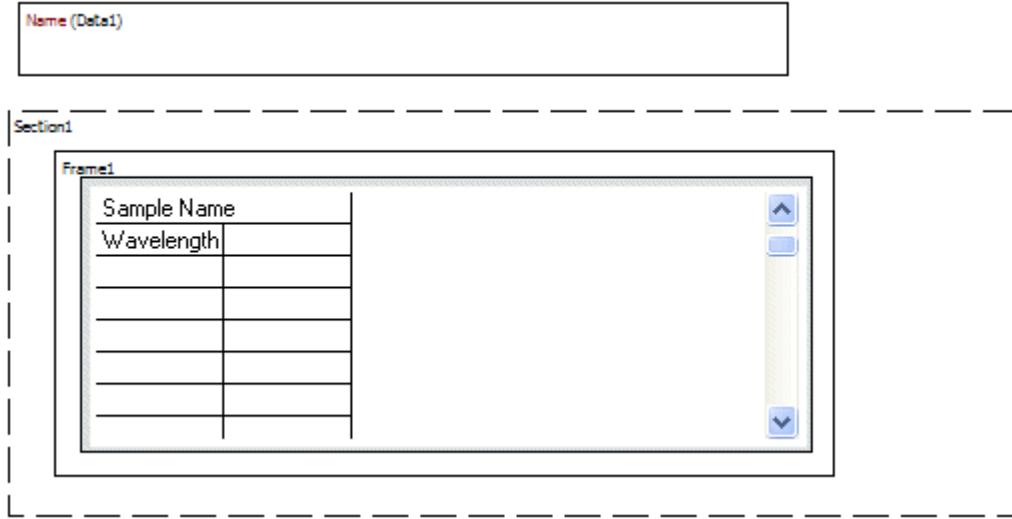
3. Select the Section layout tool.
4. Click and draw a section around the Wavelength Table:



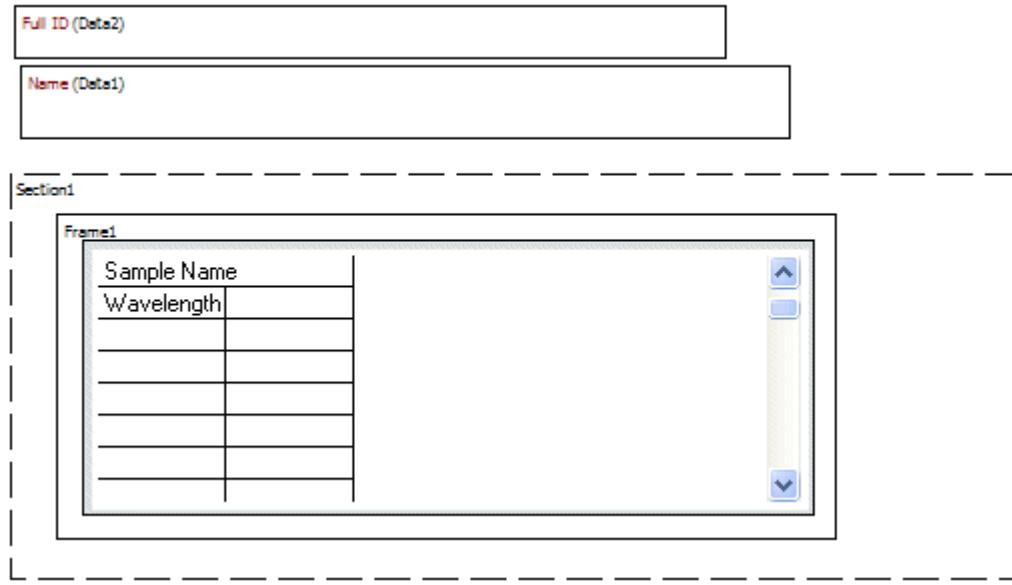
The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and select **Correction Samples** from the drop-down list.
6. Click **OK**.
7. Select the Corrections Name data object .

8. Place the object above the section.

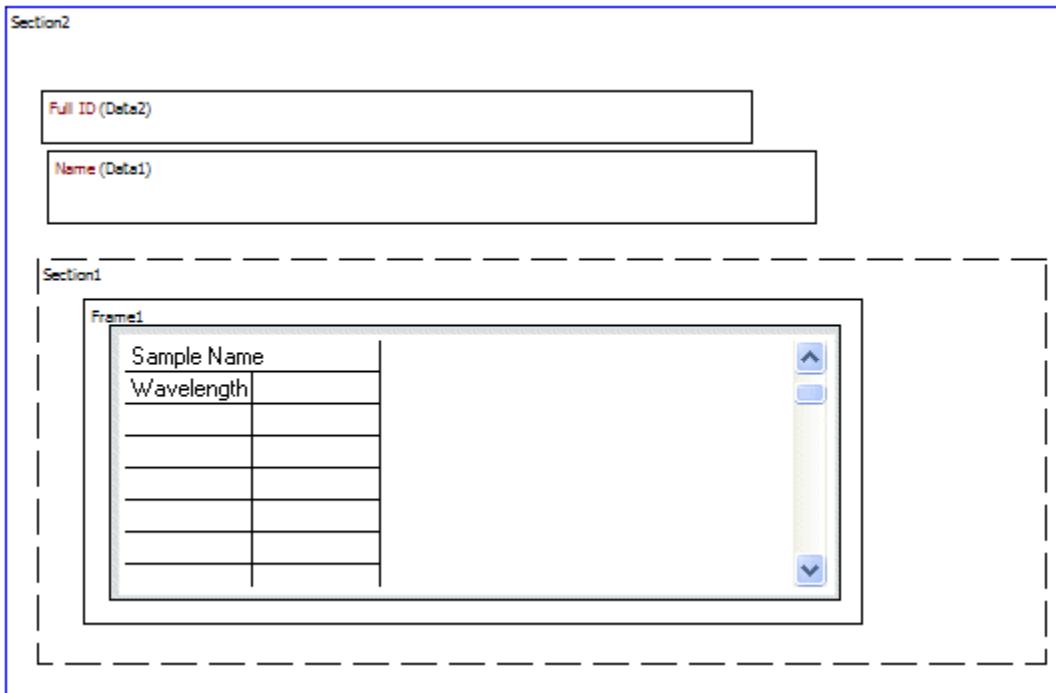


9. Select the **Full ID** data object from the Task List.
10. Place the object above the Corrections Name data object



To obtain the information for all samples, a section is needed around all this information.

11. Select the Section layout tool.
12. Click and draw a section around all the objects placed on the template in the above steps



The Section Properties dialog is displayed.

13. Ensure **Repeat section based on** is selected, and select **Task Samples** from the drop-down list.
14. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

15. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

PAGlassRM-19.Sample

100% and 0% baseline corrections

100% or 0 Absorbance Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	-0.0516375
546	-0.051752
635	-0.0549886
1700	-0.0143021
2300	-0.0175179

0% or Blocked Beam Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	10
546	10
635	10
1700	10
2300	2.53566

PAGlassRM-20.Sample

100% and 0% baseline corrections

100% or 0 Absorbance Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	-0.0516375
546	-0.051752
635	-0.0549886
1700	-0.0143021
2300	-0.0175179

0% or Blocked Beam Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	10
546	10
635	10
1700	10
2300	2.53566

Raw Points/Processed Points

How do I include a table of data points and the associated ordinate values in my report template?

The data objects allow you to display abscissa and ordinate value for raw and processed data. You can display all points, every 2 points, every 5 points, every 10 points, and/or every 100 points for raw and processed data.

Each of the objects has associated abscissa and ordinate objects.

These data objects can be used for all data collection types – scan, timedrive, polarization scan, and wavelength programming.

The example below shows how to create a table of every 5th raw data point and the associated ordinate value. For all other raw point / processed points, follow the steps below and substitute the required data objects.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:

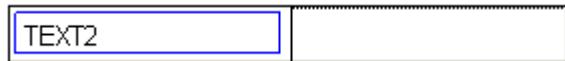


Table1



Text2

and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.
In the example below '**Abscissa (every 5 points)**' has been entered.

Abscissa (every 5 points)	

8. Repeat steps 5–7 to enter text for **Ordinate**:

Abscissa (every 5 points)	Ordinate

9. Click on the Raw Points every 5 Abscissa data object.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field below **Abscissa (every 5 points)**, and click.

The table now looks something like:

Abscissa (every 5 points)	Ordinate
Abscissa (Data1)	

11. Repeat steps 9 and 10 for the Ordinate data object.

Abscissa (every 5 points)	Ordinate
Abscissa (Data1)	Ordinate (Data2)

To display every 5th data point, row 2 must be set to expand.

12. Select the table and right-click.
13. From the menu select **Properties**.
14. Select the rows tab.
15. Click Next Row.
Row 2 is displayed at the top of the dialog.
16. Ensure Enables row to expand is selected.

17. Select **Raw Points every 5** from the drop-down list.

NOTE: The option selected from the drop-down list must correspond to the raw/processed abscissa/ordinate objects in the table. In this example Raw points every 5 Abscissa and Ordinate data objects have been used so **Raw Points every 5** must be selected for the expansion.

18. Click **OK**.

To display a table for every sample, a section must be created around the table.

NOTE: You may wish to include another column in your table for Sample ID so that you can identify which sample the data points relate to. (This is not shown here)



19. Select the Section Layout tool



The mouse pointer changes to

20. Drag the mouse around the table.

A section is created around the table:

Section1	
Abscissa (every 5 points)	Ordinate
Abscissa (Data1)	Ordinate (Data2)

and the Section Properties dialog is displayed.

21. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.
22. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

23. Click

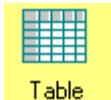
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Abscissa (every 5 points)	Ordinate
3300	0.638283140485891
3295	0.643445481285867
3290	0.632384085238443
3285	0.632167030853172
3280	0.631976126218767
3275	0.627198864356991
3270	0.62439913507893
3265	0.621620903689229
3260	0.620542520296119
3255	0.618989191884605
3250	0.616728690737875
3245	0.612557868365017
3240	0.611219488747158
3235	0.608352278702349

NOTE: You can specify the number of significant figures or decimal places by setting the properties of the data object. Close the print preview window if this is still open. Right-click on the data object and select **Properties**. The Numeric Data Object Properties dialog is displayed. You can select **Significant figures** or **Decimal places** and specify the number of Significant figures or Decimal places from the appropriate drop-down list.

How do I obtain the ordinate value at a specific abscissa value for all samples and include this in my report?

Rather than obtain all raw or processed points or every 2, 5, 10 or 100 points (see "How do I include a table of data points and the associated ordinate values in my report template ?" above), you can display the ordinate at a single specified value. This example shows how to create a table to report the ordinate value at 445 nm for all samples.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The example below has 2 columns and 2 rows.



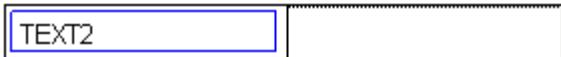
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Sample**' has been entered.

Sample	

8. Repeat steps 5–7 to enter text for **Ordinate value at 445 nm**:

Sample	Ordinate value at 445 nm

9. Select the **Full ID** data object and click in the second row of the first column of the table.

Sample	Ordinate value at 445 nm
Full ID (Data1)	

10. Click on the **Raw Points Ordinate** data object.



The mouse pointer changes to .

11. Position the mouse pointer in the empty field and click.

The table now looks something like:

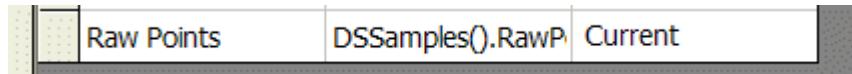
Sample	Ordinate value at 445 nm
Full ID (Data1)	Ordinate (Data3)

To display the Ordinate value at 445 nm you need to use Indexing on the Ordinate data object.

12. Right-click on the Ordinate data object in the table and select **Indexing**.

The Data Object Indexing dialog is displayed.

You need the Raw Points row in the dialog:



The **Index** is **Current** by default.

13. Click in the Raw Points Index field.

The entry turns blue and a drop-down arrow is displayed.

14. Enter the Abscissa value whose Ordinate value you wish to display (445 in this example). The value MUST be in quote marks:

 Raw Points	DSSamples().RawP "445"
--	------------------------

15. Click **OK**.

To display the ordinate value at the specified abscissa value for each sample, you need to repeat the second row based on task samples.

16. Right-click on the table and select **Properties**.

The Table Properties dialog is displayed.

17. Select the Rows tab.

18. Click Next Row.

Row 2 is displayed at the top of the dialog.

19. Ensure Enables row to expand is selected.

20. Select **Task Samples** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

21. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Sample	Ordinate value at 445 nm
Sample1.Sample	1.72816264808361
Sample2.Sample	1.7203010528572

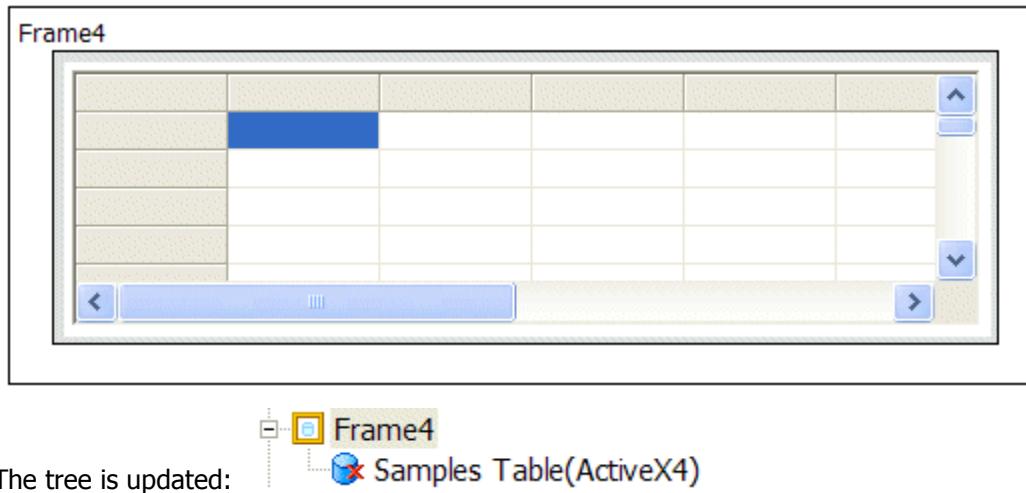
NOTE: To display the ordinate value of a processed data point, follow the procedure described above, but use the **Processed Points Ordinate** data object instead.

NOTE: You can specify the number of significant figures or decimal places by setting the properties of the data object. Close the print preview window if this is still open. Right-click on the data object and select **Properties**. The Numeric Data Object Properties dialog is displayed. You can select **Significant figures** or **Decimal places** and specify the number of Significant figures or Decimal places from the appropriate drop-down list.

Task List – Table Objects

How do I add the Samples Table to my report template?

1. Select the **Samples Table** data object within the Task List.
2. Click the mouse on the report template and drag to create the object:



To view what will actually appear when the report is printed you need to print preview the report.

3. Click .

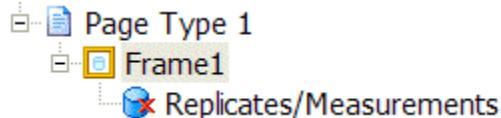
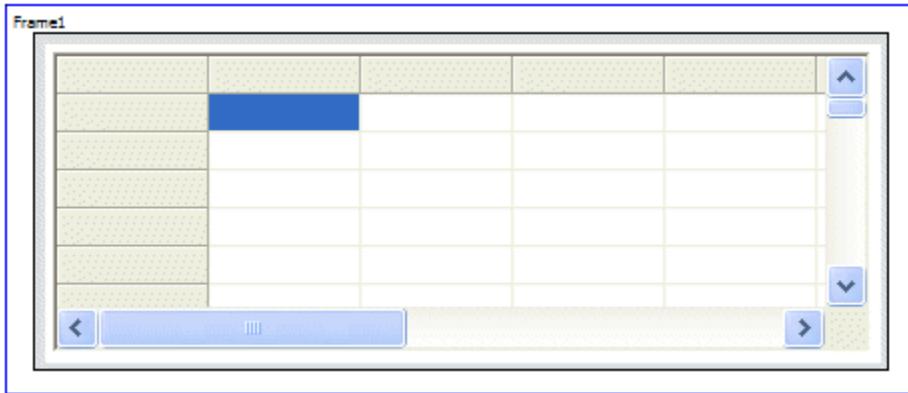
The report is displayed and the sample table are populated with the information. This is how the report will appear when printed:

	Sample ID	Description	Type
	Sample2		Replicate1
	Sample3		Replicate1
	Sample4		Replicate1
	Sample2		Replicate2
	Sample3		Replicate2
	Sample4		Replicate2

How do I add the Replicates/Measurements Table to my report template?

You can only have Replicates or Measurements in a method (not both at the same time). The table applicable to your data will be displayed when this data object is selected.

1. Select the **Replicates/Measurements Table** data object within the Task List.
2. Click the mouse on the report template and drag to create the object:



The tree is updated:

To view what will actually appear when the report is printed you need to print preview the report.

3. Click .

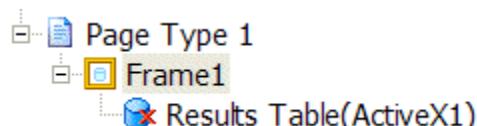
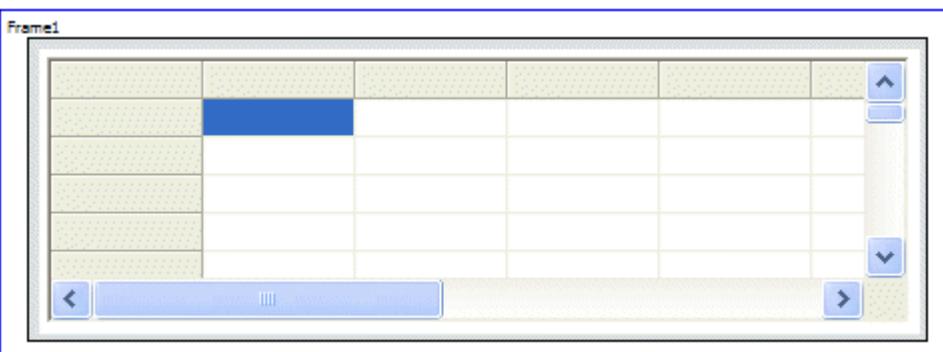
The report is displayed and the sample table are populated with the information. This is how the report will appear when printed:

Sample ID	Description
Sample12.Mean	Replicate mean of sample Sample12
Sample13.Mean	Replicate mean of sample Sample13
Sample14.Mean	Replicate mean of sample Sample14

How do I add the Results Table to my report template?

This is the information displayed on the Results tab (within Processing).

1. Select the **Results Table** data object within the Task List.
2. Click the mouse on the report template and drag to create the object:



The tree is updated:

To view what will actually appear when the report is printed you need to print preview the report.

3. Click .

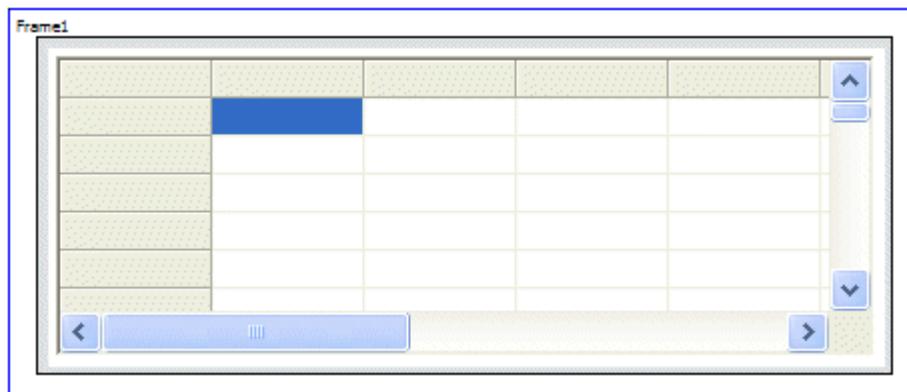
The report is displayed and the sample table are populated with the information. This is how the report will appear when printed:

	Sample ID	Concentration	Analyte (mgml-1)	Residual	Ordinate (A)
	standard06.Sample	1,5000	1,5659	-0,0659	0,6391
	standard12.Sample	2,5000	2,4999	0,0001	1,2787
	standard18.Sample	3,5000	3,4334	0,0666	1,9179
	standard24.Sample	4,5000	4,3657	0,1343	2,5563
	standard30.Sample	5,5000	5,2993	0,2007	3,1956

How do I add the Data Table to my report template?

This is the information displayed on the data tab within Processing and Results.

1. Select the Data Table data object within the Task List.
2. Click the mouse on the report template and drag to create the object:



To view what will actually appear when the report is printed you need to print preview the report.

3. Click .

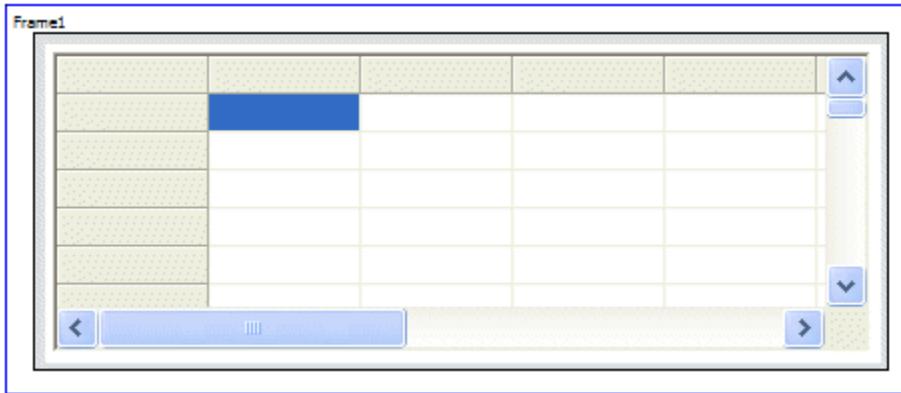
The report is displayed and the sample table are populated with the information. This is how the report will appear when printed:

	400 nm	500 nm
Sample12.Replicate1 (A)	0.18130	0.41092
Sample13.Replicate1 (A)	0.13688	0.25873
Sample14.Replicate1 (A)	0.08985	0.11177
Sample12.Replicate2 (A)	0.08964	0.11188
Sample13.Replicate2 (A)	0.08977	0.11169
Sample14.Replicate2 (A)	0.08962	0.11182

How do I add the Custom Table to my report template?

A custom table is created when one equation (defined in processing) is used to create another equation.

1. Select the **Custom Table** data object within the Task List.
2. Click the mouse on the report template and drag to create the object:



To view what will actually appear when the report is printed you need to print preview the report.

3. Click .

The report is displayed and the sample table are populated with the information. This is how the report will appear when printed:

Mean (nm)	656.090
Measured SD (nm)	0.000431
Repeatability Limit (nm)	0.02
Test Result	PASS

NOTE: The Custom Table data object is an ActiveX object. This means that the size of the frame drawn on the report template determines the size of the table (and hence the amount of information) seen in the report.

Task List – Method

This section covers the Method used to create the task.

How do I include the following information about the method used to create the task – Name, Revision, Type, Description, Status, Method ID, Date Created, Date Modified, and SOP document?

The instructions below describe how to put this information in a table as the individual values such as 1 for **Method Revision Number** would be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.



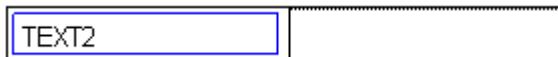
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Method Name**' has been entered.

Method Name	

8. Repeat steps 5–7 to enter text for the other output settings.

Method Name	
Method Revision number	
Method Type	
Method Description	
Method Status	
Method ID	
Date Method created	
Date Method modified	
SOP document attached	

9. Click on the required data object in the Data Object list to select it.



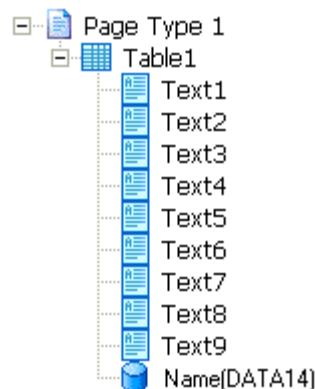
The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Method Name	Name (DATA14)
Method Revision number	
Method Type	
Method Description	
Method Status	
Method ID	
Date Method created	
Date Method modified	
SOP document attached	

and the data object is added to the tree. For example:



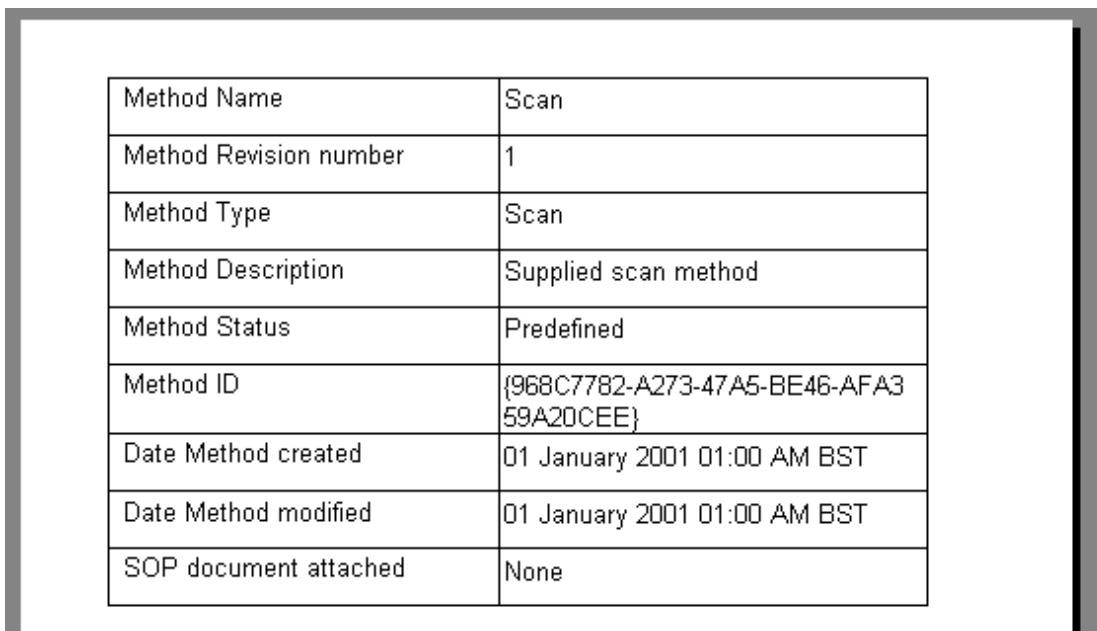
- Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Method Name	Name (DATA14)
Method Revision number	Revision (DATA15)
Method Type	Type (DATA16)
Method Description	Description (DATA17)
Method Status	Status (DATA18)
Method ID	Method ID (DATA19)
Date Method created	Date Created (DATA20)
Date Method modified	Date Modified (DATA21)
SOP document attached	SOP document (DATA22)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



A screenshot of a printed report showing a table of method details. The table has 9 rows and 2 columns. The columns are 'Method Name' and 'Scan'. The rows contain the following data:

Method Name	Scan
Method Revision number	1
Method Type	Scan
Method Description	Supplied scan method
Method Status	Predefined
Method ID	{968C7782-A273-47A5-BE46-AFA359A20CEE}
Date Method created	01 January 2001 01:00 AM BST
Date Method modified	01 January 2001 01:00 AM BST
SOP document attached	None

How do I include the name and ID of the person who created the method that was used to create the task?

The Created by sub-branch of Method (within the Task list) contains the data objects for the Name and ID of the person who created the method that was then used to create the task.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool .



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.

TEXT2	
-------	--

and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Method created by (Name)**' has been entered.

Method created by (Name)	

8. Repeat steps 5–7 to enter text for the ID:

Method created by (Name)	
Method created by (ID)	

9. Click on the required data object in the Data Object list to select it:



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Method created by (Name)	Name (DATA14)
Method created by (ID)	

and the data object is added to the tree.

11. Repeat steps 8 and 9 for the number of copies data object:

Method created by (Name)	Name (DATA14)
Method created by (ID)	ID (DATA15)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Method created by (Name)	PerkinElmer Method
Method created by (ID)	PEDeveloper

How do I include the name and ID of the person who modified the method that was used to create the task?

The Modified by sub-branch of Method (within the Task list) contains the data objects for the Name and ID of the last person who modified the method that was then used to create the task.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool  .



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



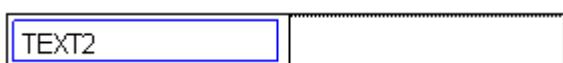
5. Select the Text Block layout tool  .



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.
In the example below '**Method modified by (Name)**' has been entered.

Method modified by (Name)	

8. Repeat steps 5 to 7 to enter text for the ID:

Method modified by (Name)	
Method modified by (ID)	

9. Click on the Modified by Name data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Method modified by (Name)	Name (Data1)
Method modified by (ID)	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the number of copies data object:

Method modified by (Name)	Name (Data1)
Method modified by (ID)	ID (Data2)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Method modified by (Name)	PerkinElmer Method
Method modified by (ID)	PEDeveloper

Task List – Created By/Modified By

How do I include the name and ID of the person who created the task?

The Created by sub-branch within the Task list contains the data objects for the Name and ID of the person who created the task.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



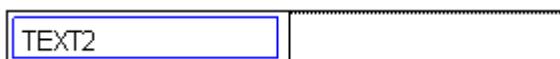
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Task created by (Name)**' has been entered.

Task created by (Name)	

8. Repeat steps 5– 7 to enter text for the ID:

Task created by (Name)	
Task created by (ID)	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Task created by (Name)	Name (Data1)
Task created by (ID)	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the number of copies data object.

Task created by (Name)	Name (Data1)
Task created by (ID)	ID (Data2)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

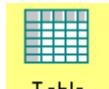
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Task created by (Name)	tracey
Task created by (ID)	PEDeveloper

How do I include the name and ID of the person who modified the task?

The Modified by sub-branch within the Task list contains the data objects for the Name and ID of the last person who modified the task.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



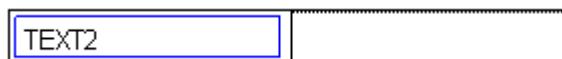
5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Task modified by (Name)**' has been entered.

Task modified by (Name)	

8. Repeat steps 5– 7 to enter text for the ID:

Task modified by (Name)	
Task modified by (ID)	

9. Click on the **Modified by Name** data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Task modified by (Name)	Name (Data1)
Task modified by (ID)	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the number of copies data object.

Task modified by (Name)	Name (Data1)
Task modified by (ID)	ID (Data2)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Task modified by (Name)	tracey
Task modified by (ID)	PEDeveloper

Task List – Instrument

How do I add the name, type, serial number, IPV status and the date the instrument information was last modified to my template?

The instructions below describe how to put this information in a table as the individual values may be meaningless on the page.

Information for all instruments can be displayed.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Format**.

The format table dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example has 2 columns and 5 rows.



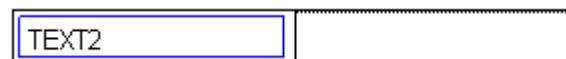
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Instrument Name**' has been entered.

Instrument name	

- Repeat steps 5–7 to enter text for the other settings.

Instrument name	
Instrument type	
Serial number	
IPV status	
Last modified	

- Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Instrument name	Name (DATA31)
Instrument type	
Serial number	
IPV status	
Last modified	

and the data object is added to the tree.

- Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Instrument name	Name (DATA31)
Instrument type	Type (DATA32)
Serial number	Serial No. (DATA33)
IPV status	IPV Status (DATA34)
Last modified	Last Modified (DATA35)

If you have more than one instrument installed you will need to create a section that will then repeat for all instruments.



- Within the Layout Tools click .



The mouse pointer changes to .

13. Drag the mouse around the table.

SECTION2	
Instrument name	Name (DATA31)
Instrument type	Type (DATA32)
Serial number	Serial No. (DATA33)
IPV status	IPV Status (DATA34)
Last modified	Last Modified (DATA35)

The Format Section dialog is displayed.

14. Select **Repeat section based on**, and then select **Instrument List** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

15. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Instrument name	Sim45
Instrument type	Lambda 45
Serial number	Simulation
IPV status	None
Last modified	16 June 2003 09:23 AM BST

Instrument name	Lambda 800
Instrument type	Lambda 800
Serial number	Simulation
IPV status	None
Last modified	11 September 2003 09:19 AM BST

NOTE: If you have more than one instrument attached, you could create a section around the table and repeat by Instrument List to obtain the information for all instruments.

NOTE: The IPV data objects are listed but they are only ever populated in templates that are associated with IPV tests. It is not possible to change the default templates that are associated with the IPV tests. It is possible to edit the default templates but WE STRONGLY RECOMMEND THAT YOU DO NOT DO THIS. Therefore, these data objects will not be discussed in the Help.

Task List – Quant Calibration

Calibration Graphs

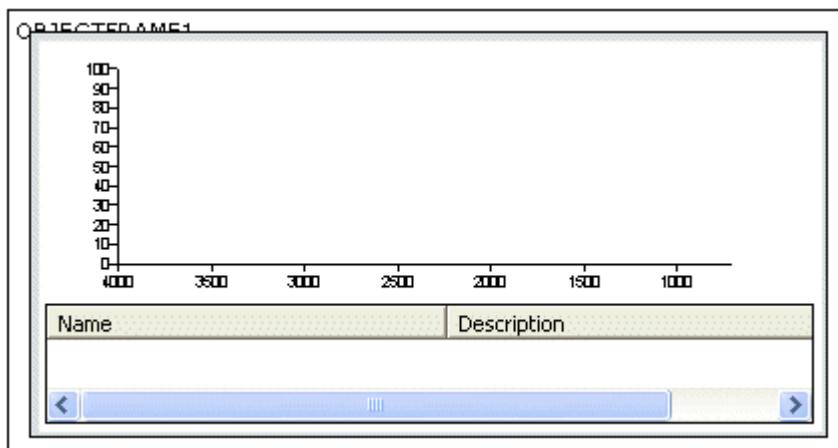
How do I display the Calibration Graph?

1. Select the **Calibration Curve** data object.

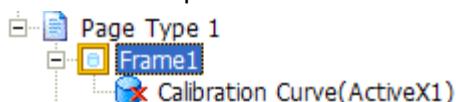


2. Drag the mouse on the report template.

An object frame with an empty graph is displayed

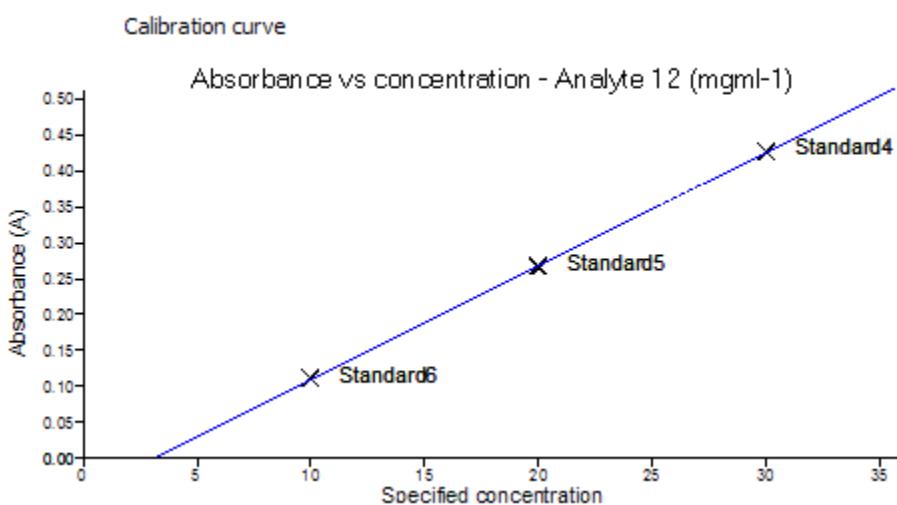


and the tree is updated:



To view what will actually appear when the report is printed you need to print preview the report.

3. Click .



How do I display the Residuals Plot?

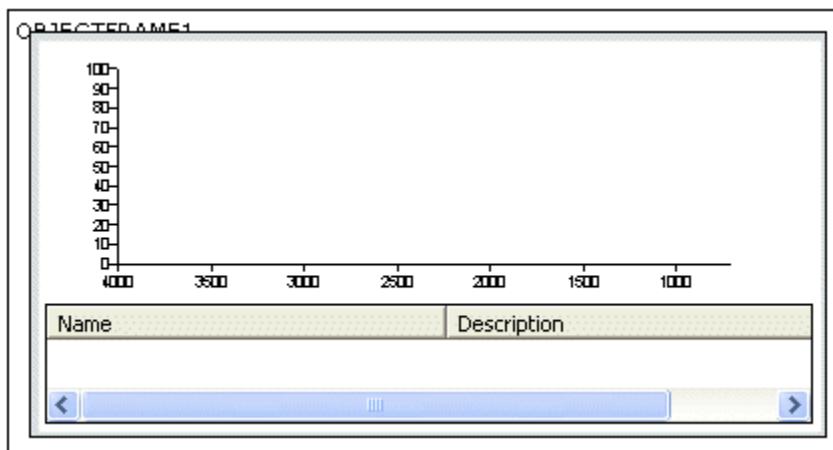
1. Select the **Residuals Plot** data object.



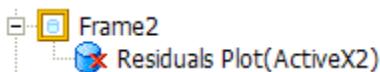
The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:

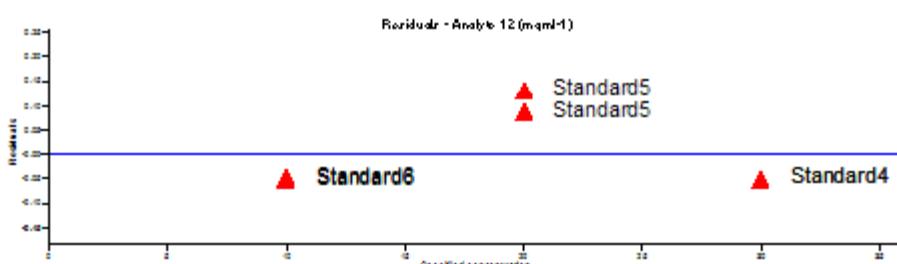


and the tree is updated:



To view what will actually appear when the report is printed you need to print preview the report.

3. Click



How do I display the Specified vs Actual Plot?

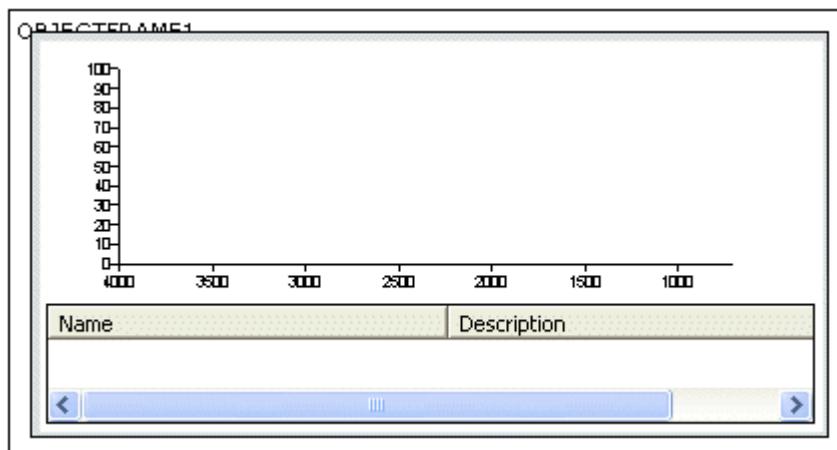
1. Select the **Specified vs. Actual Plot** data object.



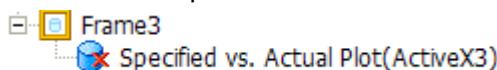
The mouse pointer changes to

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed.

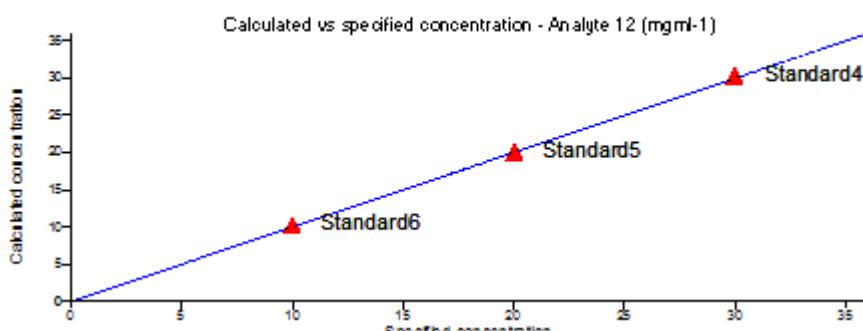


and the tree is updated:



To view what will actually appear when the report is printed you need to print preview the report.

3. Click



Calibration Details

How do I include a table of the Calibration coefficients and Correlation coefficient in my report template?



1. Select the Table Layout tool:



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 5 columns and 2 rows.



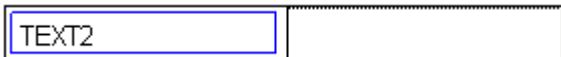
5. Select the Text Block layout tool:



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Coefficient x0**' has been entered.

Coefficient x0				

8. Repeat steps 5–7 to enter text for the other coefficients:

Coefficient x0	Coefficient x1	Coefficient x2	Coefficient x3	Correlation Coefficient

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like

Coefficient x0	Coefficient x1	Coefficient x2	Coefficient x3	Correlation Coefficient
Coefficient x0 (Data1)				

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Coefficient x0	Coefficient x1	Coefficient x2	Coefficient x3	Correlation Coefficient
Coefficient x0 (Data1)	Coefficient x1 (Data2)	Coefficient x2 (Data3)	Coefficient x3 (Data4)	Correlation Coefficient (Data5)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Coefficient x0	Coefficient x1	Coefficient x2	Coefficient x3	Correlation Coefficient
-0.04811	0.01581	0	0	1.000

How do I include the analyst name and ID from the person who performed the calibration and the date the calibration was performed in my report template?



1. Select the Table Layout tool:



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 3 rows.



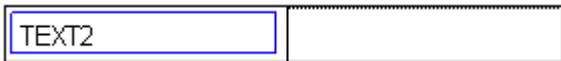
5. Select the Text Block layout tool:



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Calibrated by (Name)**' has been entered.

Calibrated by (Name)	

8. Repeat steps 5–7 to enter text for Analyst ID and Calibrated on:

Calibrated by (Name)	
Calibrated by (ID)	
Calibrated on	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Calibrated by (Name)	Name (Data1)
Calibrated by (ID)	
Calibrated on	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the analyst ID data object:

Calibrated by (Name)	Name (Data1)
Calibrated by (ID)	ID (Data2)
Calibrated on	

When using a date data object, we recommend that the data object is placed inside a text block. This ensures that all the date is displayed. If a text block is not used, and the field is not large enough to display the date, you will see ##### in the field.



12. Select the Text Block from the layout tools -



The mouse pointer changes to

13. Drag the mouse on the report template to create a Text Block:



14. Click inside the Text Block and remove the default text.

15. Select the Calibrated on data object.

16. Click inside the Text Block.

The object is placed inside the Text Block.

Calibrated by (Name)	Name (Data1)
Calibrated by (ID)	ID (Data2)
Calibrated on	[D3:Calibrated on]

To view what will actually appear when the report is printed you need to print preview the report.

17. Click .

Calibrated by (Name)	tracey
Calibrated by (ID)	tracey
Calibrated on	04/26/2004 11:24:54 AM BST

Standards Table

How do I add the Standards table from my Quant task to my report template?

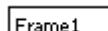
1. Select the **Standards Table** from the list of Data Objects..



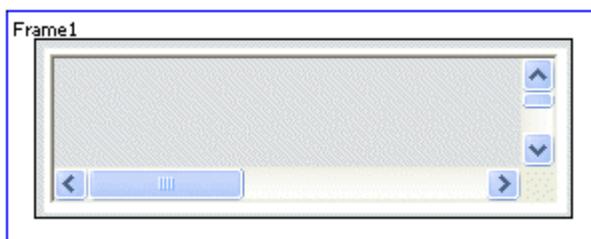
The mouse pointer changes to .

2. Click the mouse on the page.

A frame is added to the page:

 Frame1

3. Drag the frame to enlarge it:



If the frame is too small, when you select Print Preview an error message will be displayed and the Standards Table will not be displayed correctly.

The frame object is added to the tree:



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the table is populated:

	Sample ID	Concentration
	Standard3	0.10
	Standard4	0.20
	Standard5	0.30

Standards

How do I include the ID, description, status and comments for each of the standards in my task in the report template?



1. Select the Table Layout tool: 



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 4 columns and 2 rows.



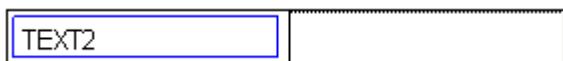
5. Select the Text Block layout tool: 



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.
In the example below 'ID' has been entered.

ID			

8. Repeat steps 5–7 to enter text for the other output settings:

ID	Description	Status	Comments

NOTE: The text headings should be at the top of each column (rather than all rows in the first column) as the table will be repeated on the second row to obtain the information for all samples in the task.

9. Click on the required data object in the Data Object list to select it.



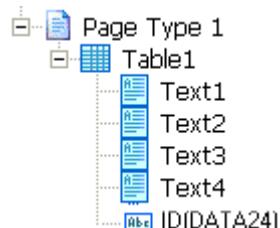
The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like

ID	Description	Status	Comments
ID (DATA24)			

and the data object is added to the tree:



11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

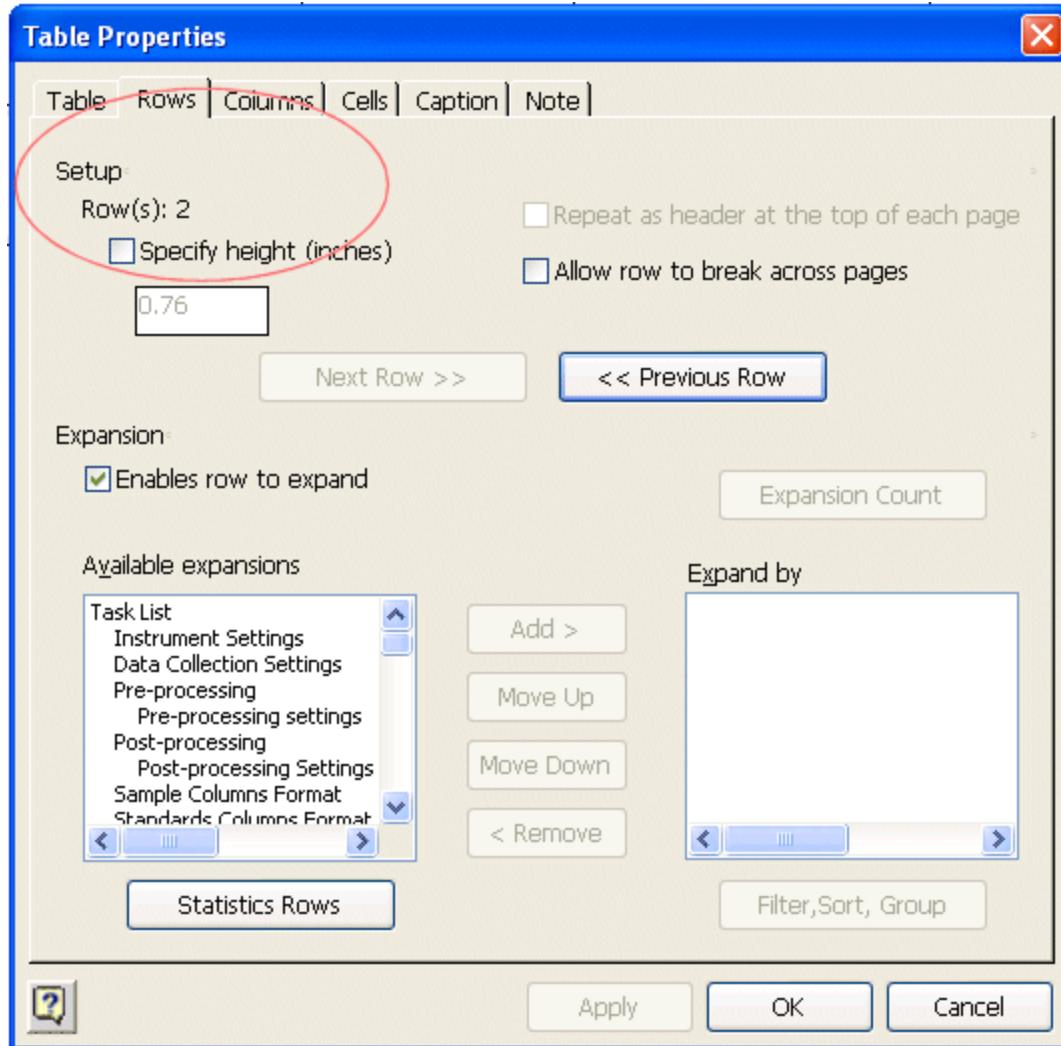
ID	Description	Status	Comments
ID (DATA24)	Description (DATA25)	Status (DATA26)	Comments (DATA27)

To obtain the ID, description, status and comments for all the samples in the task, you need to repeat the second row of the table.

12. Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

13. Select the Rows tab.
 14. Click Next Row.
- Row 2 is now specified at the top of the dialog.



15. Ensure Enables row to expand is selected.
16. From the list of Available expansions, select **Standards**, and then click **Add**. Samples moves from the Available expansions list to the Expand by list.
17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

ID	Description	Status	Comments
s10std1	Batch 10	Imported	
s10std2	Batch 10	Imported	
s10std3	Batch 10	Imported	
s10std4	Batch 10	Imported	

NOTE: In this example there are no comments associated with any of the standards and so the Comments fields are empty.

For information about the Full ID and Type data objects, see:

- What is the difference between ID and Full ID?
- What is the Type data object?

How do I include the analyst name and ID and the date the standards were analyzed in my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 3 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Analyst Name' has been entered.

Analyst Name	

8. Repeat steps 5–7 to enter text for Analyst ID and Date Analyzed:

Analyst Name	
Analyst ID	
Analysis Date	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Analyst Name	Name (Data1)
Analyst ID	
Analysis Date	

and the data object is added to the tree.

- Repeat steps 9 and 10 for the analyst ID data object:

Analyst Name	Name (Data1)
Analyst ID	ID (Data2)
Analysis Date	

When using a date data object, we recommend that the data object is placed inside a text block. This ensures that all the date is displayed. If a text block is not used, and the field is not large enough to display the date, you will see ##### in the field.



Text Block

- Select the Text Block from the layout tools -



The mouse pointer changes to

- Drag the mouse on the report template to create a Text Block.

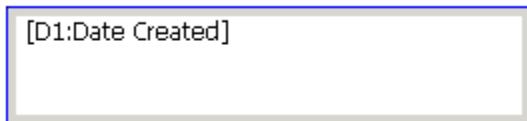


- Click inside the Text Block and remove the default text.

- Select the Date analyzed data object.

16. Click inside the Text Block.

The object is placed inside the Text Block.



Analyst Name	Name (Data1)
Analyst ID	ID (Data2)
Analysis Date	[D3:Date Analyzed]

To view what will actually appear when the report is printed you need to print preview the report.

17. Click .

Analyst Name	tracey
Analyst ID	tracey
Analysis Date	04/26/2004 04:23:53 PM BST

How do I display the raw spectrum of each sample on a separate graph?

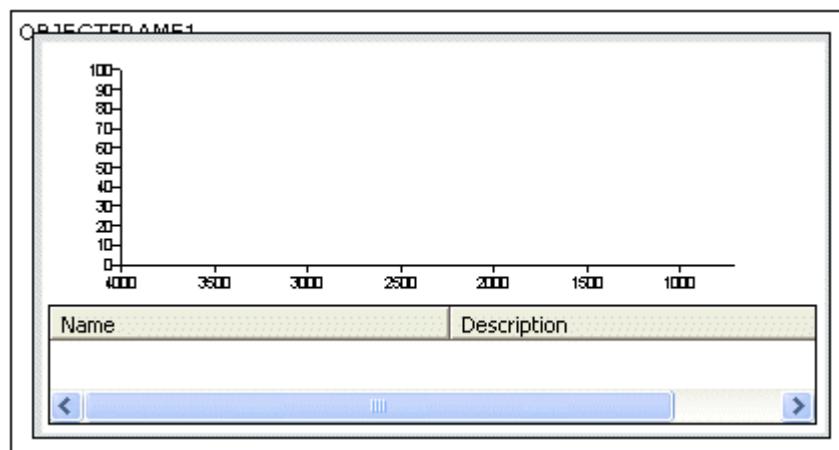
1. Select the **Raw Spectrum** data object.



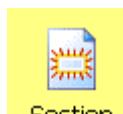
The mouse pointer changes to

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



To display the raw spectrum for each sample on a separate graph, a section must be created.

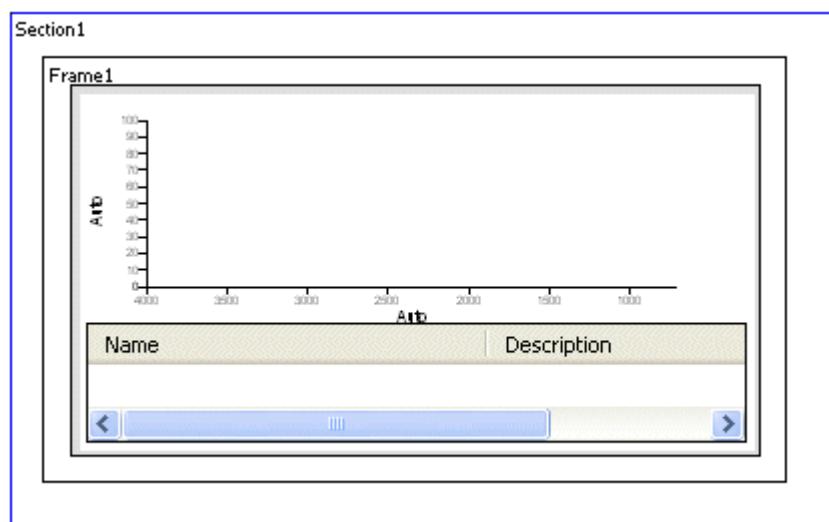


3. From the Layout Tools list, select



The mouse pointer changes to

4. Drag the mouse around the object frame to create a section containing the spectrum:



The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and then select **Standards** from the drop-down list.
6. Click **OK**.

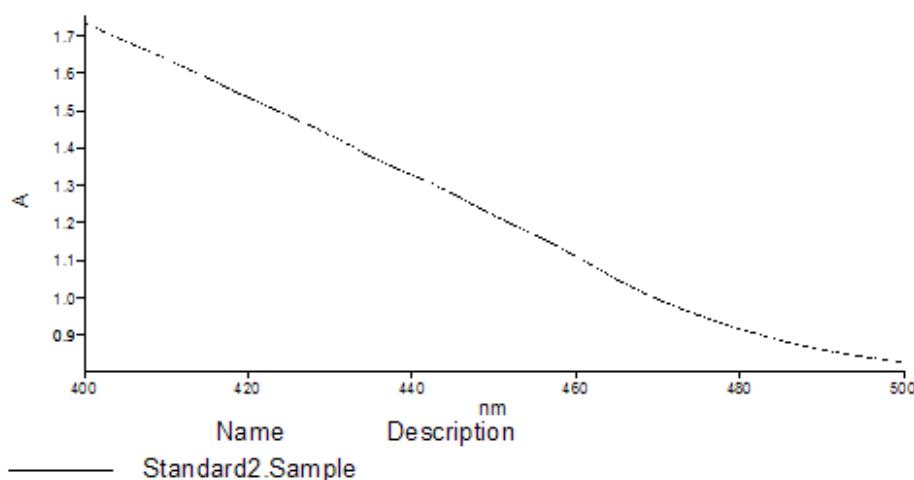
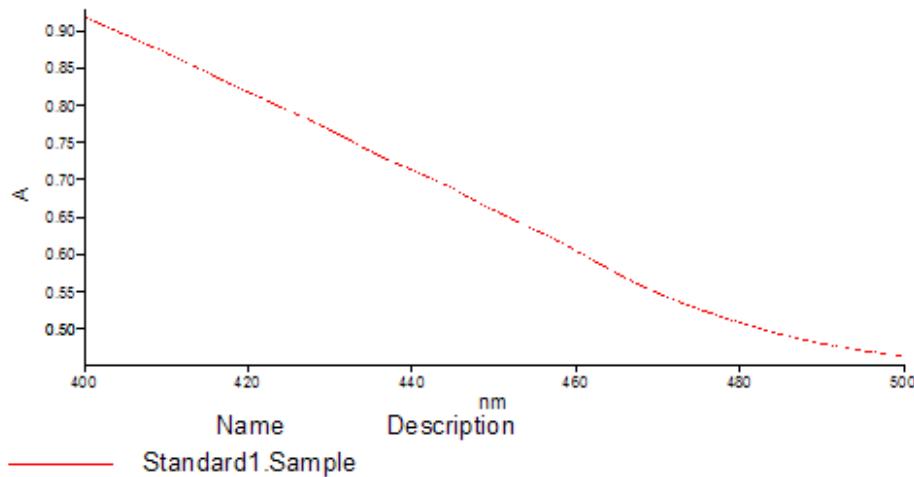


The section is added to the tree:

To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

An example is shown below:



How do I overlay the raw spectra of each standard on the same graph?

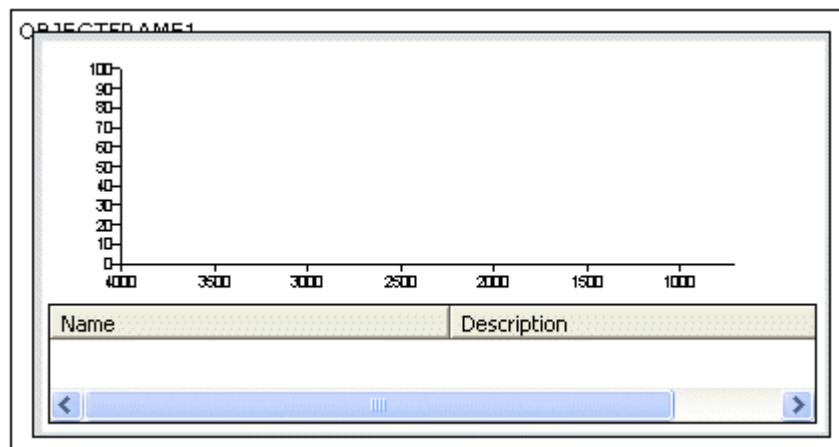
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



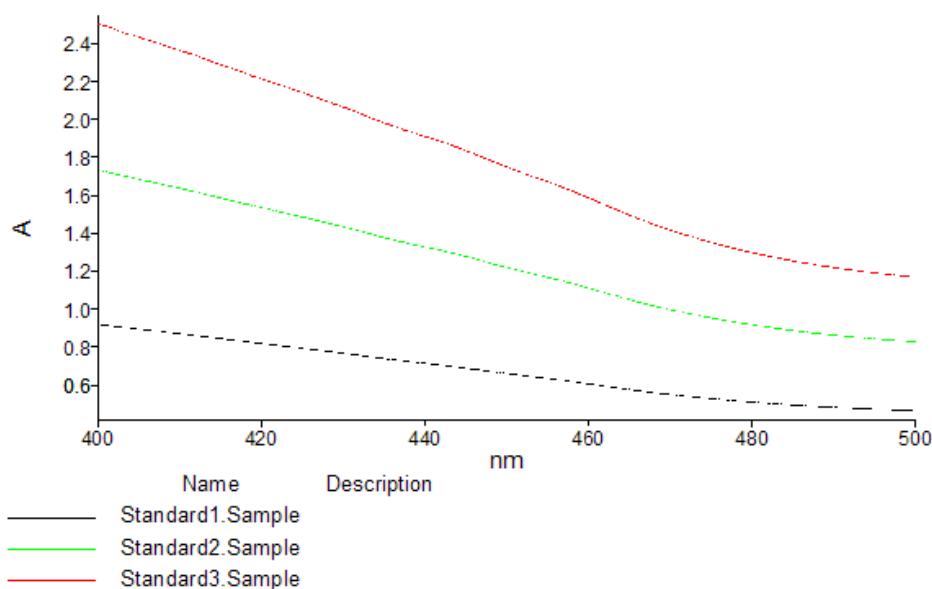
3. Right-click on the frame and select **Properties**.

4. Select the Sequence tab.

5. Select **Repeat based on**, and then select **Standards** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

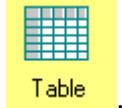
6. Click .



Task List – Task Event Log

How do I include the Task Event Log information in my report template?

Events such saving and reviewing are recorded in the Task Event Log. If these data objects were just placed on the page, only the most recent event would be displayed in the report. To display all events, a section must be created that repeats on the Task Event Log.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 6 rows.



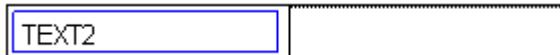
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.
In the example below 'Event' has been entered.

Event	

8. Repeat steps 5–7 to enter text for the other output settings:

Event	
Time	
Reason	
Comment	
Name	
ID	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Event	Event (Data1)
Time	
Reason	
Comment	
Name	
ID	

11. Repeat steps 9 and 10 for the other output settings data objects except **Time**.
12. In the Time field add a text block.
13. Delete the default text.
14. Select the Time data object and then click inside the text block.

The Time data object is placed inside the text block.

This enables the time to text wrap within the text block. Otherwise the cell would need to be very long to display all of the date.

Event	Event (Data1)
Time	[D7:Time]
Reason	Reason (Data3)
Comment	Comment (Data4)
Name	Name (Data5)
ID	ID (Data6)

To display all the event log entries a section must be created around the table.



15. Select the Section Layout tool



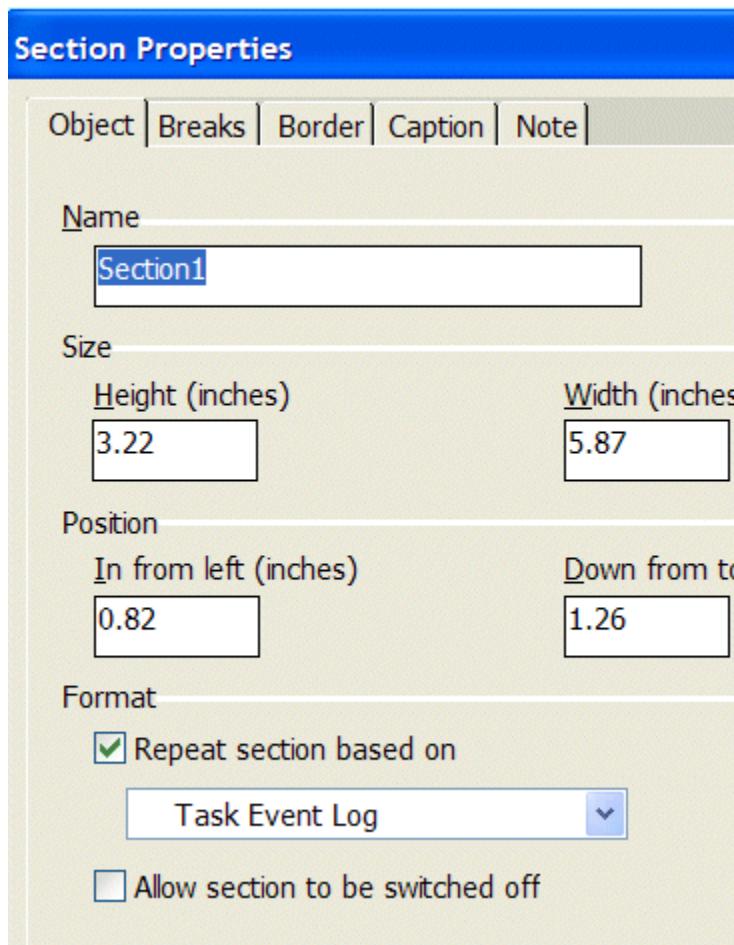
The mouse pointer changes to

16. Drag the mouse around the table.

A section is created around the table:

Section1	
Event	Event (Data1)
Time	[D7:Time]
Reason	Reason (Data3)
Comment	Comment (Data4)
Name	Name (Data5)
ID	ID (Data6)

and the Section Properties dialog is displayed.



17. Ensure **Repeat section based on** is selected, and then select **Task Event Log** from the drop-down list.
18. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

19. Click .

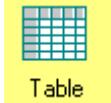
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Event	Saved
Time	04/06/2004 02:27:26 PM GMT
Reason	Saved data
Comment	
Name	tracey
ID	tracey

Event	Reviewed
Time	04/28/2004 09:32:50 AM GMT
Reason	Data reviewed
Comment	
Name	tracey
ID	tracey

Task List – Measurements or Replicates Table Samples

How do I include the Full ID, Description and Status for all measurements or replicates?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 3 columns and 2 rows.



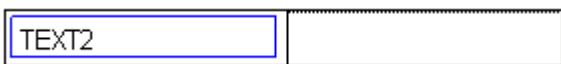
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'FullID' has been entered.

Full ID		

8. Repeat steps 5–7 to enter text for the other output settings:

Full ID	Description	Status

9. Click on the **Full** data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Full ID	Description	Status
Full ID (Data1)		

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Full ID	Description	Status
Full ID (Data1)	Description (Data2)	Status (Data3)

To obtain the Full ID, description and status for all the measurements in the task, you need to repeat the second row of the table.

12. Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

13. Select the Rows tab.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.

15. Ensure Enables row to expand is selected.

16. From the list of Available expansions, select **Measurements or Replicates Table Samples**, and then click **Add**.

Samples moves from the Available expansions list to the Expand by list.

17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Full ID	Description	Status
Sample1.Mean	Measurement mean of sample Sample1	Pending
Sample2.Mean	Measurement mean of sample Sample2	Pending

NOTE: The Description is not the description entered for each measurement in the sample table. Instead, the software enters the description to reflect that it is a mean of the measurements.

How do I include a table of data points and the associated ordinate values for measurements or replicates in my report template?

The data objects allow you to display abscissa and ordinate value for raw data associated with replicates or measurements. You can display all points, every 2 points, every 5 points, every 10 points and/or every 100 points for raw data. Each of the objects has associated abscissa and ordinate objects.

These data objects can be used for all data collection types – Scan, Timedrive, Polarization scan, and Wavelength Program.

The example below shows how to create a table of every 10th raw data point and the associated ordinate value. For all other raw points, follow the steps below and substitute the required data objects.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The example below has 2 columns and 2 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Abscissa (every 10 points)**' has been entered.

Abscissa (every 10 points)	

8. Repeat steps 5–7 to enter text for **Ordinate**:

Abscissa (every 10 points)	Ordinate

9. Click on the **Raw Points every 10 Abscissa** data object.



The mouse pointer changes to

10. Position the mouse pointer in the empty field below **Abscissa (every 10 points)**, and click.

The table now looks something like:

Abscissa (every 10 points)	Ordinate
Abscissa (Data1)	

11. Repeat steps 9 and 10 for the Ordinate data object.

Abscissa (every 10 points)	Ordinate
Abscissa (Data1)	Ordinate (Data2)

To display every 10th data point, row 2 must be set to expand.

12. Select the table and right-click.
13. From the menu select **Properties**.
14. Select the rows tab.
15. Click Next Row.
Row 2 is displayed at the top of the dialog.
16. Ensure Enables row to expand is selected.
17. Select **Raw Points every 10** (beneath Measurement or Replicates) from the drop-down list.

NOTE: The option selected from the drop-down list must correspond to the raw/processed abscissa/ordinate objects in the table. In this example Raw points every 10 Abscissa and Ordinate data objects have been used so **Raw Points every 10** must be selected for the expansion, and it must be below the Measurements or Replicates option. If the incorrect option is chosen, the table will not expand.

18. Click **OK**.

To determine which sample the data belongs to it is useful to include a text header.

19. Select the **Full ID** data object from the task list and position this above the table.



Abscissa (every 10 points)	Ordinate
Name (Data1)	ID (Data2)

To display a table for every sample, a section must be created around the table and Full ID data object.



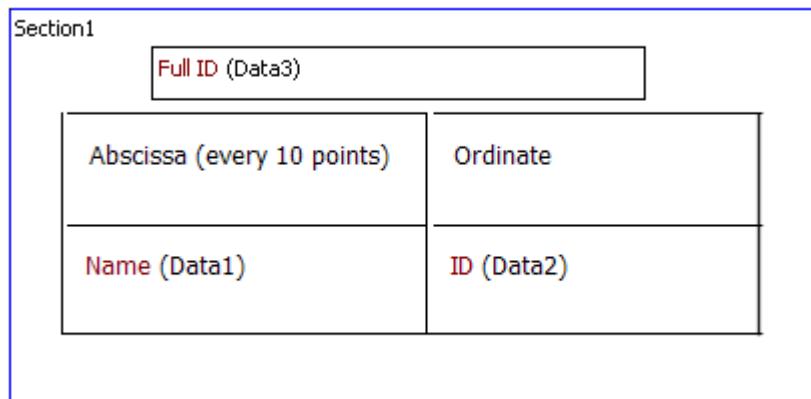
20. Select the Section Layout tool



The mouse pointer changes to

21. Drag the mouse around the table.

A section is created around the table:



Section1	
Full ID (Data3)	
Abscissa (every 10 points)	Ordinate
Name (Data1)	ID (Data2)

and the Section Properties dialog is displayed.

22. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.
23. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

24. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Sample31.Mean

500	0.397299242817583
490	0.390321818452634
480	0.381721202023122
470	0.373869560485572
460	0.37212714517834
450	0.36168258380768
440	0.359095741464726
430	0.359006635514089
420	0.362487489049711

The Full ID is SampleX.Mean as this is the mean of all the replicates for the sample.

NOTE: You can specify the number of significant figures or decimal places by setting the properties of the data object. Close the print preview window if this is still open. Right-click on the data object and select **Properties**. The Numeric Data Object Properties dialog is displayed. You can select **Significant figures** or **Decimal places** and specify the number of Significant figures or Decimal places from the appropriate drop-down list.

Task List – Custom Table Samples

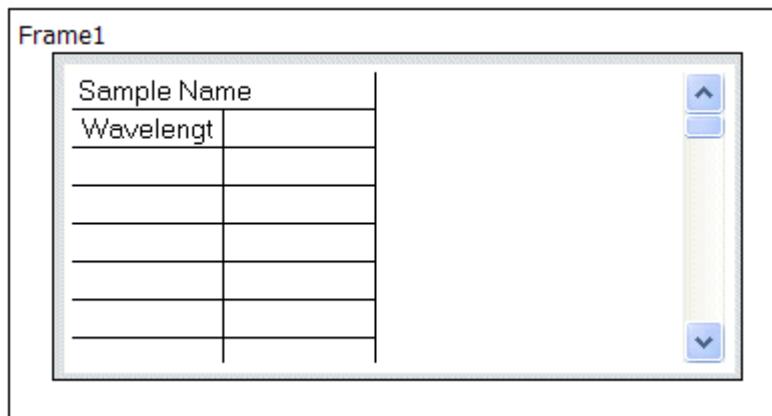
How do I add a Wavelength Table to my report template?

1. Select the **Wavelength Table** data object:



The mouse pointer changes to .

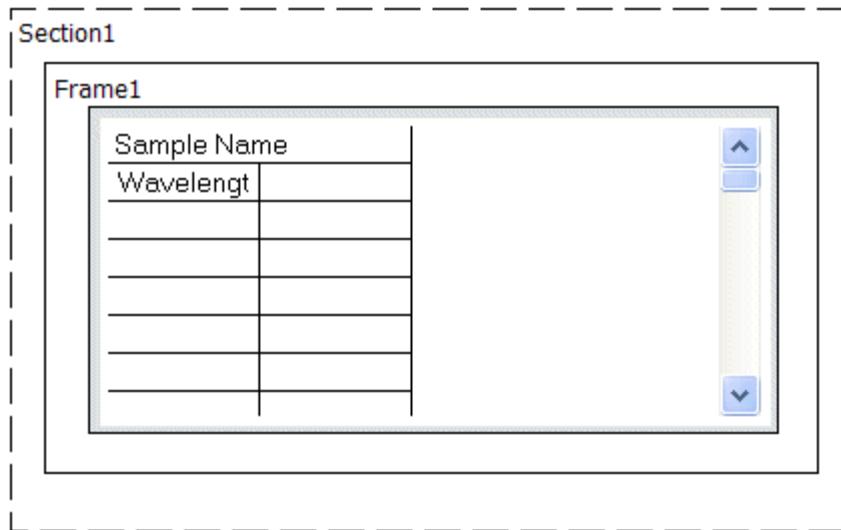
2. Click and drag the mouse to create a frame:



The size of the Frame determines the size of the table in the printed report.

3. Select the Section layout tool

4. Create a section around the frame:



The Section Properties dialog is displayed.

5. Select **Repeat section based on**, and then select **Wavelength Data** from the drop-down list.
6. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

The report is displayed. This is how the report will appear when printed.

Sample1.Sample	
Wavelength (nm)	Value (A)
175	0.94224
865	0.720284
3300	10

Sample2.Sample	
Wavelength (nm)	Value (A)
175	0.231817
865	1.35717
3300	10

Sample3.Sample	
Wavelength (nm)	Value (A)
175	-0.00440505
865	1.35774
3300	10

Method List

How do I add the following information to my report template – Method Name, Revision, Type, Description, Status, Method ID, Created by (Name and ID), Modified by (Name and ID), Date Created, Date Modified and SOP Document attached?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 13 rows.



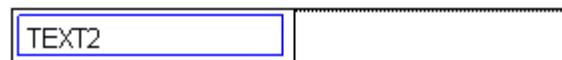
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field.



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Name' has been entered.

8. Repeat steps 5 to 7 to enter text for the other output settings:

Name	
Revision	
Type	
Description	
Status	
Method ID	
Created by (Name)	
Created by (ID)	
Modified by (Name)	
Modified by (ID)	
Date Created	
Date Modified	
SOP Document	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Name	Name (Data62)
Revision	
Type	
Description	
Status	
Method ID	
Created by (Name)	
Created by (ID)	
Modified by (Name)	
Modified by (ID)	
Date Created	
Date Modified	
SOP Document	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Name	Name (Data62)
Revision	Revision (Data63)
Type	Type (Data64)
Description	Description (Data65)
Status	Status (Data66)
Method ID	Method ID (Data67)
Created by (Name)	Name (Data68)
Created by (ID)	ID (Data69)
Modified by (Name)	Name (Data70)
Modified by (ID)	ID (Data71)
Date Created	Date Created (Data72)
Date Modified	Date Modified (Data73)
SOP Document	SOP Document (Data74)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Name	Quant Method 3
Revision	1
Type	Wavelength quant
Description	
Status	Draft
Method ID	{4167AA4A-E0B9-4512-A03F-38411BC54531}
Created by (Name)	tracey
Created by (ID)	tracey
Modified by (Name)	Q
Modified by (ID)	q
Date Created	09/09/2003 02:09:41 PM GMT Standard Time
Date Modified	09/09/2003 02:09:41 PM GMT Standard Time
SOP Document	None

NOTE: The **Description** field is empty as a Description of the Method was not entered when the Method was created.

Data Collection Settings

What Data Collection Settings are available to display on my template?

The following data collection settings can be added to your template.

NOTE: The terms used below are the Index keywords that must be used to return the value for the setting. Some of the terms are specific to the type of method that you are running (for example, Total Time is applicable to Timedrive but not Scan methods).

For High performance instruments:

- Total Time
- Mode
- D2 Lamp On
- Tungsten Lamp On
- Lamp Change-over Wavelength
- UV/Vis Slit Mode
- UUVis Slit Width
- Photomultiplier Gain
- Photomultiplier Response
- Sample Beam Position
- Ordinate Type
- Common Beam Mask
- Common Beam Mask Mode
- Cycle Count
- Display Time Units
- Time Interval
- Cycle Time
- Advanced Slit Width state @ 900nm
(the value depends on the setting in the method)
- Advanced Slit Width value @ 900nm
(the value depends on the setting in the method)
- Advanced Detector Response state @ 900nm (the value depends on the setting in the method)

For Medium performance instruments:

- Scan Range Start
- Scan Range End
- Scan Speed
- Data Interval
- Cycle Count
- Cycle Time
- Ordinate Type
- Slit Width
- UV Lamp On
- Visible Lamp On
- Lamp Change-over Wavelength
- Total Time
- Response
- Wavelengths
- Auto-sampler Delay Duration
- Auto-sampler Description
- Auto-sampler Use Internal Sipper
- Auto-sampler Probe Depth
- Auto-sampler Return Cell
- Auto-sampler Sample Cell
- Auto-sampler Fill Duration
- Auto-sampler Flush Duration
- Auto-sampler Return to Cell

- Advanced Detector Response value @ 900nm (the value depends on the setting in the method)
- Wavelength @ 400.00nm (the value depends on the setting in the method)
- Polarization Scan Start Angle
- Polarization Scan End Angle
- Polarization Scan Step Angle
- Scan Range Start
- Scan Range End
- Data Interval
- Common Beam Depolarizer
- Polarizer Depolarizer
- Polarizer Depolarizer Mode
- Sample Beam Attenuator
- Reference Beam Attenuator
- Auto-sampler Delay Duration
- Auto-sampler Description
- Auto-sampler Use Internal Sipper
- Auto-sampler Probe Depth
- Auto-sampler Return Cell
- Auto-sampler Sample Cell
- Auto-sampler Fill Duration
- Auto-sampler Flush Duration
- Auto-sampler Return to Cell
- Auto-sampler Tray Layout
- Auto-sampler Autozero Cell
- Cell Changer Description
- Cell Changer Type
- References
- Auto-sampler Tray Layout
- Auto-sampler Autozero Cell
- Cell Changer Description
- Cell Changer Type
- References
- Autozero
- Front Cell 1
- Front Cell 2
- Front Cell 3
- Front Cell 4
- Front Cell 5
- Front Cell 6
- Front Cell 7
- Front Cell 8
- Front Cell 9
- Front Cell 10
- Front Cell 11
- Front Cell 12
- Front Cell 13
- Rear Cell 1
- Rear Cell 2
- Rear Cell 3
- Rear Cell 4
- Rear Cell 5
- Rear Cell 6
- Rear Cell 7
- Rear Cell 8
- Rear Cell 9
- Rear Cell 10

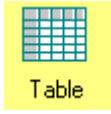
- Autozero
- Front Cell 1
- Front Cell 2
- Front Cell 3
- Front Cell 4
- Front Cell 5
- Front Cell 6
- Front Cell 7
- Front Cell 8
- Front Cell 9
- Front Cell 10
- Front Cell 11
- Front Cell 12
- Front Cell 13
- Rear Cell 1
- Rear Cell 2
- Rear Cell 3
- Rear Cell 4
- Rear Cell 5
- Rear Cell 6
- Rear Cell 7
- Rear Cell 8
- Rear Cell 9
- Rear Cell 10
- Rear Cell 11
- Rear Cell 12
- Rear Cell 13
- Peltier Control
- Peltier Description
- Rear Cell 11
- Rear Cell 12
- Rear Cell 13
- Peltier Control
- Peltier Description
- Peltier Type
- Peltier Temperature
- Peltier External Probe
- Sipper Delay Duration
- Sipper Description
- Sipper Fill Duration
- Sipper Flush Duration
- Sipper Return to Cell
- Sipper Mode

- Peltier Type
- Peltier Temperature
- Peltier External Probe
- Sipper Delay Duration
- Sipper Description
- Sipper Fill Duration
- Sipper Flush Duration
- Sipper Return to Cell
- Sipper Mode

How do I add a table of selected Data Collection Settings to my template?

By creating a custom table that only contains specific data collection settings, you avoid having all entries in the table.

Before you create the table, you need to decide which data collection settings you wish to include so you know how big to create the table. This list is not in the software, but is listed within "What data collection settings are available to display on my template?" on this page of the Help.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **3** columns and the required number of rows.

The number of rows should include one extra row for the table headings.



5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field in the first row of the table.

A text object is placed in the field:

TEXT2	
-------	--



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Name' has been entered.

Name		

8. Repeat steps 5-7 for 'Value' and 'Units' headings.

Name	Value	Units

9. Click on the **Name** data object within Data Collection Settings in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks something like:

Name	Value	Units
Name (DATA1)		

and the data object is added to the tree.

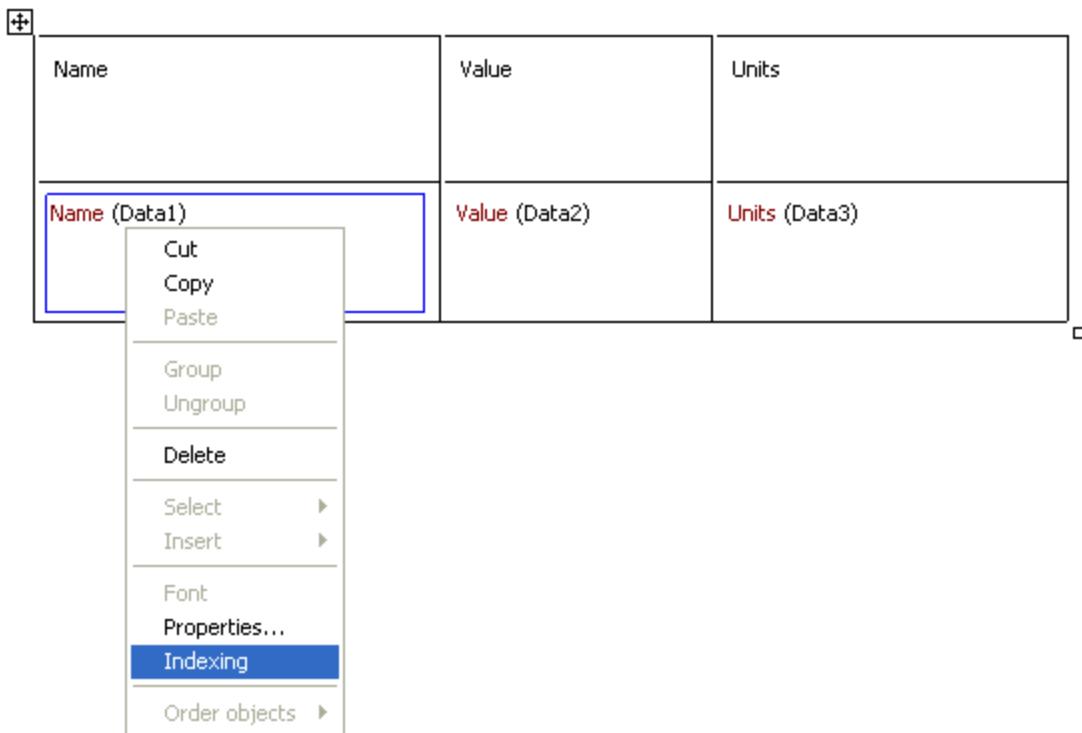
11. Repeat steps 9 and 10 for the **Value** and **Units** data objects.

The table should now look like this:

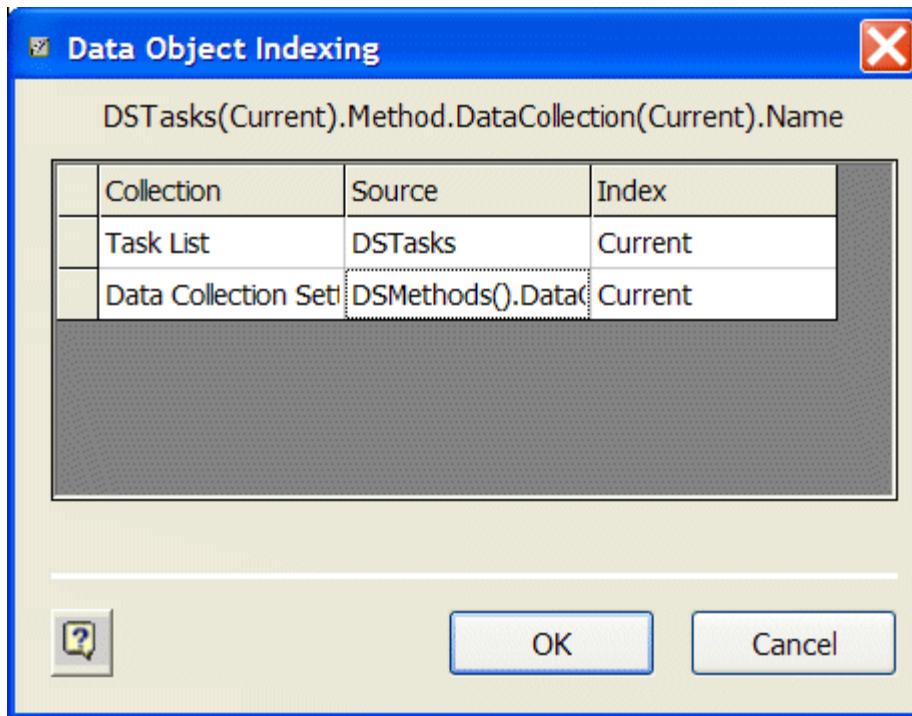
Name	Value	Units
Name (DATA1)	Value (DATA2)	Units (DATA3)

The instrument setting (DATA1) that has been placed in the table will by default be the first setting in the instrument setting list shown above. This needs to be amended for the setting you wish to report.

12. Using the right mouse button, click on the Name data object and select **Indexing**.

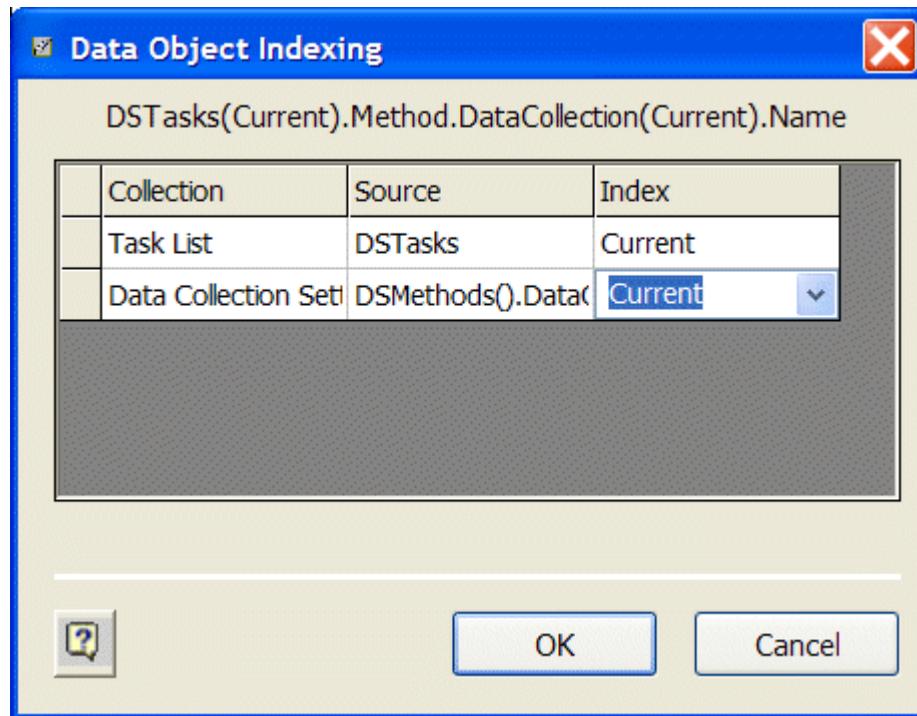


The Data Object Indexing dialog is displayed.



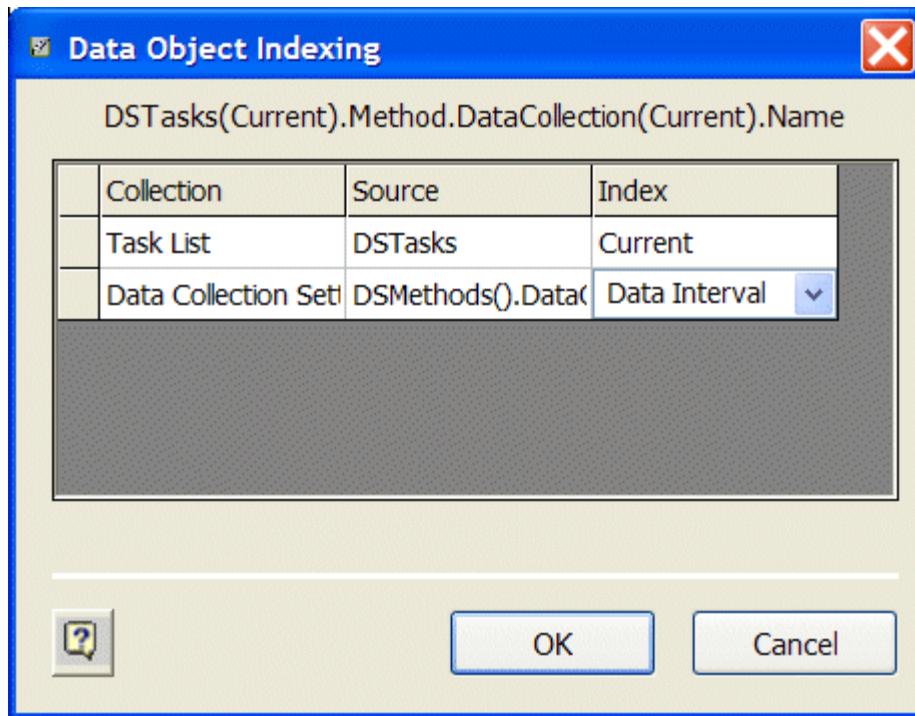
To change the setting, the Index must be changed.

13. Click the right mouse button on the Index field for the Data Collection Settings.
The text is highlighted and a drop-down arrow appears ▾.



14. Using the instrument settings list given in What data collection settings are available to display on my template ? below, enter the instrument settings you wish to include.

NOTE: You must use the exact terms used in the list above for the software to understand and return the required value. If an incorrect Index term is used, the Value field will be empty when the report is printed.



15. Repeat steps 9 to 14 for all the other instrument settings you wish to add to your table.

You can verify that you have the correct Index terms by using print preview. The value field for each instrument setting will be populated.

16. Click .

The report is displayed and the data objects are populated with the information.
This is how the report will appear when printed:

Name	Value	Units
Scan Range Start	780	nm
Scan Range End	380	nm
Scan Speed	960	nm/min
Data Interval	1	nm

How do I add a table of all the Data Collection Settings to my template?

The following data collection settings can be added to your template.

NOTE: The terms used below are the Index keywords that must be used to return the value for the setting. Some of the terms are specific to the type of method that you are running (for example, Total Time is applicable to Timedrive but not Scan methods).

For High performance instruments:

- Total Time
- Mode
- D2 Lamp On
- Tungsten Lamp On
- Lamp Change-over Wavelength
- UVVis Slit Mode
- UVVis Slit Width
- Photomultiplier Gain
- Photomultiplier Response
- Sample Beam Position
- Ordinate Type
- Common Beam Mask
- Common Beam Mask Mode
- Cycle Count
- Display Time Units
- Time Interval
- Scan Range Start
- Scan Range End
- Scan Speed
- Data Interval
- Cycle Count
- Cycle Time
- Ordinate Type
- Slit Width
- UV Lamp On
- Visible Lamp On
- Lamp Change-over Wavelength
- Total Time
- Response
- Wavelengths
- Auto-sampler Delay Duration
- Auto-sampler Description

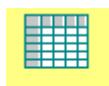
For Medium performance instruments:

- Cycle Time
- Advanced Slit Width state @ 900nm (the value depends on the setting in the method)
- Advanced Slit Width value @ 900nm (the value depends on the setting in the method)
- Advanced Detector Response state @ 900nm (the value depends on the setting in the method)
- Advanced Detector Response value @ 900nm (the value depends on the setting in the method)
- Wavelength @ 400.00nm (the value depends on the setting in the method)
- Polarization Scan Start Angle
- Polarization Scan End Angle
- Polarization Scan Step Angle
- Scan Range Start
- Scan Range End
- Data Interval
- Common Beam Depolarizer
- Polarizer Depolarizer
- Polarizer Depolarizer Mode
- Sample Beam Attenuator
- Reference Beam Attenuator
- Auto-sampler Delay Duration
- Auto-sampler Description
- Auto-sampler Use Internal Sipper
- Auto-sampler Probe Depth
- Auto-sampler Return Cell
- Auto-sampler Sample Cell
- Auto-sampler Fill Duration
- Auto-sampler Use Internal Sipper
- Auto-sampler Probe Depth
- Auto-sampler Return Cell
- Auto-sampler Sample Cell
- Auto-sampler Fill Duration
- Auto-sampler Flush Duration
- Auto-sampler Return to Cell
- Auto-sampler Tray Layout
- Auto-sampler Autozero Cell
- Cell Changer Description
- Cell Changer Type
- References
- Autozero
- Front Cell 1
- Front Cell 2
- Front Cell 3
- Front Cell 4
- Front Cell 5
- Front Cell 6
- Front Cell 7
- Front Cell 8
- Front Cell 9
- Front Cell 10
- Front Cell 11
- Front Cell 12
- Front Cell 13
- Rear Cell 1
- Rear Cell 2
- Rear Cell 3

- Auto-sampler Flush Duration
- Auto-sampler Return to Cell
- Auto-sampler Tray Layout
- Auto-sampler Autozero Cell
- Cell Changer Description
- Cell Changer Type
- References
- Autozero
- Front Cell 1
- Front Cell 2
- Front Cell 3
- Front Cell 4
- Front Cell 5
- Front Cell 6
- Front Cell 7
- Front Cell 8
- Front Cell 9
- Front Cell 10
- Front Cell 11
- Front Cell 12
- Front Cell 13
- Rear Cell 1
- Rear Cell 2
- Rear Cell 3
- Rear Cell 4
- Rear Cell 5
- Rear Cell 6
- Rear Cell 7
- Rear Cell 8
- Rear Cell 9
- Rear Cell 10
- Rear Cell 11
- Rear Cell 12
- Rear Cell 13
- Peltier Control
- Peltier Description
- Peltier Type
- Peltier Temperature
- Peltier External Probe
- Sipper Delay Duration
- Sipper Description
- Sipper Fill Duration
- Sipper Flush Duration
- Sipper Return to Cell
- Sipper Mode

- Rear Cell 9
- Rear Cell 10
- Rear Cell 11
- Rear Cell 12
- Rear Cell 13
- Peltier Control
- Peltier Description
- Peltier Type
- Peltier Temperature
- Peltier External Probe
- Sipper Delay Duration
- Sipper Description
- Sipper Fill Duration
- Sipper Flush Duration
- Sipper Return to Cell
- Sipper Mode

All the data collection settings can be displayed by creating a table that repeats on data collection settings. However, this will display ALL settings and these may not be applicable to your particular task.



1. Select the Table Layout tool



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **3** columns and **2** rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field in the first row of the table.

A text object is placed in the field:

TEXT2	
-------	--



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Name' has been entered.

Name		

8. Repeat steps 5-7 for 'Value' and 'Units' headings.

Name	Value	Units

9. Click on the **Name** data object in the Data Object list to select it.

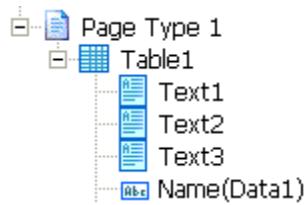


The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks something like:

Name	Value	Units
Name (DATA1)		

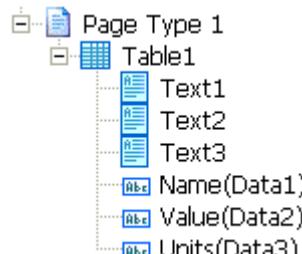


and the data object is added to the tree:

11. Repeat steps 9 and 10 for the **Value** and **Units** data objects.

The table should now look like this:

Name	Value	Units
Name (DATA1)	Value (DATA2)	Units (DATA3)

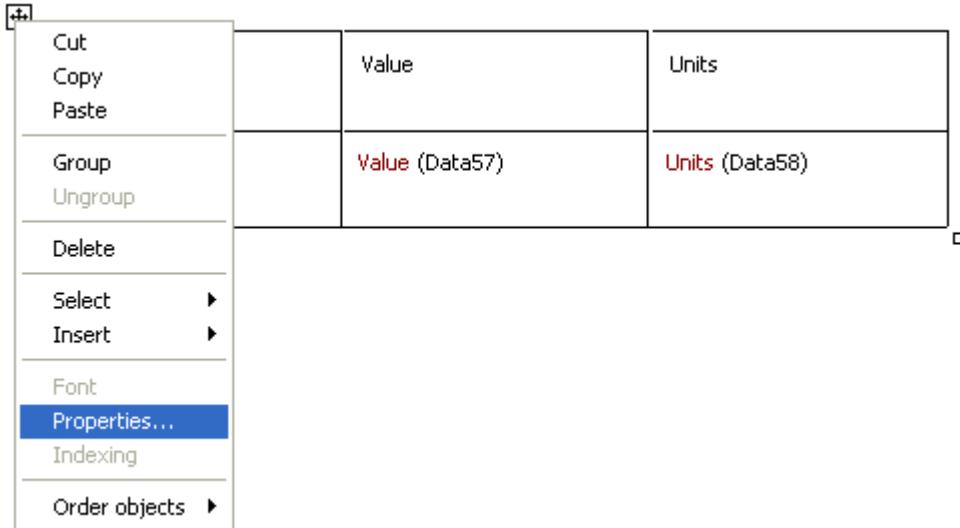


and the tree:



12. Click on TABLE in the tree to select the table

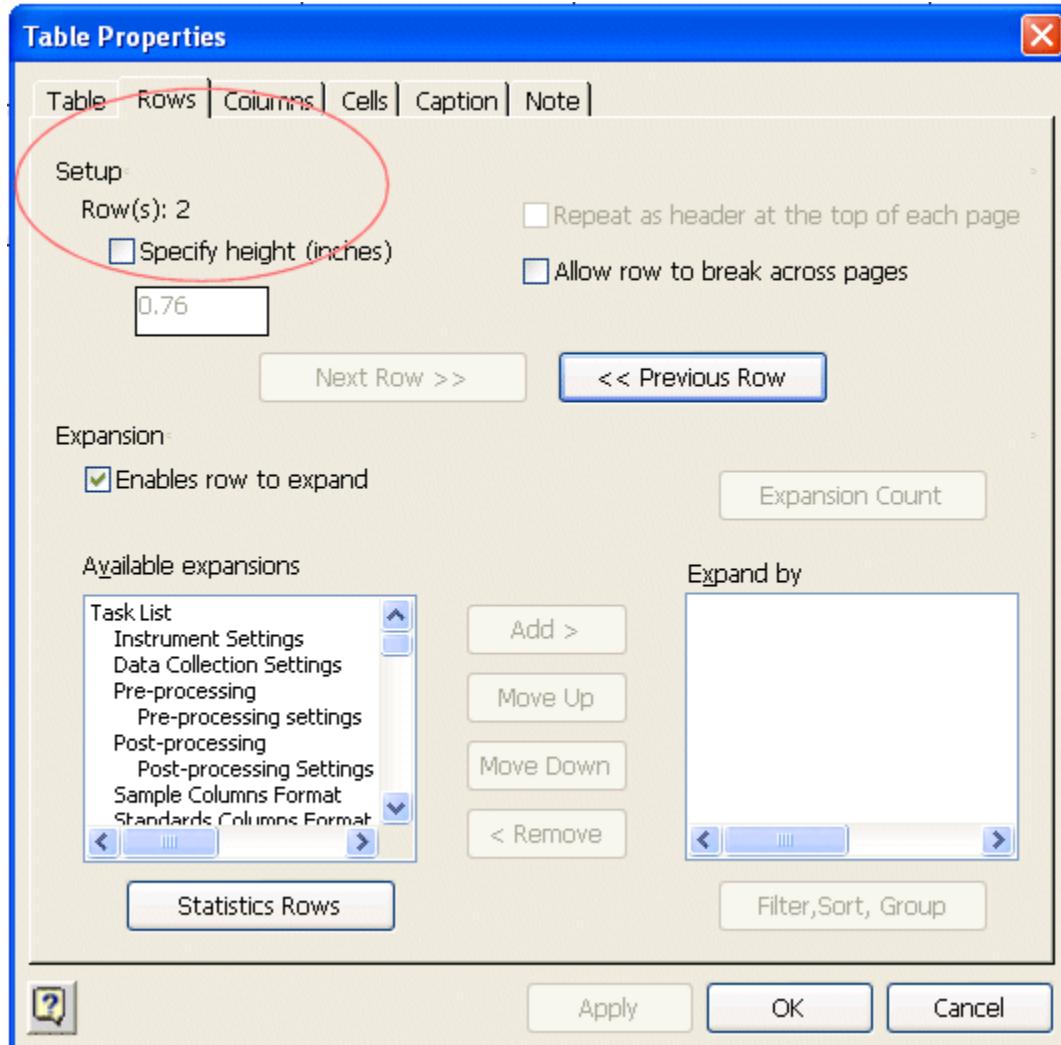
13. Right-click on the table and select **Properties**



The Table Properties dialog is displayed.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.



15. Ensure **Enables row to expand** is selected.
16. From the list of Available expansions, select **Data Collection Settings**, and then click **Add**.
Data Collection Settings moves from the Available expansions list to the Expand by list.
17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click 

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed (only part of the report is shown below for illustration):

Name	Value	Units
Mode	Wavelength Program	
D2 Lamp On	True	
Tungsten Lamp On	True	
Lamp Change-over Wavelength	319.2	nm
UVVis Slit Mode	Fixed	
UVVis Slit Width	2	nm

Timedrive

How do I add the Timedrive time interval and total time to my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

The Table is added to the tree: Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 2 rows.



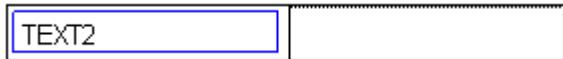
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Total Time' has been entered.

Total Time	

8. Repeat steps 5–7 to enter text for Time Interval:

Total Time	
Time Interval	

9. Click on the **Total Time** data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Total Time	Total time (Data5)
Time Interval	

11. Repeat steps 9 and 10 for Time Interval:

Total Time	Total time (Data5)
Time Interval	Time interval (Data6)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Total Time	180 secs
Time Interval	60 secs

Corrections Settings

How do I include information about the Correction Settings in my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree: Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 6 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

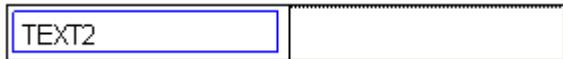


Table1

and the text object is added below table on the tree:



Text2

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Collection Type' has been entered.

Collection Type	

8. Repeat steps 5–7 to enter text for the other output settings:

Collection Type	
Expiry Time	
Baseline Correction	
Automatic Attenuator Correction	
Reflectance Correction	
Absorptance Correction	

9. Click on the required data object in the Data Object list to select it:



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Collection Type	Collection Type (Data1)
Expiry Time	
Baseline Correction	
Automatic Attenuator Correction	
Reflectance Correction	
Absorptance Correction	

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Collection Type	Collection Type (Data1)
Expiry Time	Expiry Time (Data2)
Baseline Correction	Baseline Correction (Data3)
Automatic Attenuator Correction	Automatic Attenuator Correction (Data4)
Reflectance Correction	Reflectance Correction (Data5)
Absorptance Correction	Absorptance Correction (Data6)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Collection Type	On task start
Expiry Time	5 days
Baseline Correction	None
Automatic Attenuator Correction	Do not invalidate attenuator corrections
Reflectance Correction	None
Absorptance Correction	None

Quant Settings

How do I add the Quant Settings information to my report template?

Any of these data objects can be placed individually on the report template. However, the example below shows all this information (excluding wavelengths, which is explained separately below) in one table, as the individual values may otherwise be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 18 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

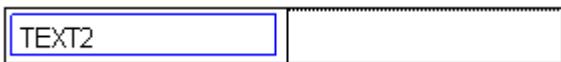


Table1

and the text object is added below table on the tree:

Text2

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Component Name**' has been entered.

Component Name	

8. Repeat steps 5–7 to enter text for the other output settings:

Component Name	
Units	
Calibration type	
Type of curve	
Force through zero	
Force recalibration	
Factor	
Slope	
Intercept	
Use correlation	
Correlation limit	
Use standard tolerance	
Standard tolerance limit (%)	
Control sample tolerance limit (%)	
Allow 10% extrapolation	
Proceed on error	
Ordinate mode	
Baseline corection	

9. Click on the required data object in the Data Object list to select it:



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Component Name	Component name (Data38)
Units	
Calibration type	
Type of curve	
Force through zero	
Force recalibration	
Factor	
Slope	
Intercept	
Use correlation	
Correlation limit	
Use standard tolerance	
Standard tolerance limit (%)	:
Control sample tolerance limit (%)	
Allow 10% extrapolation	
Proceed on error	
Ordinate mode	
Baseline corection	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Component Name	Component name (Data38)
Units	Units (Data39)
Calibration type	Calibration type (Data40)
Type of curve	Type of curve (Data41)
Force through zero	Force through zero (Data42)
Force recalibration	Force recalibration (Data43)
Factor	Factor (Data44)
Slope	Slope (Data45)
Intercept	Intercept (Data46)
Use correlation	Use correlation (Data47)
Correlation limit	Correlation limit (Data48)
Use standard tolerance	Use standard tolerance (Data49)
Standard tolerance limit (%)	Standard tolerance limit (%) (Data50)
Control sample tolerance limit (%)	Control sample tolerance limit (%) (Data51)
Allow 10% extrapolation	Allow 10% extrapolation (Data52)
Proceed on error	Proceed on error (Data53)
Ordinate mode	Ordinate mode (Data54)
Baseline corection	Baseline correction (Data55)

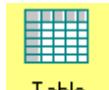
To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Component Name	None
Units	None
Calibration type	Calibration curve
Type of curve	Linear
Force through zero	No
Force recalibration	No
Factor	N/A
Slope	N/A
Intercept	N/A
Use correlation	Yes
Correlation limit	0.98
Use standard tolerance	No
Standard tolerance limit (%)	N/A
Control sample tolerance limit (%)	N/A
Allow 10% extrapolation	Yes
Proceed on error	Yes
Ordinate mode	N/A
Baseline corection	None

How do I add the following Quant information to my report template – Single point wavelength, Max height start and end wavelength, Area start and end wavelength, and base points?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 7 rows.



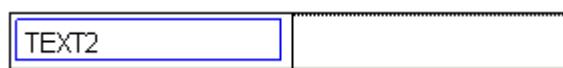
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Single Point Wavelength**' has been entered.

Single Point Wavelength	

8. Repeat steps 5–7 to enter text for the other output settings:

Single Point Wavelength	
Max Height Start Wavelength	
Max Height End Wavelength	
Area Start Wavelength	
Area End Wavelength	
Base Point 1	
Base Point 2	

9. Click on the required data object in the Data Object list to select it:



The mouse pointer changes to

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Single Point Wavelength	Single point wavelength (nm) (DATA)
Max Height Start Wavelength	
Max Height End Wavelength	
Area Start Wavelength	
Area End Wavelength	
Base Point 1	
Base Point 2	

and the data object is added to the tree.

- Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Single Point Wavelength	Single point wavelength (nm) (DATA)
Max Height Start Wavelength	Max height start wavelength (nm) (D
Max Height End Wavelength	Max height end wavelength (nm) (DA
Area Start Wavelength	Area start wavelength (nm) (DATA4)
Area End Wavelength	Area end wavelength (nm) (DATA5)
Base Point 1	Base point 1 (nm) (DATA6)
Base Point 2	Base point 2 (nm) (DATA7)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Single Point Wavelength	N/A
Max Height Start Wavelength	500
Max Height End Wavelength	350
Area Start Wavelength	N/A
Area End Wavelength	N/A
Base Point 1	400
Base Point 2	500

NOTE: In this example, Single Point Wavelength, Area Start Wavelength and Area End Wavelength are not applicable to the task and so N/A (not applicable) is reported on the report).

Rate Settings

How do I add the Calculation Type to my report template?

1. Click on **Calculation Type** in the Data Object list to select it.



2. The mouse pointer changes to

3. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Calculation Type (Data1)

and the Calculation Type object is added to the tree on the left hand side of the template.

To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Slope analysis

NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See "How do I place the data object in a table with text to explain what the object is?"

How do I add the following information to my report template – Start slope, End slope, baseline correction, basepoints, enzyme activity factor, enzyme component name and activity units?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 9 rows.



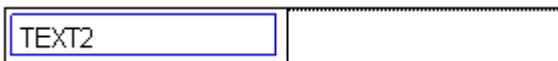
5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Start Slope**' has been entered.

Start Slope	

8. Repeat steps 5–7 to enter text for the other output setting:

Start Slope	
End Slope	
Baseline Correction	
Base point 1	
Base point 2	
Enzyme activity	
Enzyme activity factor	
Enzyme component name	
Enzyme activity units	

9. Click on the required data object in the Data Object list to select it:



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Start Slope	Start slope (Data3)
End Slope	
Baseline Correction	
Base point 1	
Base point 2	
Enzyme activity	
Enzyme activity factor	
Enzyme component name	
Enzyme activity units	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Start Slope	Start slope (DATA1)
End Slope	End slope (DATA2)
Baseline Correction	Use baseline correction (DATA3)
Base point 1	Base point 1 (DATA4)
Base point 2	Base point 2 (DATA5)
Calculate enzyme activity	Calculate enzyme activity (DATA6)
Enzyme activity factor	Enzyme activity factor (DATA7)
Enzyme component name	Enzyme component name (DATA8)
Enzyme activity units	Enzyme activity units (DATA9)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Start Slope	300 seconds
End Slope	450 seconds
Baseline Correction	Yes
Base point 1	200 seconds
Base point 2	250 seconds
Calculate enzyme activity	Yes
Enzyme activity factor	0,000001
Enzyme component name	Component 1
Enzyme activity units	mmol

How do I add the following information to my report template – end point analysis type, manual end point, base points, whether substrate concentration was calculated, molecular weight, pathlength of cuvette, molar extinction coefficient, substrate component name and substrate concentration units?

The individual data objects can be placed on the report template. The example below shows how to enter them into a table with identifying text.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 10 rows.



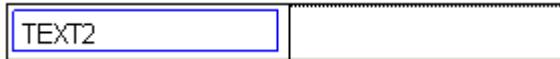
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'End point analysis type' has been entered.

8. Repeat steps 5–7 to enter text for the other output settings:

End point analysis type	
Manual end point	
Base point 1	
Base point 2	
Substrate concentration	
Molecular weight (gmol-1)	
Pathlength of cuvette (cm)	
Molar extinction coefficient (mmol-1 cm-1)	
Substrate component name	
Substrate concentration units	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

End point analysis type	End point analysis type (Data12)
Manual end point	
Base point 1	
Base point 2	
Substrate concentration	
Molecular weight (gmol-1)	
Pathlength of cuvette (cm)	
Molar extinction coefficient (mmol-1 cm-1)	
Substrate component name	
Substrate concentration units	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

End point analysis type	End point analysis type (Data12)
Manual end point	Manual end point (Data13)
Base point 1	Base point 1 (Data14)
Base point 2	Base point 2 (Data15)
Substrate concentration	Calculate substrate concentration (Data16)
Molecular weight (gmol-1)	Molecular weight (gmol-1) (Data17)
Pathlength of cuvette (cm)	Pathlength of cuvette (cm) (Data18)
Molar extinction coefficient (mmol-1 cm-1)	Molar extinction coefficient (mmol-1 cm-1) (D
Substrate component name	Substrate component name (Data20)
Substrate concentration units	Substrate concentration units (Data21)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

End point analysis type	Baseline corrected endpoint
Manual end point	N/A
Base point 1	450 seconds
Base point 2	550 seconds
Substrate concentration	Yes
Molecular weight (gmol-1)	1
Pathlength of cuvette (cm)	1
Molar extinction coefficient (mmol-1 cm-1)	1
Substrate component name	Component 1
Substrate concentration units	mmol

Pre-Processing and Post-Processing

How do I add the Pre-processing and Post-processing settings to my report template?

The Pre-processing data objects can be placed individually on a page, but this example shows how to place them in a table and repeat on a row so that all the Pre-processing settings are listed when the report is printed.

NOTE: The Post-processing options within a task are a subset of the Pre-processing options. The Pre-processing are discussed below. The same procedure should be followed for placing Post-processing information within a report template, but substituting the Post-processing data objects.

In this example, the task processing page contains two processing commands:

Process	Settings
Derivative	Order: Second; Width: 9
Arithmetic	Multiply; 20

This information will be added to the report template.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **3** columns and **2** rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field in the first row of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Pre-processing ProcessName**' has been entered.

Pre-processing Process Name		

8. Repeat steps 5–7 for 'Pre-processing settings name' and 'Pre-processing settings value' headings.

Pre-processing Process Name	Pre-processing settings name	Pre-processing settings value

9. Click on the **ProcessName** data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks something like:

Pre-processing Process Name	Pre-processing settings name	Pre-processing settings value
Process Name (DATA1)		

and the data object is added to the tree.

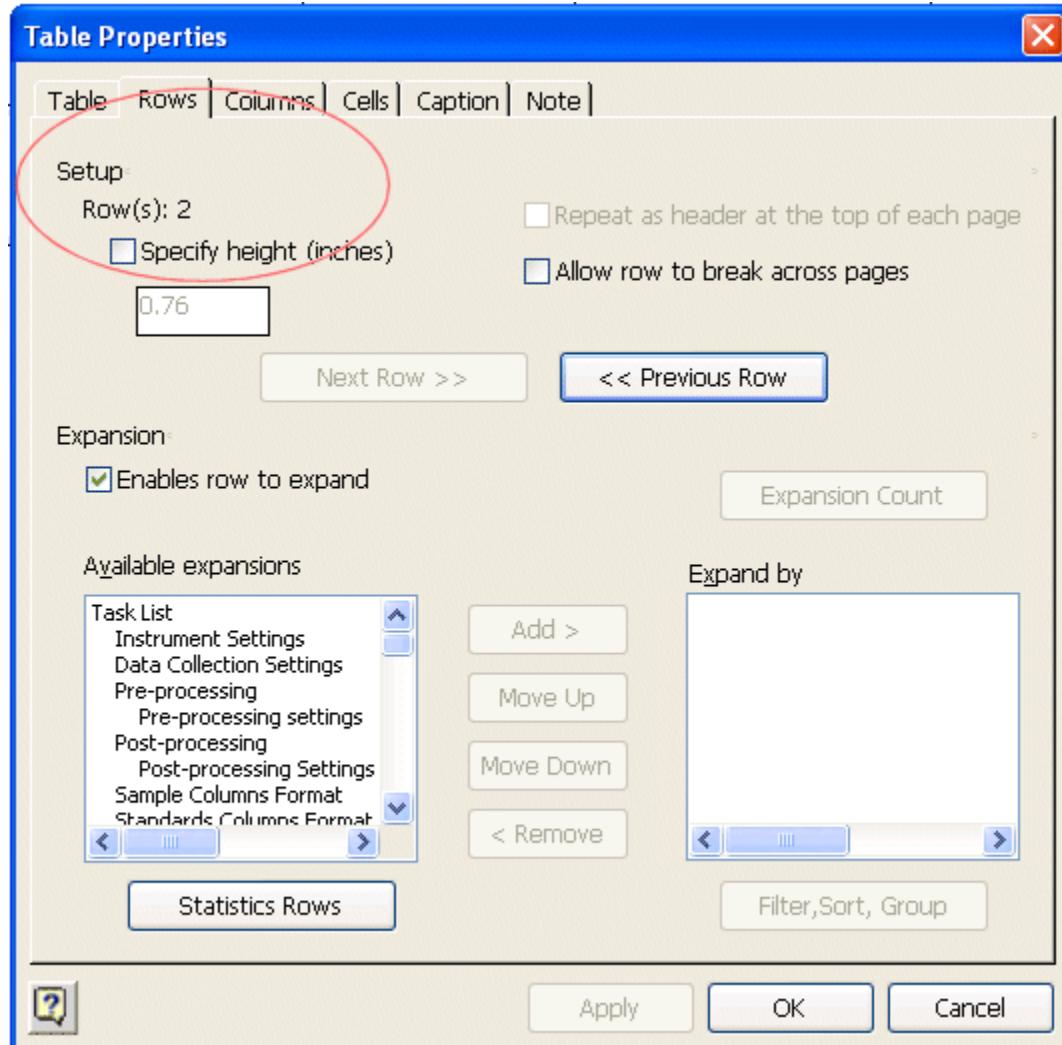
11. Repeat steps 9 and 10 for the **Pre-processing settingsName** and **Value** data objects.
 The table should now look like this:

Pre-processing Process Name	Pre-processing settings name	Pre-processing settings value
Process Name (DATA1)	Name (DATA2)	Value (DATA3)

12. Click on TABLE in the tree to select the table: 
13. Right-click on the table and select **Properties**:
- 
- | | Pre-processing settings name | Pre-processing settings value |
|----|------------------------------|-------------------------------|
| 1) | Name (Data2) | Value (Data3) |

The Table Properties dialog is displayed.

14. Click Next Row.
 Row 2 is now specified at the top of the dialog.



15. Ensure **Enables row to expand** is selected.
16. From the list of Available expansions, select **Pre-processing**, and then click **Add**. Pre-processing moves from the Available expansions list to the Expand by list.

NOTE: If you create a table that only contains the Pre-processing settings (Name and Value) and not the Process name, you should repeat on Pre-processing settings.

17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Pre-processing Process Name	Pre-processing settings name	Pre-processing settings value
Derivative	Settings for Derivative	Order: Second; Width: 9
Arithmetic	Settings for Arithmetic	Multiply; 20

Sample Table Definition

How do I include all the information about the Sample Table in the report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 9 rows.



Text Block

5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

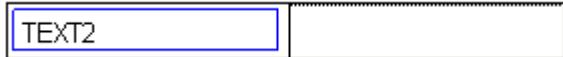
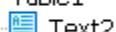


Table1



Text2

and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Preparation Type' has been entered.

Preparation Type	

8. Repeat steps 5–7 to enter text for the other sample table settings:

Preparation type	
Preparation parameters	
Pre-run	
Replicates per sample	
Replicates sequence type	
Sample control	
Sample ID format	
Visible columns	
Hidden columns	

This example uses Replicates per sample and Replicates sequence type. If you have used Measurements rather than Replicates, you should replace these examples.

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Preparation type	Preparation type (Data1)
Preparation parameters	
Pre-run	
Replicates per sample	
Replicates sequence type	
Sample control	
Sample ID format	
Visible columns	
Hidden columns	

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Preparation type	Preparation type (Data1)
Preparation parameters	Preparation parameters (Data2)
Pre-run	Pre-run (Data3)
Replicates per sample	Replicates/Measurements per sample (Data4)
Replicates sequence type	Replicates/Measurements sequence type (Data5)
Sample control	Sample control (Data6)
Sample ID format	Sample ID format (Data7)
Visible columns	Visible columns (Data8)
Hidden columns	Hidden columns (Data9)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Preparation type	Concentration
Preparation parameters	N/A
Pre-run	User must complete table
Replicates per sample	1
Replicates sequence type	By sample
Sample control	None
Sample ID format	Auto increment using format: Sample
Visible columns	Sample ID, Description, Position [Autosampler], Probe Depth [Autosampler], Return Cell [Autosampler], Fill Time (secs) [Autosampler], Flush/Return Time (secs) [Autosampler], Delay Time (secs) [Autosampler]
Hidden columns	Type, Concentration, yellow, Residual, Ordinate

How do I include information about the formatting of the Sample Table in my report template?

Any of these data objects can be placed individually on the report template. However, the example below shows all this information in one table, as the individual values may otherwise be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 6 rows.



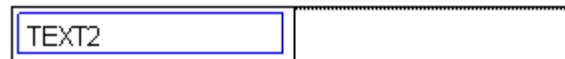
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Column Title**' has been entered.

Column title	

8. Repeat steps 5–7 to enter text for the other standards table settings:

Column title	
Column type	
Units	
Mandatory	
Numerical Format	
Font	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Column title	Column title (Data9)
Column type	
Units	
Mandatory	
Numerical Format	
Font	

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Column title	Column title (Data9)
Column type	Column type (Data10)
Units	Units (Data11)
Mandatory	Mandatory (Data12)
Numerical Format	Numerical format (Data13)
Font	Font (Data14)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Column title	Sample ID
Column type	System
Units	N/A
Mandatory	Yes
Numerical Format	N/A
Font	Tahoma, 10, Black, Regular

Standards Table Definition

How do I include all the information about the Standards Table in the report template?

Any of these data objects can be placed individually on the report template. However, the example below shows all this information in one table, as the individual values may otherwise be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree: Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 8 rows.



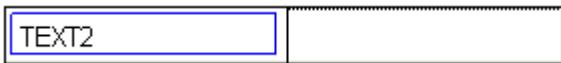
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Preparation Type**' has been entered.

Preparation Type	

8. Repeat steps 5–7 to enter text for the other standards table settings:

Preparation Type	
Preparation Parameters	
Pre-run	
Replicates per standard	
Replicates sequence type	
Standard ID format	
Visible columns	
Hidden columns	

Replicates have been used in this example, but if you have used Measurements, replace the text as required. The data object will automatically report Replicates or Measurements correctly.

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Preparation Type	Preparation Type (Data1)
Preparation Parameters	
Pre-run	
Replicates per standard	
Replicates sequence type	
Standard ID format	
Visible columns	
Hidden columns	

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Preparation Type	Preparation Type (Data1)
Preparation Parameters	Preparation Parameters (Data2)
Pre-run	Pre-run (Data3)
Replicates per standard	Replicates/Measurements per standard (Data4)
Replicates sequence type	Replicates/Measurements sequence type (Data5)
Standard ID format	Standard ID format (Data6)
Visible columns	Visible columns (Data7)
Hidden columns	Hidden columns (Data8)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Preparation Type	Concentration
Preparation Parameters	N/A
Pre-run	User must complete table
Replicates per standard	1
Replicates sequence type	By standard
Standard ID format	Auto increment using format: Standard
Visible columns	Standard ID, Concentration, Position [Autosampler], Probe Depth [Autosampler], Return Cell [Autosampler], Fill Time (secs) [Autosampler], Flush/Return Time (secs) [Autosampler], Delay Time (secs) [Autosampler]
Hidden columns	Description, Type, Sample Tag, yellow, Residual, Ordinate

How do I include information about the formatting of the Standards Table in my report template?

1. Select the Table Layout tool  .

The mouse pointer changes to  .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 6 rows.



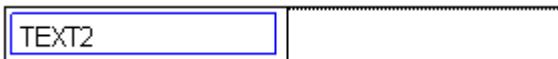
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Column Title**' has been entered.

Column title	

8. Repeat steps 5–7 to enter text for the other standards table settings:

Column title	
Column type	
Units	
Mandatory	
Numerical Format	
Font	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Column title	Column title (Data9)
Column type	
Units	
Mandatory	
Numerical Format	
Font	

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Column title	Column title (Data9)
Column type	Column type (Data10)
Units	Units (Data11)
Mandatory	Mandatory (Data12)
Numerical Format	Numerical format (Data13)
Font	Font (Data14)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Column title	Sample ID
Column type	System
Units	N/A
Mandatory	Yes
Numerical Format	N/A
Font	Tahoma, 10, Black, Regular

Results Table Definition

How do I add the results table definition information to the report template?

Any of these data objects can be placed individually on the report template. However, the example below shows all this information in one table, as the individual values may otherwise be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 8 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

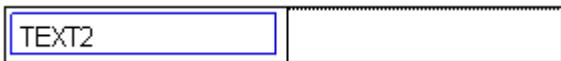
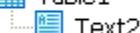


Table1



Text2

and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Visible Columns' has been entered.

Visible columns	

8. Repeat steps 5–7 to enter text for the other standards table settings:

Visible columns	
Hidden columns	
Column title	
Column type	
Units	
Mandatory	
Numerical format	
Font	.

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Visible columns	Visible columns (Data1)
Hidden columns	
Column title	
Column type	
Units	
Mandatory	
Numerical format	
Font	

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Visible columns	Visible columns (Data1)
Hidden columns	Hidden columns (Data2)
Column title	Column title (Data3)
Column type	Column type (Data4)
Units	Units (Data5)
Mandatory	Mandatory (Data6)
Numerical format	Numerical format (Data7)
Font	Font (Data8)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

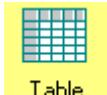
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Visible columns	Sample ID, yellow, Residual, Ordinate
Hidden columns	Description, Type, Concentration, Position [Autosampler], Probe Depth [Autosampler], Return Cell [Autosampler], Fill Time (secs) [Autosampler], Flush/Return Time (secs) [Autosampler], Delay Time (secs) [Autosampler]
Column title	Sample ID
Column type	System
Units	N/A
Mandatory	Yes
Numerical format	N/A
Font	Tahoma, 10, Black, Regular

Reporting Options

How do I add the report template name and revision to my template?

The report template name and revision can be included in the report template. The instructions below describe how to put this information in a small table.



1. Select the Table Layout tool



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter 2 columns and 2 rows.



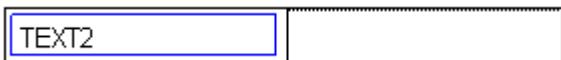
5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field:



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Report Template' has been entered.

Report Template	

8. Repeat steps 5 to 7 to enter text for **Template Revision**:

Report Template	
Template Revision	

- Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Report Template	Report Template (Data1)
Template Revision	

and the data object is added to the tree.

- Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Report Template	Report Template (Data1)
Template Revision	Template Revision (Data2)

To view what will actually appear when the report is printed you need to print preview the report.

- Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Report Template	Default-Quant
Template Revision	1

How do I add the report output settings to my template?

The report output settings (for example, print hardcopy) defined on the Output page of the Workspace can be included in the report template. The instructions below describe how to put this information in a table as the individual values such as Yes for Print hardcopy would be meaningless on the page.



- Select the Table Layout tool



The mouse pointer changes to .

- Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

- To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many reporting options you wish to include. The example below has 2 columns and 4 rows as all the information is reported.



5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Report frequency**' has been entered.

Report frequency	

8. Repeat steps 5–7 to enter text for the other output settings:

Report frequency	
Print Hardcopy	
Print to File	
Print to Database	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Report frequency	Report frequency (DATA1)
Print Hardcopy	
Print to File	
Print to Database	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Report frequency	Report frequency (DATA1)
Print Hardcopy	Print hardcopy (DATA2)
Print to File	Print to file (DATA3)
Print to Database	Print to database (DATA4)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

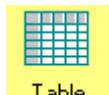
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Report frequency	On demand
Print Hardcopy	Yes
Print to File	Yes
Print to Database	No

How do I add the information about hard-copies of reports to my templates?

These data objects describe the printer which prints the hardcopy of the report, and the number of copies of the report.

The instructions below describe how to put this information in a table as the individual values such as **1** for **Number of copies** would be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Format**.

The format table dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--

and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below 'Printer' has been entered.

Printer	

8. Repeat steps 5–7 to enter text for the number of copies:

Printer	
Number of copies	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Printer	Printer (DATA6)
Number of copies	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the number of copies data object:

Printer	Printer (DATA6)
Number of copies	Number of copies (DATA7)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

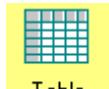
The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Printer	HP LaserJet 4000 Series PCL6
Number of copies	1

How do I add the information about report file output to my templates?

These data objects describe the location to which the report is saved and the file format of the report.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Format**.

The format table dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



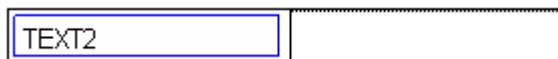
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Output folder path**' has been entered.

Output folder path	

8. Repeat steps 5–7 to enter text for the file type:

Output folder path	
File path	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Output folder path	Output folder path (DATA8)
File path	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the number of copies data object:

Output folder path	Output folder path (DATA8)
File path	File type (DATA9)

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

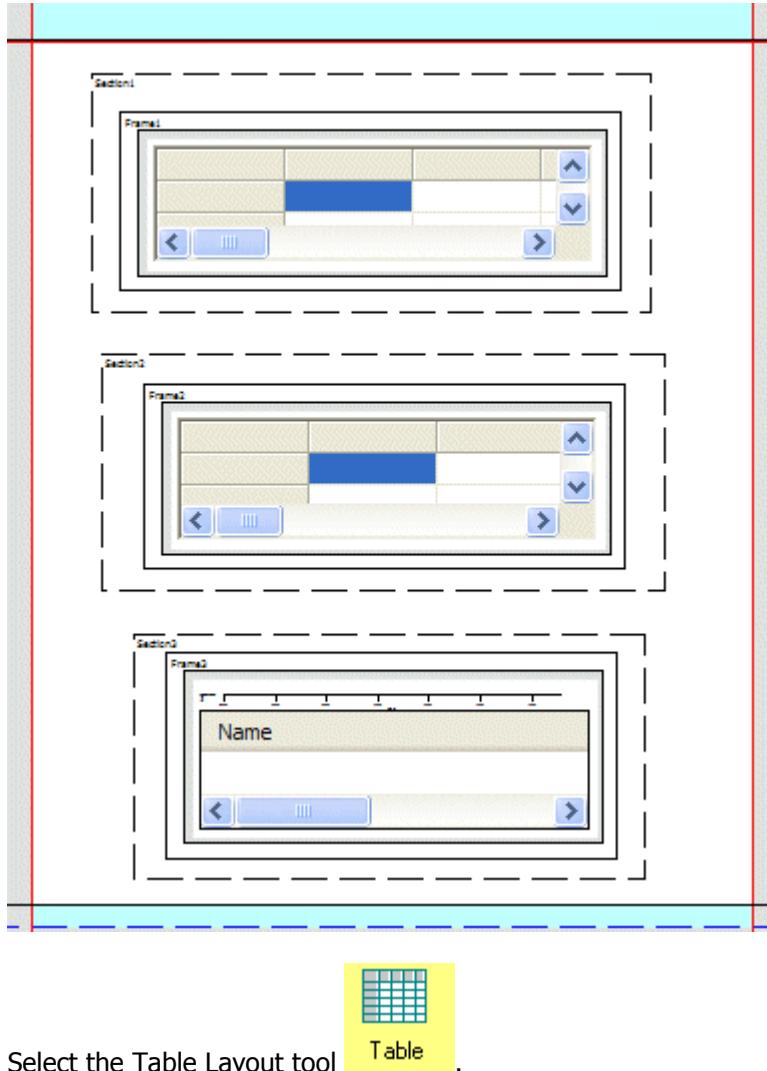
Output folder path	C:\Documents and
File path	ASCII (.txt)

How do I add information about which sections of the report are selected on or off?

If you have created a report template that contains sections, within the reporting setup you can specify which of these sections are on or off when the report is output. This information can be included within the report.

In the example below, a report is created that contains 3 sections. It also contains a table to report which of these sections are on and off.

1. Create a report that contains at least 1 sections. The example below contains 3 sections.



2. Select the Table Layout tool



The mouse pointer changes to

3. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

4. To format the size of the table, right click on the table and select **Properties**.

The format table dialog is displayed.

5. Select the Table tab and enter **2** columns and **2** rows.



6. Select the Text Block layout tool



The mouse pointer changes to

7. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree:

8. Click inside the blue box in the table and edit the text as required.

In the example below '**Sections on**' has been entered.

Sections On	

9. Repeat steps 5–7 to enter text for Sections Off:

Sections On	
Sections Off	

10. Select the **Sections On** data object.

Click in the field next to Sections On text:

Sections On	Sections on (Data1)
Sections Off	

11. Select the **Sections Off** data object, and then click in the field next to the Sections Off text:

Sections On	Sections on (Data1)
Sections Off	Sections off (Data2)

The Sections that are specified on and off are set on the Output page in the Workspace.

In this example 2 of the 3 sections are switched off (this means that the information within these sections would not be included when the report is generated).



To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

The table in the report shows which sections are switched on and off:

Sections On	Section2
Sections Off	Section1, Section3

How do I add the Call Application details to my templates?

These data objects describe the executable path of the application started, the files that were sent to the application and when during the task they were sent.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



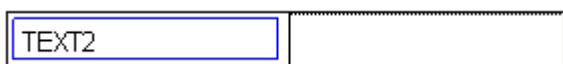
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field:



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

8. Repeat steps 5–7 to enter text for the Command:

Path	Commands	Input Files	Start	Wait

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.
11. Repeat steps 8–9 for the other data objects.

Palli	Commands	Input Files	Start	Wait
Executable path (Data57)	Commands (Data56)	Input files (Data55)	Start (Data54)	Wait (Data89)

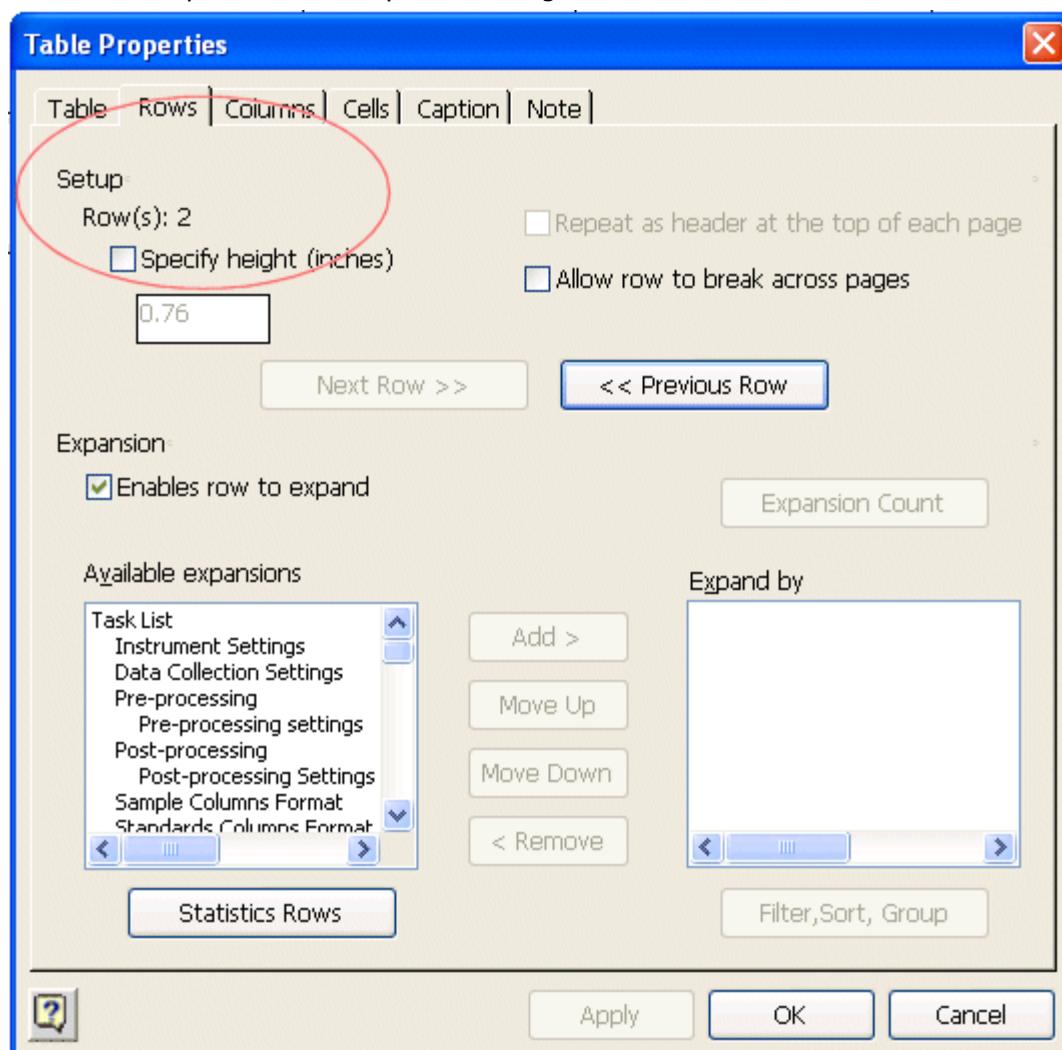
To obtain the information for more than one executable path, you need to repeat the second row of the table.

12. Right-click on the table and select **Properties** from the context menu.
The Table Properties dialog is displayed.

13. Select the Rows tab.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.



15. Ensure Enables row to expand is selected.

16. From the list of Available expansions, select **Call Application**, and then click **Add**.
Call Application moves from the Available expansions list to the Expand by list.
17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information.

NOTE: A number of templates supplied with UV WinLab provide a **Task Summary** that includes an expanding table containing the details of Call Application.

How do I add the Data Export details to my templates?

These data objects describe the data that was exported from the task at and the files formats used.

The instructions below describe how to put this information in a table as the individual values could be meaningless on the page.



1. Select the Table Layout tool  .


The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.
4. Select the Table tab and enter **2** columns and **2** rows.



5. Select the Text Block layout tool  .


The mouse pointer changes to

6. Click the mouse inside a field of the table.
A text object is placed in the field:

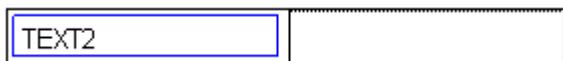


 Table1

and the text object is added below table on the tree:

 Text2

7. Click inside the blue box in the table and edit the text as required.

8. Repeat steps 5–7 to enter text for the other Data Export options.
9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.
11. Repeat steps 8–9 for the other Data Export data objects.

To view what will actually appear when the report is printed you need to print preview the report.

12. Click .

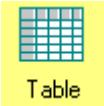
The report is displayed and the data objects are populated with the information.

NOTE: Some example templates are available that cover the reporting options. These are the **Task Summary Report** templates, which allow you to print all the details of your task, including the Output settings. They are available for each type of method.

Method Event Log

How do I add all the Method Event Log information to my report template?

Every event such saving is recorded in the Method Event Log. If these data objects were just placed on the page, only the most recent event would be displayed in the report. To display all events, a section must be created that repeats on the Method Event Log.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 7 rows.



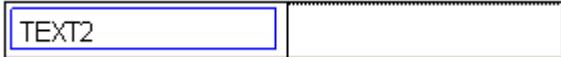
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Method Event**' has been entered.

Method Event	

8. Repeat steps 5–7 to enter text for the other output settings:

Method Event	
Revision	
Time	
Reason	
Comment	
Name	
ID	

9. Click on the required data object in the Data Object list to select it

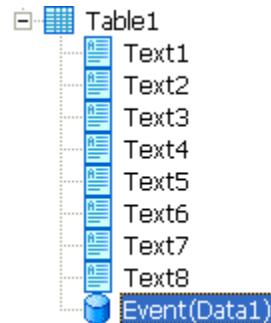


The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Method Event	Event (DATA1)
Revision	
Time	
Reason	
Comment	
Name	
ID	



and the data object is added to the tree:

11. Repeat steps 9 and 10 for the other output settings data objects except **Time**.
12. In the Time field add a text block:

Time	Text8
------	-------

13. Delete the default text (Text8 in the example above).
14. Select the Time data object and then click inside the text block.

The Time data object is placed inside the text block:

Time	[8:Time]
------	----------

This enables the time to text wrap within the text block. Otherwise the cell would need to be very long to display all of the date.

Method Event	Event (DATA1)
Revision	Revision (DATA2)
Time	[8:Time]
Reason	Reason (DATA4)
Comment	Comment (DATA5)
Name	Name (DATA6)
ID	ID (DATA7)

To display all the event log entries a section must be created around the table.



15. Select the Section Layout tool



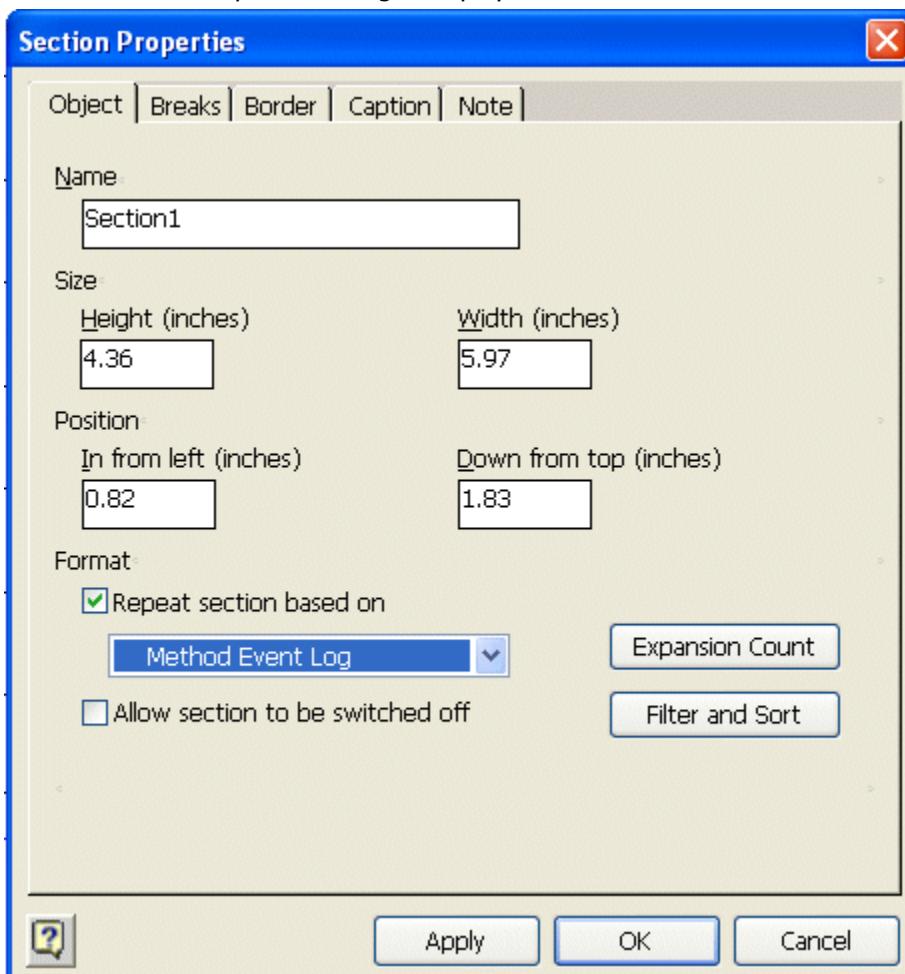
The mouse pointer changes to

16. Drag the mouse around the table.

A section is created around the table

SECTION1	
Method Event	Event (DATA1)
Revision	Revision (DATA2)
Time	[8:Time]
Reason	Reason (DATA4)
Comment	Comment (DATA5)
Name	Name (DATA6)
ID	ID (DATA7)

and the Section Properties dialog is displayed.



17. Ensure **Repeat section based on** is selected, and then select **Method Event Log** from the drop-down list.

18. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

19. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Method Event	Saved
Revision	1,000
Time	29 September 2003 09:15 Westeuropäische Normalzeit
Reason	Saved
Comment	
Name	Administrator
ID	Administrator

Method Event	Imported
Revision	1,000
Time	29 September 2003 09:04 Westeuropäische Normalzeit
Reason	
Comment	Imported from file D:\Lambda Max.wlm
Name	Administrator
ID	Administrator

Sample List

How do I include the ID, description, status and comments for each of the samples in my task in the report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 4 columns and 2 rows.



Text Block

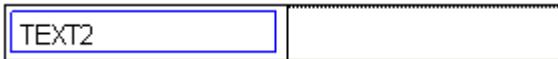
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below 'ID' has been entered.

ID			

8. Repeat steps 5–7 to enter text for the other output settings:

ID	Description	Status	Comments

NOTE: The text headings should be at the top of each column (rather than all rows in the first column) as the table will be repeated on the second row to obtain the information for all samples in the task.

9. Click on the required data object in the Data Object list to select it.

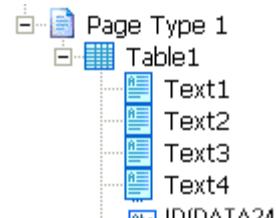


The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

ID	Description	Status	Comments
ID (DATA24)			



and the data object is added to the tree:

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

ID	Description	Status	Comments
ID (DATA24)	Description (DATA25)	Status (DATA26)	Comments (DATA27)

To obtain the ID, description, status and comments for all the samples in the task, you need to repeat the second row of the table.

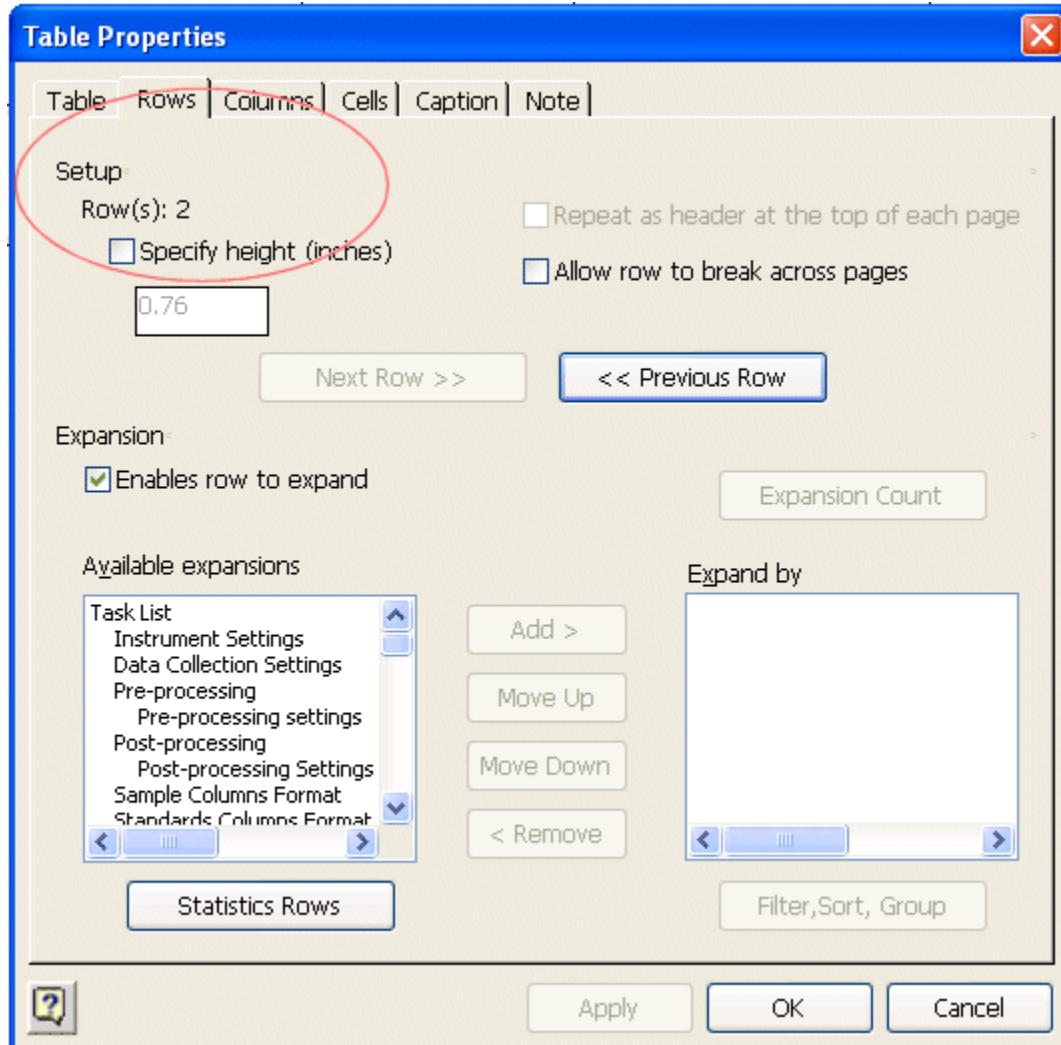
12. Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

13. Select the Rows tab.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.



15. Ensure **Enables row to expand** is selected.
16. From the list of Available expansions, select **Samples**, and then click **Add**. Samples moves from the Available expansions list to the Expand by list.
17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

ID	Description	Status	Comments
s10std1	Batch 10	Imported	
s10std2	Batch 10	Imported	
s10std3	Batch 10	Imported	
s10std4	Batch 10	Imported	

NOTE: In this example there are no comments associated with any of the samples and so the Comments fields are empty.

What is the difference between ID and Full ID?

The **ID** data object will report the Sample ID as seen in the Sample Table. This is sufficient if you have a fairly simple Sample Table, for example just 5 samples.

ID
Sample38
Sample37
Sample39
Sample40
Sample41

However, this does not report the full sample name which includes the extension (as seen in the Results Table). The **Full ID** is particularly useful if you are using Replicates or Measurements. If, for example, you have 1 sample with 2 replicates, and you create a table in the report template using **ID**, you will see the Sample ID twice but it will not show that these are replicates.

ID
Sample1
Sample1

If you use the **Full ID** data object, the full name including the extension will be shown.

Full ID
Sample1.Replicate1
Sample1.Replicate2

Sample Table and Results Table

How do I create a table that contains all the information in my Results Table?

To ensure that all your results are printed, you must use a custom table.

In this example we will recreate a Results Table from a task which has three columns – sample ID, Abs541 (where an equation was used to calculate the absorbance at 451 nm for each sample), and Conc.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 3 columns and 2 rows.



Text Block

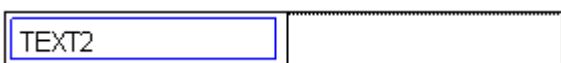
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

7. Click inside the blue box in the table and edit the text as required.
In the example below '**SampleID**' has been entered.

Sample ID		

8. Repeat steps 5–7 to enter text for the other output settings:

Sample ID	Abs451	Conc

NOTE: The text headings should be at the top of each column (rather than all rows in the first column) as the table will be repeated on the second row to obtain the information for all samples in the task.

9. Click on the ID data object in the Sample List to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Sample ID	Abs451	Conc
ID (Data1)		

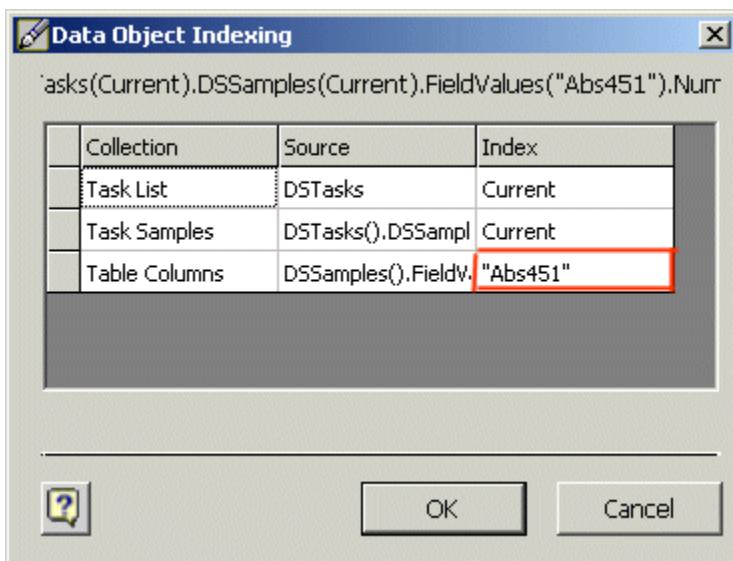
11. Select the Number data object in Table Columns (within the Sample List).
12. Position the mouse pointer in the empty field in the table below the Abs451 header and drag to the required size.

13. Repeat steps 11 and 12 and position the data object in the field beneath the Conc header.

The table should now look like:

Sample ID	Abs451	Conc
ID (Data1)	Number (Data2)	Number (Data3)

14. To display the correct values in the Abs451 an Conc columns you must use Indexing.
15. Right-click inside the data object beneath the Abs451 header and select Indexing.
The Data Object Indexing dialog is displayed.
16. Select the Index field for the Table columns:



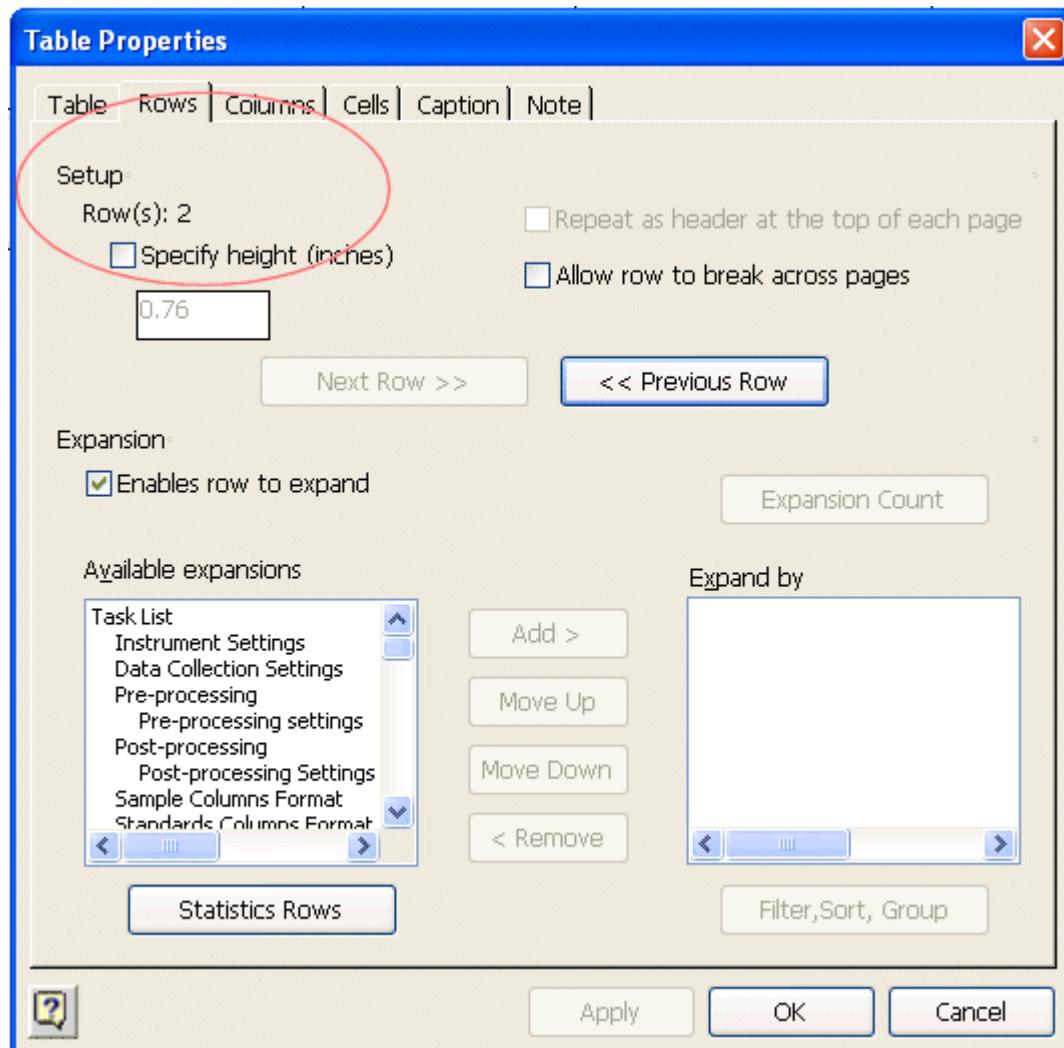
17. Within quotes (" "), enter the name of the column as it appears in the results table.
If the name is not exactly correct, the table will not populate with data.
18. Repeat the indexing (steps 13 to 15) for the Conc data object.

To obtain the ID, Abs451 and Conc values for all the samples in the task, you need to repeat the second row of the table.

19. Right-click on the table and select **Properties** from the context menu.
The Table Properties dialog is displayed.
20. Select the Rows tab.

21. Click Next Row.

Row 2 is now specified at the top of the dialog.



22. Ensure Enables row to expand is selected.

23. From the list of Available expansions, select **Samples**, and then click **Add**.

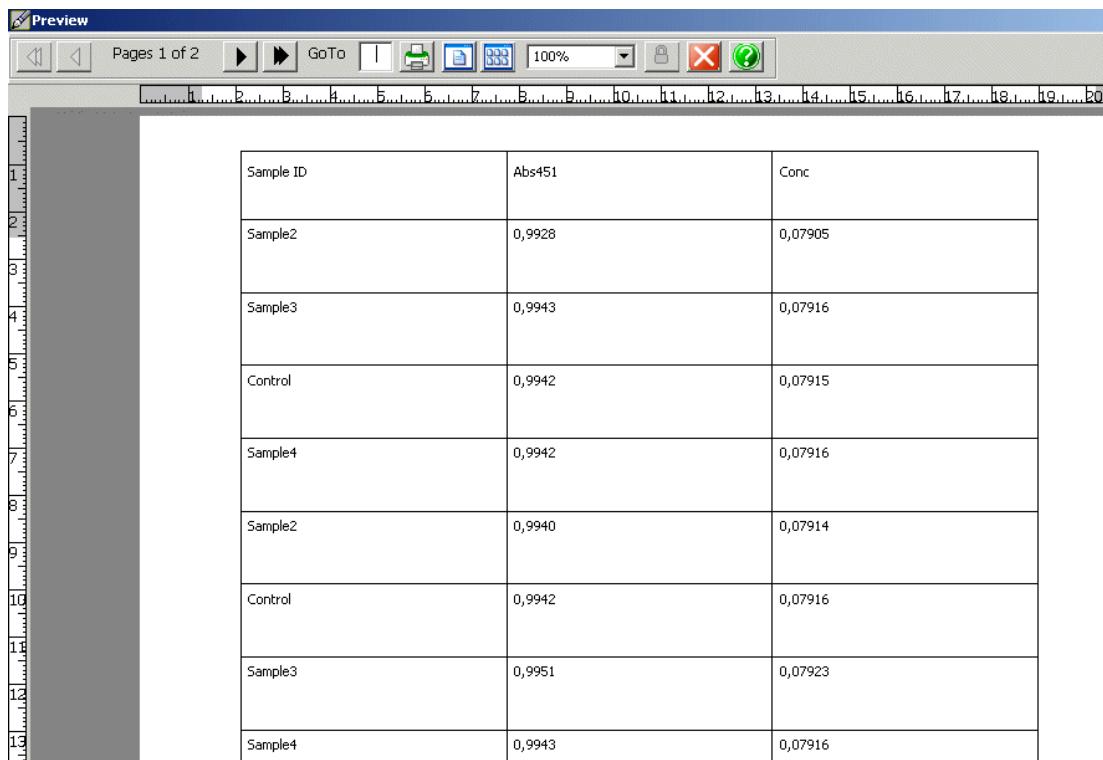
Samples moves from the Available expansions list to the Expand by list.

24. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

25. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

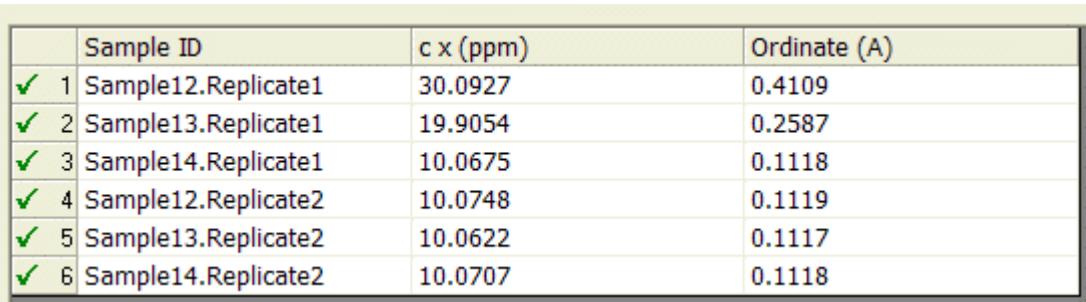


A screenshot of the WinLab software's Preview window. The window title is "Preview". At the top, there are buttons for navigating between pages (1-2), zooming (100%), and a help icon. Below the toolbar is a horizontal bar with page numbers from 1 to 20. The main area contains a table with 13 rows of data. The columns are labeled "Sample ID", "Abs451", and "Conc". The data is as follows:

Sample ID	Abs451	Conc
Sample2	0,9928	0,07905
Sample3	0,9943	0,07916
Control	0,9942	0,07915
Sample4	0,9942	0,07916
Sample2	0,9940	0,07914
Control	0,9942	0,07916
Sample3	0,9951	0,07923
Sample4	0,9943	0,07916

How do I create a table that contains all the information displayed in the Results Table?

The simplest way to do this is to use the Results Table data object (See How do I add the Results Table to my report template ? above). However, you can also create your own table (known as an expanding table). The example below shows how to create an expanding table for all the information in a ***Results Table***. The table will be created with 2 columns. The first is for the sample ID. The second will expand so that all columns in the Results Table are reported.



A screenshot of a table in the WinLab software. The table has 7 rows and 4 columns. The first column contains a checkbox followed by a number (1-6). The second column is labeled "Sample ID" and contains sample names. The third column is labeled "c x (ppm)" and the fourth is "Ordinate (A)".

	Sample ID	c x (ppm)	Ordinate (A)
✓ 1	Sample12.Replicate1	30.0927	0.4109
✓ 2	Sample13.Replicate1	19.9054	0.2587
✓ 3	Sample14.Replicate1	10.0675	0.1118
✓ 4	Sample12.Replicate2	10.0748	0.1119
✓ 5	Sample13.Replicate2	10.0622	0.1117
✓ 6	Sample14.Replicate2	10.0707	0.1118



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.
4. Select the Table tab and enter the number of columns and rows.
The size will depend on how many options you wish to include. The example below has 2 columns and 2 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.
In the example below '**SampleID**' has been entered.

Sample ID	

8. Select the **Title** data object beneath the Results Table Columns (within Task Samples in the Task List).
9. Click in the top cell of the second column:

Sample ID	Title (Data3)

10. Select the ID data object beneath Task Samples.

11. Click in the field beneath the Sample ID:

Sample ID	Title (Data3)
ID (Data1)	

The data object to be used in the final cell of the table depends on the data in the Results Table. If there is text as well as numbers, you should use the Text data object. If there is only numerical data, use the Number data object.

12. Select the **Number** data object beneath the Result Table Columns.

13. Click in the field beneath the Title data object:

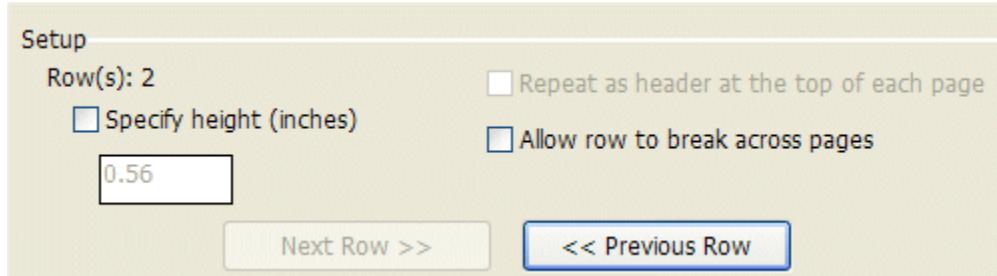
Sample ID	Title (Data3)
ID (Data1)	Text (Data2)

14. Right-click on the table and select **Properties** from the menu.

15. Select the Rows tab.

16. Click Next row.

Row 2 should be specified at the top of the Setup:



17. Select **Enables row to expand**, and then select **Task Samples** from the drop-down list.

18. Select the Columns tab.

19. Click **Next Column** so that Column 2 is specified at the top of the setup.

20. Select Enables column to expand, and then select Result Table Column.

21. Click **OK** to close the Properties dialog.

To view what will actually appear when the report is printed you need to print preview the report.

22. Click .

The report is displayed and the data objects are populated with the information.

NOTE: If the Sample ID contains a replicates or measurements extension, e.g. Sample1.replicate1, the extension is not reported in the table.

NOTE: You must repeat on the same type that is specified in the table. In the example above, the text data object from the Result Table Column is used in the table, and then the column is expanded by Result Table Column. If you were to repeat by Sample Table Column in this example, the table would not expand and populate correctly.

If you wish to create an expanding table for the sample table, use the procedure described above, but change the following:

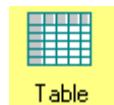
- In step 8, use the **Title** data object within the Sample Table Columns.
- In step 12 use the **Text** data object within Sample Table Columns.
- In step 20, select Enables column to expand by Sample Table Columns.

What is the Type data object?

The **Type** data object displays the information from the Type field in the Sample Table – that is, what type of sample is to be run. For example, it could be a Sample, Blank, Control, Replicate or Measurement. If you are using the **ID** data object, you may wish to use the **Type** data object as well so that you can tell what type each sample is. However, if you use the **Full ID** data object, the extension (for example Sample1.Replicate) explains what type of sample it is and so the Type field is not necessary.

How do I create a table of all the sample names and types in my report template?

This example explains how to use the **ID** and **Type** data objects together. This information can be also be displayed using the **Full ID** data object (see 'What is the difference between ID and Full ID ?' above).



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 2 rows.



5. Select the Text Block layout tool

The mouse pointer changes to  .

- Click the mouse inside a field of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree:

- Click inside the blue box in the table and edit the text as required.

In the example below 'ID' has been entered.

ID	

- Repeat steps 5–7 to enter text for 'Type':

ID	Type

NOTE: The text headings should be at the top of each column (rather than all rows in the first column) as the table will be repeated on the second row to obtain the information for all samples in the task.

- Click on the **ID** data object beneath Task Samples in the Task List.

The mouse pointer changes to  .

- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

ID	Type
ID (Data1)	

- Repeat steps 9 and 10 for the Type data object to complete your table:

ID	Type
ID (Data1)	Type (Data2)

To obtain the ID and Type for all the samples in the task, you need to repeat the second row of the table.

12. Right-click on the table and select **Properties** from the context menu.
The Table Properties dialog is displayed.
13. Select the Rows tab.
14. Click Next Row.
Row 2 is now specified at the top of the dialog.
15. Ensure Enables row to expand is selected.
16. From the list of Available expansions, select **Task Samples**, and then click **Add**.
Task Samples moves from the Available expansions list to the Expand by list.
17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

ID	Type
Sample1	Replicate1
Sample1	Replicate2
Sample2	Blank
Sample4	Control
Sample5	Replicate1
Sample5	Replicate2

Analyst

How do I include the analyst name and ID and the date the samples were analyzed in my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 3 rows.



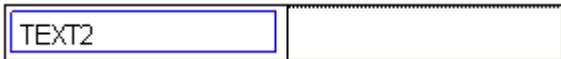
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below 'Analyst Name' has been entered.

Analyst Name	

8. Repeat steps 5–7 to enter text for Analyst ID and Date Analyzed:

Analyst Name	
Analyst ID	
Analysis Date	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Analyst Name	Name (Data1)
Analyst ID	
Analysis Date	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the analyst ID data object:

Analyst Name	Name (Data1)
Analyst ID	ID (Data2)
Analysis Date	

When using a date data object, we recommend that the data object is placed inside a text block. This ensures that all the date is displayed. If a text block is not used, and the field is not large enough to display the date, you will see ##### in the field.



12. Select the Text Block from the layout tools



The mouse pointer changes to

13. Drag the mouse on the report template to create a Text Block

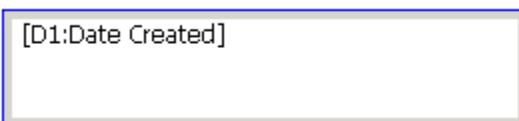


14. Click inside the Text Block and remove the default text.

15. Select the Date analyzed data object.

16. Click inside the Text Block.

The object is placed inside the Text Block



Analyst Name	Name (Data1)
Analyst ID	ID (Data2)
Analysis Date	[D3:Date Analyzed]

To view what will actually appear when the report is printed you need to print preview the report.

17. Click

Analyst Name	tracey
Analyst ID	tracey
Analysis Date	04/26/2004 04:23:53 PM BST

Spectra

How do I display the raw spectrum of each sample on a separate graph?

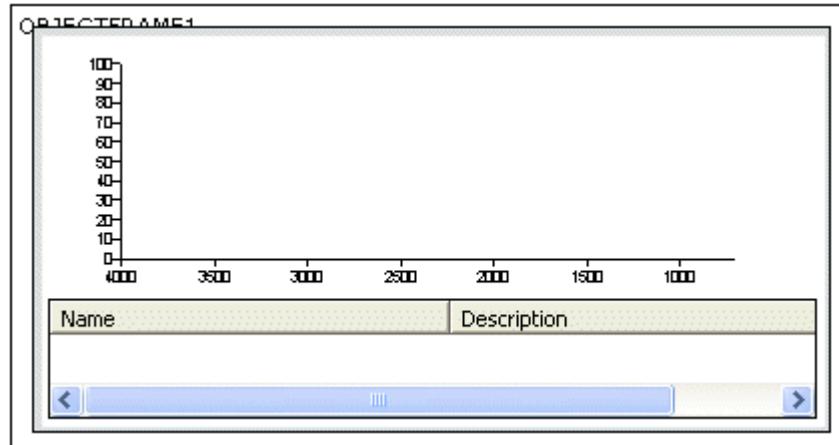
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



To display the raw spectrum for each sample on a separate graph, a section must be created.

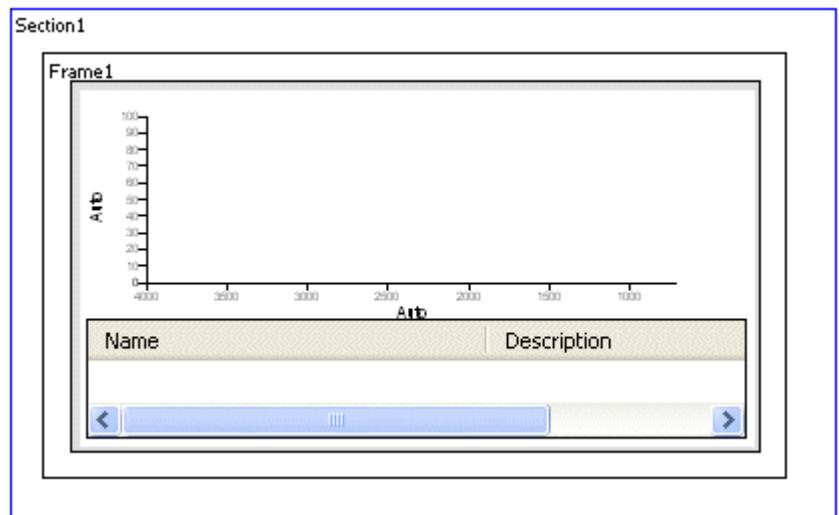


3. From the Layout Tools list, select .



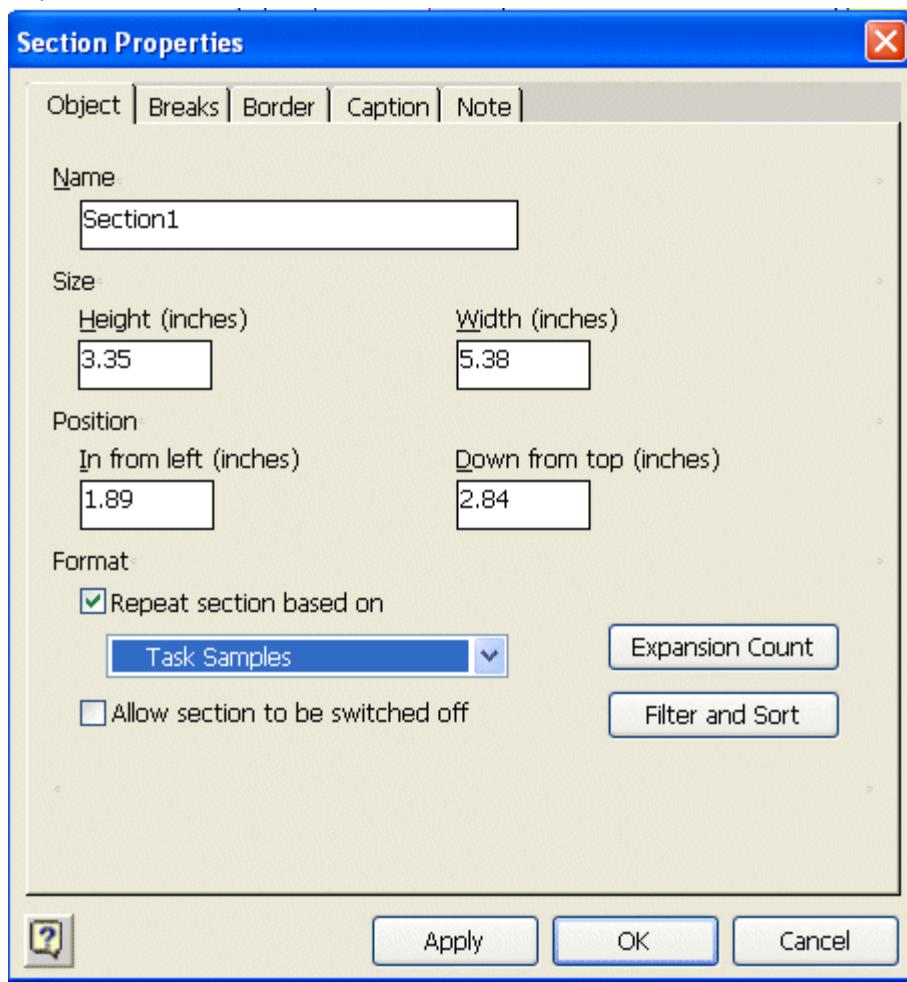
The mouse pointer changes to .

4. Drag the mouse around the object frame to create a section containing the spectrum:



The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.



6. Click **OK**.

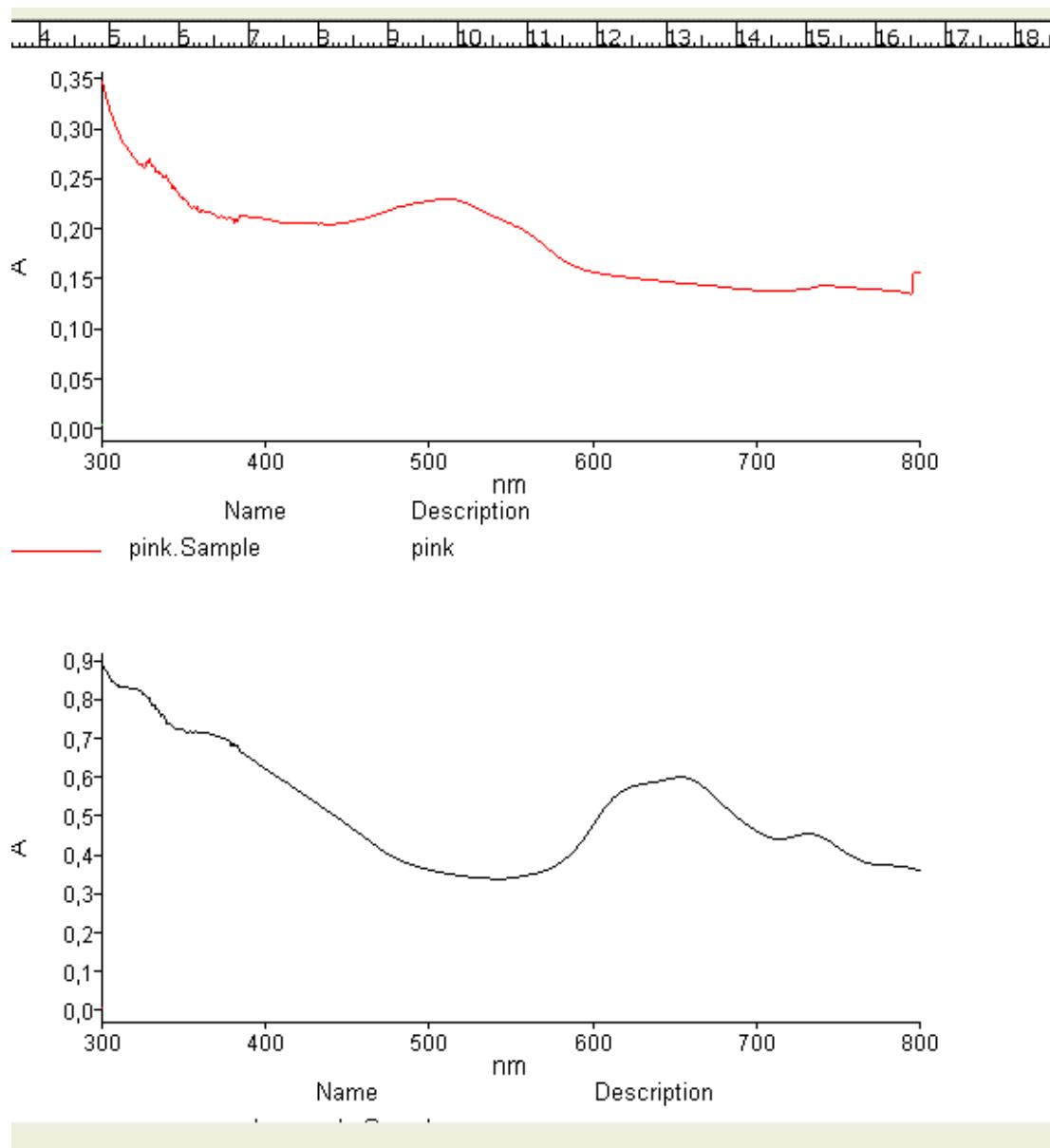


The section is added to the tree:

To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

An example is shown below:



How do I overlay the raw or processed spectra of each sample on the same graph?

The example below uses the **Raw Spectrum** data object. To overlay all processed spectra on one graph, follow the instructions below but use the **Processed Spectrum** data object.

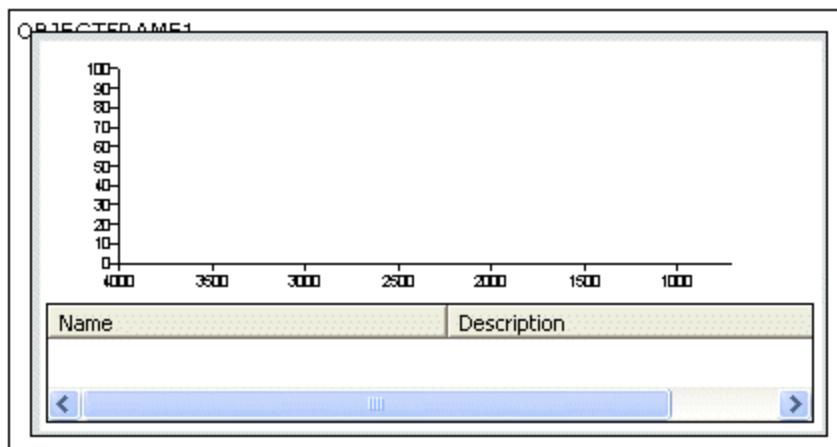
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to

2. Drag the mouse on the report template.

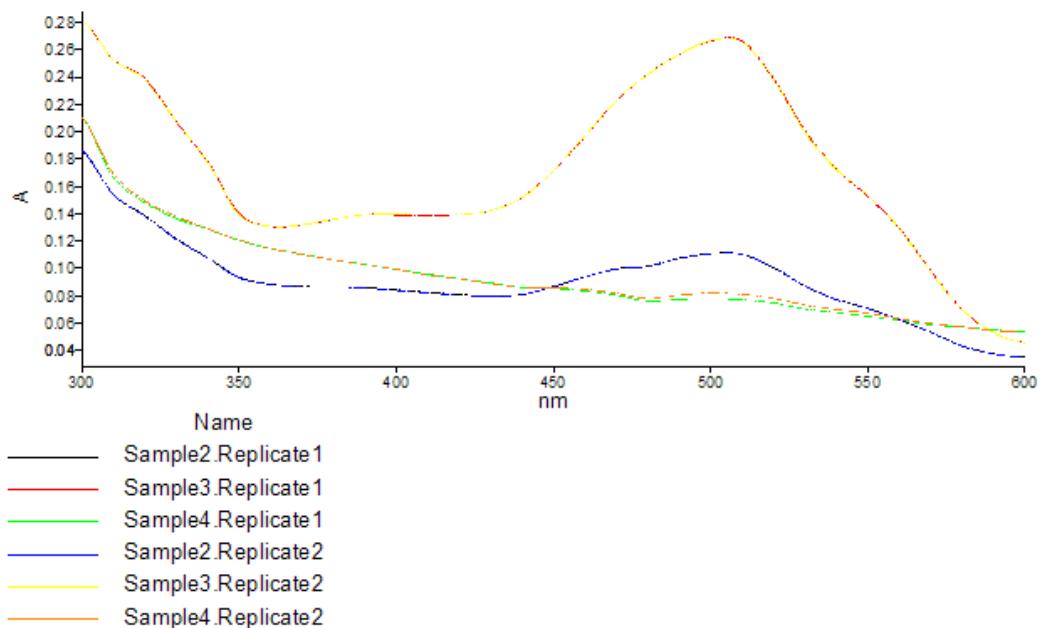
An object frame with an empty graph is displayed:



3. Right-click on the frame and select **Properties**.
4. Select the Sequence tab.
5. Select **Repeat based on**, and then select **Task Samples** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

6. Click .



How do I display the processed spectrum of each sample?

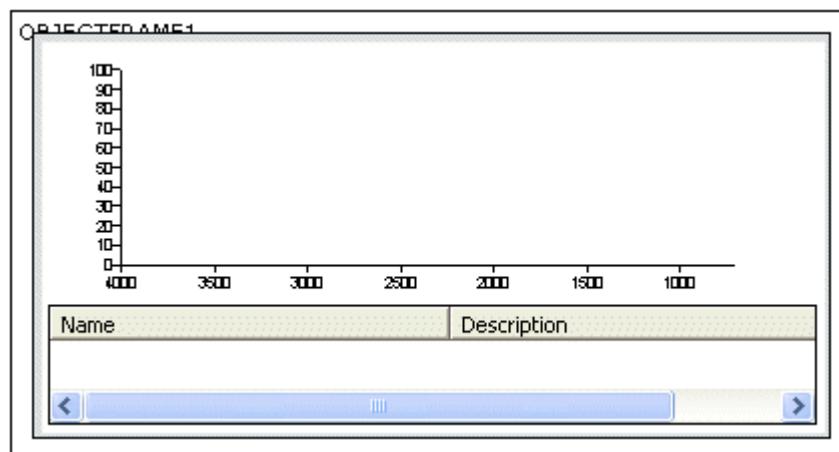
1. Select the **Processed Spectrum** data object.



The mouse pointer changes to

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



To display the processed spectrum for each sample, a section must be created.

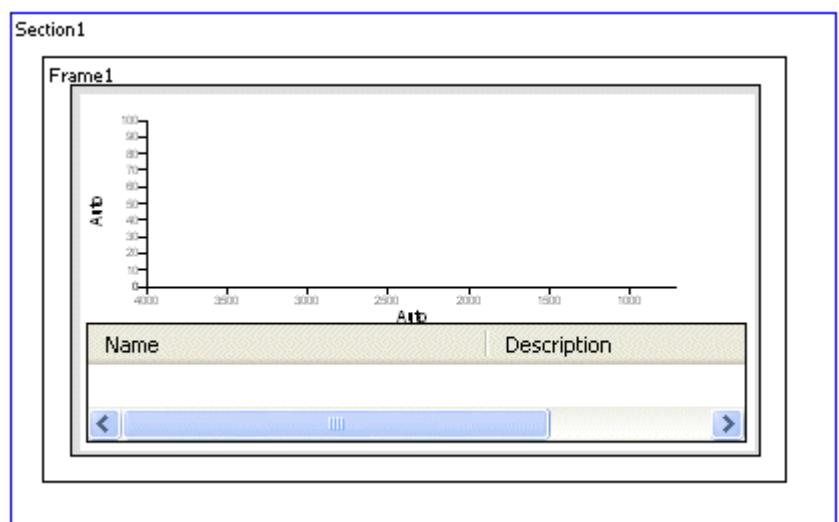


3. From the Layout Tools list, select



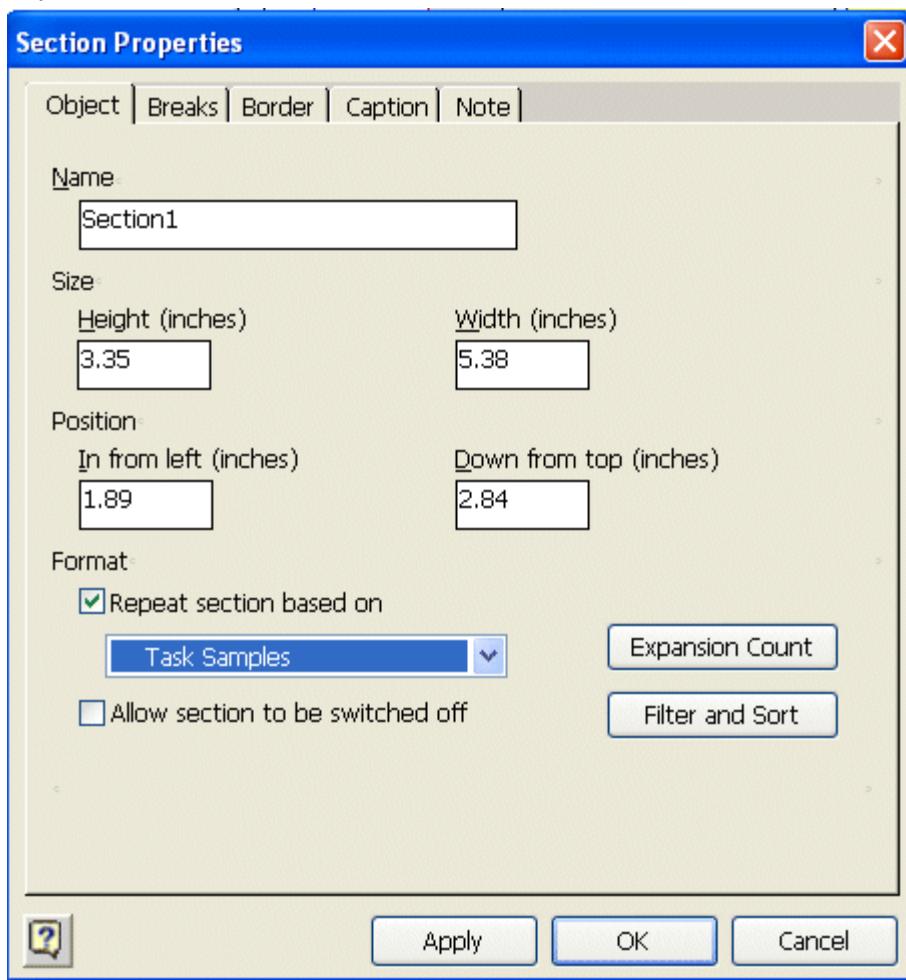
The mouse pointer changes to

4. Drag the mouse around the object frame to create a section containing the spectrum:

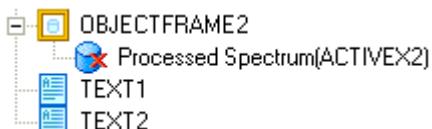


The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.



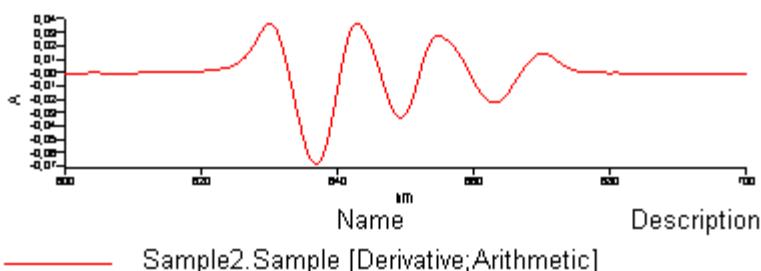
6. Click **OK**.



The section is added to the tree:

To view what will actually appear when the report is printed you need to print preview the report.

7. Click .



How do I overlay the raw and processed spectra of each sample?

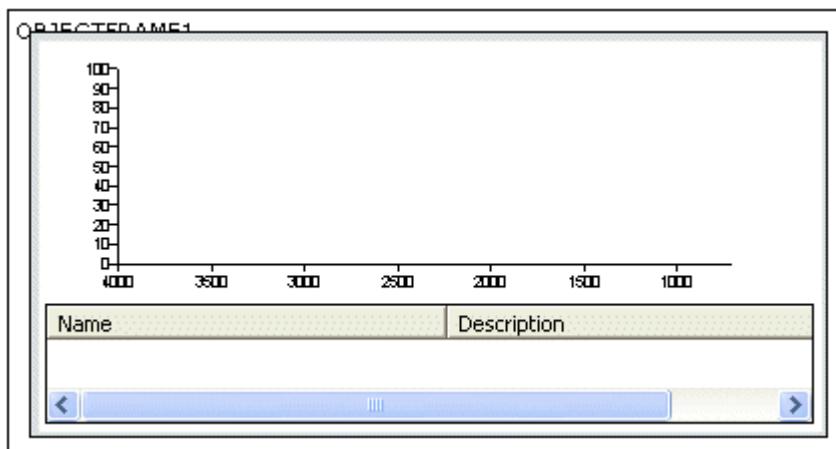
1. Select the **Raw Spectrum** data object.



The mouse pointer changes to .

2. Drag the mouse on the report template.

An object frame with an empty graph is displayed:



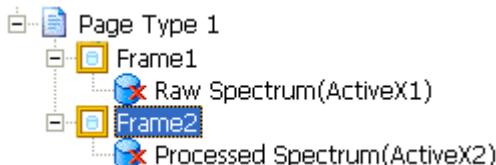
3. Select the **Processed Spectrum** data object.



The mouse pointer changes to .

4. Drag the mouse exactly over the raw spectrum object.

The processed spectrum frame is added to the tree:



To display the overlaid spectra for each sample (each sample on a separate graph), a section must be created.

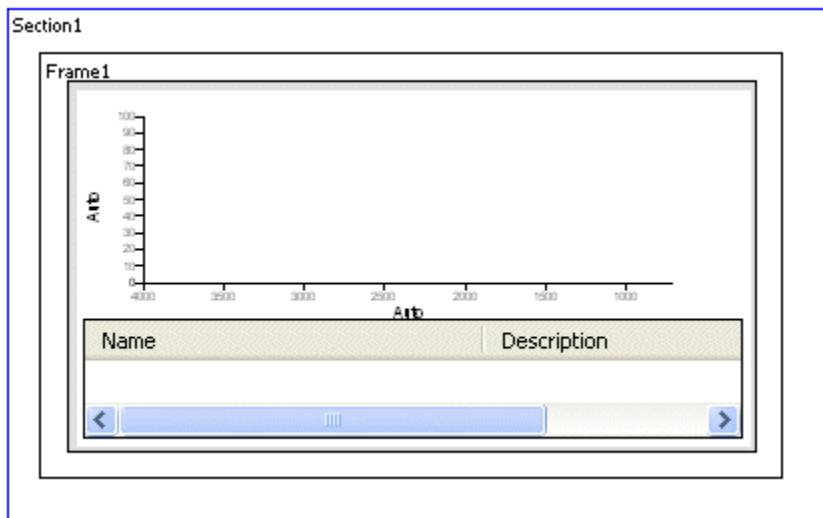


5. From the Layout Tools list, select .



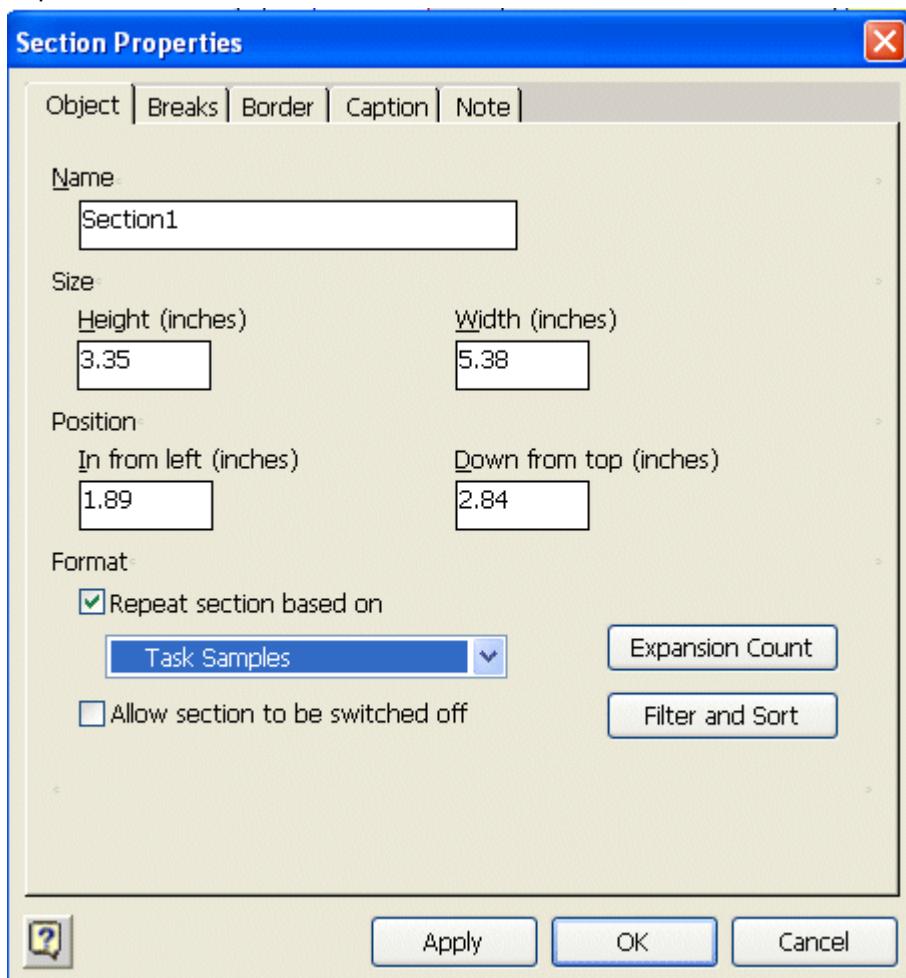
The mouse pointer changes to .

6. Drag the mouse around the object frame to create a section containing the spectrum:



The Section Properties dialog is displayed.

7. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.



8. Click **OK**.

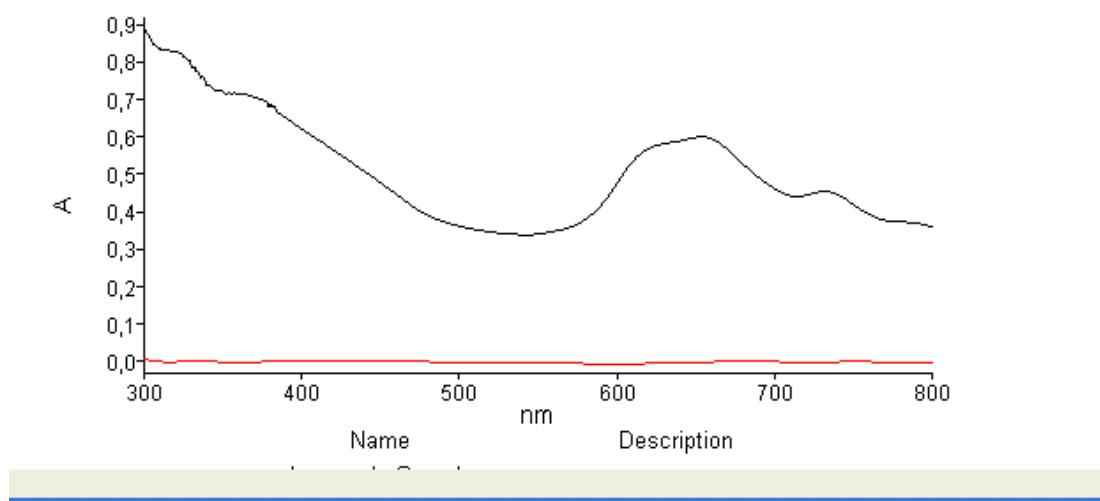
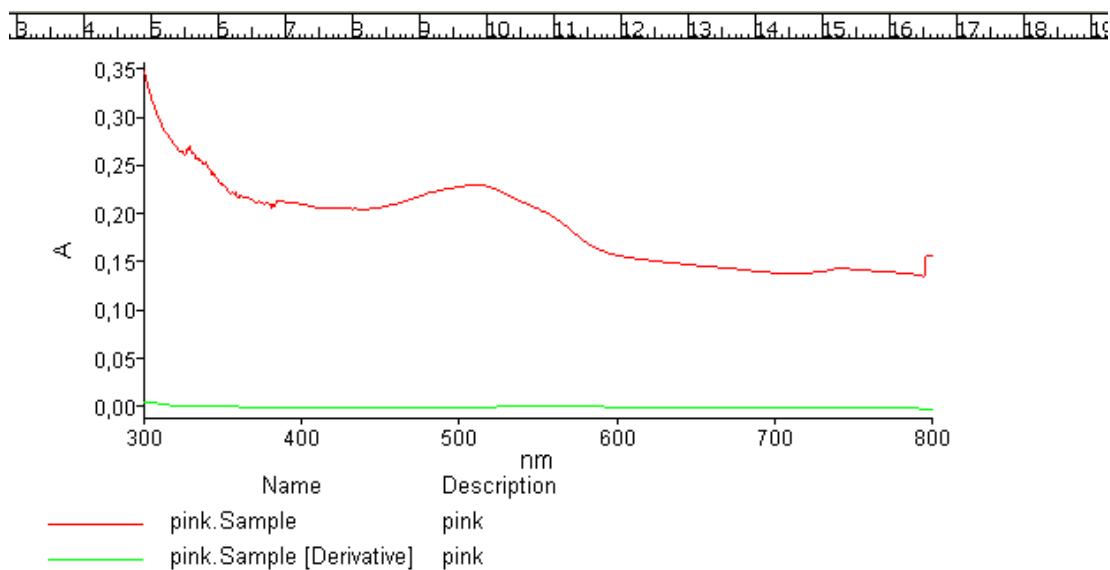


The section is added to the tree:

To view what will actually appear when the report is printed you need to print preview the report.

9. Click .

An example is shown below:



Wavelength Data

How do I add a Wavelength Table to my report template?

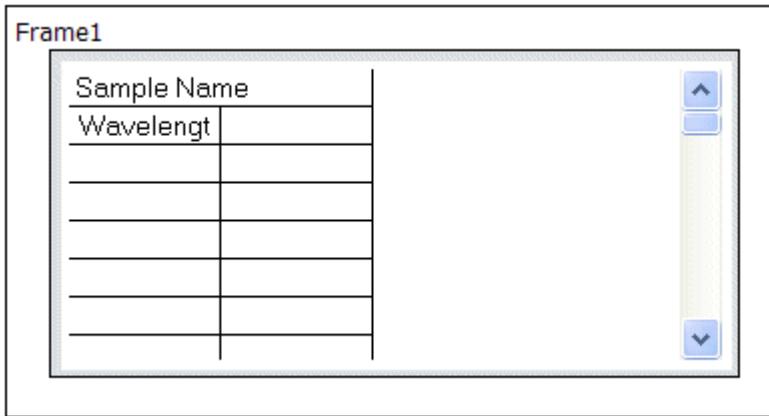
This data object creates a table that lists the absorbances for each of the wavelengths specified in the task. By default only the data relating to the first samples is displayed. You must use a section to display the information for all samples.

1. Select the **Wavelength Table** data object.



The mouse pointer changes to .

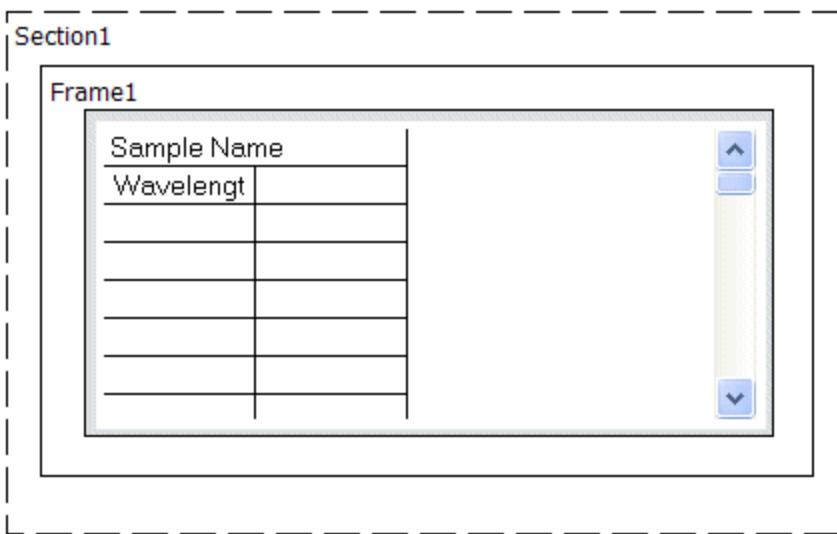
2. Click and drag the mouse to create a frame



The size of the Frame determines the size of the table in the printed report.



3. Select the Section layout tool
4. Create a section around the frame.



The Section Properties dialog is displayed.

5. Select **Repeat section based on**, and then select **Wavelength Data** from the drop-down list.
6. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

7. Click .

The report is displayed. This is how the report will appear when printed:

Sample1.Sample	
Wavelength (nm)	Value (A)
175	0.94224
865	0.720284
3300	10

Sample2.Sample	
Wavelength (nm)	Value (A)
175	0.231817
865	1.35717
3300	10

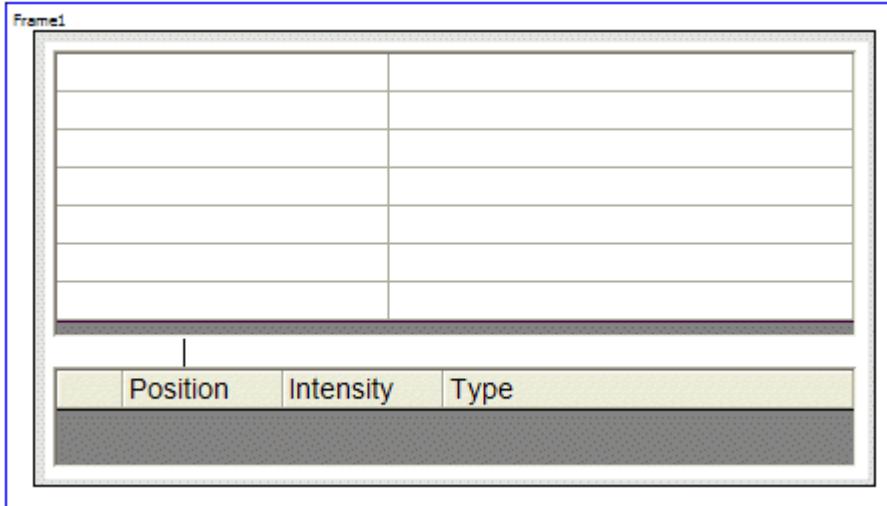
Sample3.Sample	
Wavelength (nm)	Value (A)
175	-0.00440505
865	1.35774
3300	10

Peak Table

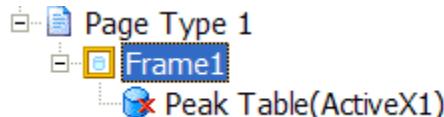
How do I include a peak table in my report template?

A peak table must have been setup as part of the processing for it to be available within the report template.

1. Select the **Peak Table** data object within the Sample List.
2. Click the mouse on the report template and drag to create the object:



The size of the frame will determine the size of the peak table in the report. If the frame is too small, not all of the results will be displayed.

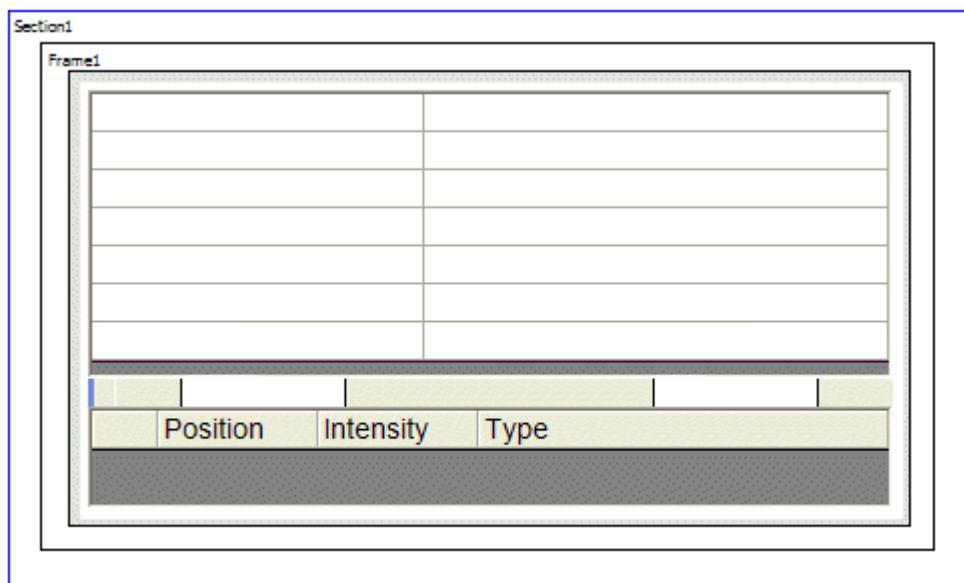


The tree is updated:

To display a peak table for each sample you need to create a section.



3. Select the Section Layout Tool **Section**.
4. Click and drag the mouse over the Peak table frame:



The Section Properties dialog is displayed.

5. Select **Repeat section based on**, and then select **Task Samples** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

6. Click .

This is how the report will appear when printed:

Sample ID	Sample5.Sample		
Description			
Threshold	0.001		
Range start (nm)	800		
Range end (nm)	300		
Find	Peaks		
Display	List by position		
	Position (nm)	Intensity (A)	Type
1	798.3	0.2448	Peak
2	796.02	0.245	Peak
3	793.96	0.2417	Peak
4	791.1	0.2393	Peak
5	789	0.2383	Peak
6	786.8	0.2352	Peak
7	783.99	0.2361	Peak
8	782.03	0.237	Peak
9	779.07	0.2337	Peak
10	777.06	0.2342	Peak
11	774.95	0.2398	Peak

Sample ID	Sample6.Sample		
Description			
Threshold	0.001		
Range start (nm)	800		
Range end (nm)	300		
Find	Peaks		
Display	List by position		
	Position (nm)	Intensity (A)	Type
1	776.25	0.03531	Peak
2	571.56	0.03371	Peak

NOTE: Peak Tables cannot wrap over pages. If you have created the object the size of a page and not all the information is displayed, we recommend that you adjust the settings of the Peak Table within the Processing page of the Task (for example, increasing the Threshold).

Table Columns

How do I create a table that contains all the information displayed in the Sample Table and Results Table (excluding mandatory columns)?

You can create a table (known as an expanding table) using the Table columns data objects. This will display all columns from the sample table and results table excluding mandatory columns (such as Type).

The example below shows how to create an expanding table . The table will be created with 2 columns. The first is for the sample ID. The second will expand so that all columns in the both the Sample and Results Table are reported.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 2 rows.



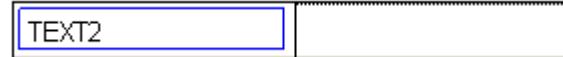
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**SampleID**' has been entered.

Sample ID	

8. Select the **Title** data object beneath the Table Columns (within the Sample List).
9. Click in the top cell of the second column.

Sample ID	Title (Data3)

10. Select the **ID** or **Full ID** data object from the Sample List.

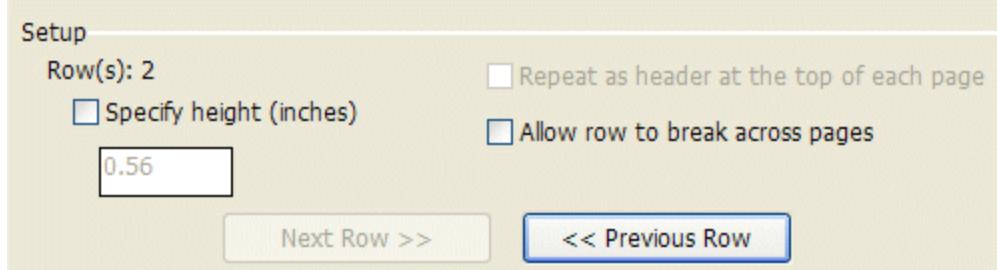
11. Click in the field beneath the Sample ID.

Sample ID	Title (Data3)
ID (Data1)	

The data object to be used in the final cell of the table depends on the data in the Sample and Results Table. If there is text as well as numbers, you should use the Text data object. If there is only numerical data, use the Number data object.

12. Select the **Number** data object beneath the Table Columns.
13. Click in the field beneath the Title data object.
14. Right-click on the table and select **Properties** from the menu.
15. Select the Rows tab.
16. Click Next row.

Row 2 should be specified at the top of the Setup



17. Select **Enables row to expand**, and then select **Task Samples** from the drop-down list.
18. Select the Columns tab.
19. Click **Next Column** so that Column 2 is specified at the top of the setup.
20. Select **Enables column to expand**, and then select **Table Columns** (beneath Sample List).
21. Click **OK** to close the Properties dialog.

To view what will actually appear when the report is printed you need to print preview the report.

22. Click .

The report is displayed and the data objects are populated with the information.

NOTE: If the Sample ID contains a replicates or measurements extension, e.g. Sample1.replicate1, the extension is not reported in the table if ID is used. Use Full ID to include the extension.

Sample Event Log

How do I include a list of all sample events for all samples in my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The example below has 2 columns and 2 rows.



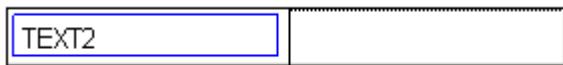
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**Sample**' has been entered.

Sample	

8. Repeat steps 5–7 to enter text for the Sample Event:

Sample	Event

9. Select the **Full ID** data object (from below Task Samples in the Task List).



The mouse pointer changes to .

10. Click in the field below 'Sample'.

The table now looks something like:

Sample	Event
Full ID (Data1)	

11. Repeat steps 9 and 10 for the Sample Event Log Event data object:

Sample	Event
Full ID (Data1)	Event (Data2)

To obtain all sample event log information for all samples, you need to repeat on the rows (to get all samples), and on the second column (to get all events for each sample).

12. Right-click on the table and select **Properties** from the context menu.

The Table Properties dialog is displayed.

13. Select the Rows tab.

14. Click Next Row.

Row 2 is now specified at the top of the dialog.

15. Ensure Enables row to expand is selected.

16. From the list of Available expansions, select **Task Samples**, and then click **Add**.

Task Samples moves from the Available expansions list to the Expand by list.

17. Select the Columns tab.

18. Click Next Column.

Column 2 is now specified at the top of the dialog.

19. Ensure Enables column to expand is selected.

20. From the list of Available expansions, select **Sample Event Log**, and then click **Add**.

21. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

22. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Sample	Event	Event
Sample7.Sample.Cycle1	Saved	Comment Added
Sample7.Sample.Cycle2	Saved	
Sample7.Sample.Cycle3	Saved	
Sample13.Sample.Cycle1	Saved	
Sample13.Sample.Cycle2	Saved	
Sample13.Sample.Cycle3	Saved	
Sample14.Sample.Cycle1	Saved	Excluded

NOTE: Where cycles are used, any comment added to a sample in the Sample Table only appears against the first cycle in the report table (as shown above).

How do I include the Sample Event Log information (event, time, reason/comment, name and ID) for each sample in my report template?

This example shows how to create a table of sample event log information. Each sample has a separate table.



1. Select the Table Layout tool .



The mouse pointer changes to .

- Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

- To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.

- Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 5 rows. The columns will be set to expand for all the events recorded for a particular sample.



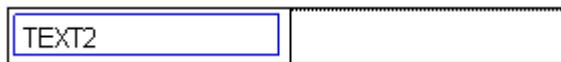
- Select the Text Block layout tool



The mouse pointer changes to 

- Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

- Click inside the blue box in the table and edit the text as required.

In the example below '**Sample**' has been entered.

Sample	

- Repeat steps 5–7 to enter text for the other sample event log settings:

Sample	
Event	
Time	
Reason / Comment	
Name	
ID	

9. Click on the **Full ID** data object (Task Samples) in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Sample	Full ID (Data6)
Event	
Time	
Reason / Comment	
Name	
ID	

11. Repeat steps 9 and 10 for the other Sample Event Log data objects except **Time**.

12. In the Time field add a text block.

13. Delete the default text.

14. Select the **Time** data object and then click inside the text block.

The Time data object is placed inside the text block.

This enables the time to text wrap within the text block. Otherwise the cell would need to be very long to display all of the date.

Sample	Full ID (Data6)
Event	Event (Data1)
Time	[D5:Time]
Reason / Comment	Reason/Comment (Data2)
Name	Name (Data3)
ID	ID (Data4)

To include all sample event log information for the sample, the second column must be set to repeat on the Sample Event Log.

15. Select the table and right-click.
16. From the menu select **Properties**.
17. Select the columns tab.
18. Click Next Column.
Column 2 is displayed at the top of the dialog.
19. Ensure Enables column to expand is selected.
20. Select **Sample Event Log** from the drop-down list.
21. Click **OK**.

To display a table for every sample, a section must be created around the table.



22. Select the Section Layout tool



The mouse pointer changes to

23. Drag the mouse around the table.
A section is created around the table:

Section1	
Sample	Full ID (Data6)
Event	Event (Data1)
Time	[D5:Time]
Reason / Comment	Reason/Comment (Data2)
Name	Name (Data3)
ID	ID (Data4)

and the Section Properties dialog is displayed.

24. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.
25. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

26. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Sample	Sample5.Sample	Sample5.Sample
Event	Comment Added	Comment Added
Time	04/29/2004 03:11:22 PM GMT Daylight Time	04/29/2004 03:11:40 PM GMT Daylight Time
Reason / Comment		first sample of new batch
Name	pksidhu	pksidhu
ID	pks1	pks1

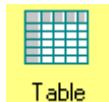
Sample	Sample6.Sample
Event	Excluded
Time	04/29/2004 03:11:58 PM GMT Daylight Time
Reason / Comment	incorrect sample
Name	pksidhu
ID	pks1

In this example, there is an event log for sample 5 and an event log for sample 6. Sample 5 has 2 entries in the sample event log so the table has expanded to show both of these. For sample 6 there was only one entry.

Corrections

How do I include correction information for each sample in my report template?

In this example a table of sample name, correction name and corrections samples ID will be created for a sample. In addition a graph containing all the correction spectra for the sample will be created. All this information will be put within a section which will then be repeated on to get all the information for all samples in the task.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree

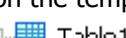


Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 3 columns and 2 rows.

The correction samples ID columns will be set to expand for all correction IDs for a particular name. For example, the 100% and 0% baseline corrections (Corrections Name) has 2 Corrections IDs (spectra) associated with it – 100% or 0 Absorbance Baseline and 0% or Blocked Beam Baseline.



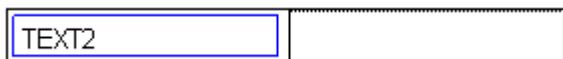
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.
 In the example below '**Correction Name**' has been entered.

Correction Name		

8. Repeat steps 5–7 to enter text for the Correction ID and Full Sample ID.

Correction Name	Correction ID	Full Sample ID

9. Click on the **Corrections Name** in the Data Object list to select it.



The mouse pointer changes to

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Correction Name	Correction ID	Full Sample ID
Name (Data1)		

11. Repeat steps 9 and 10 for the Correction Samples ID and the Full ID.
 The Full ID data object is further up the Task List, beneath Task Samples.

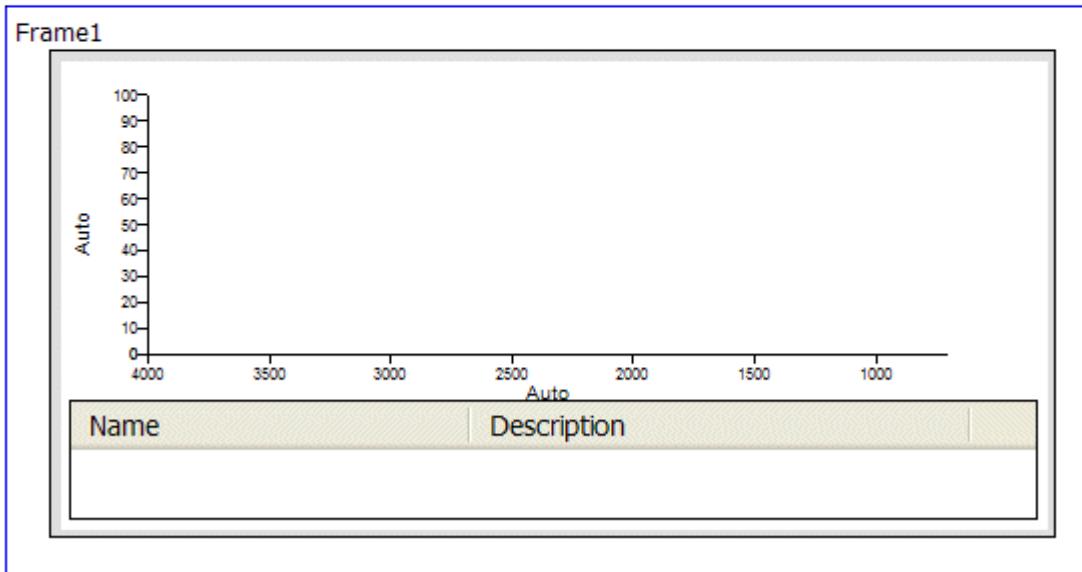
The table now looks like:

Correction Name	Correction ID	Full Sample ID
Name (Data1)	ID (Data2)	Full ID (Data3)

To show all the Correction IDs for a particular correction you need to expand column 2.

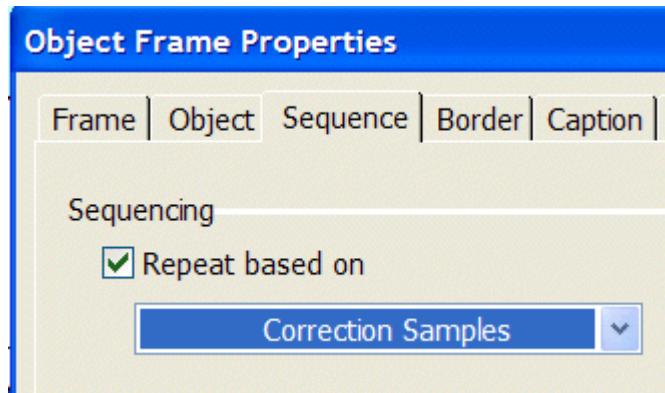
12. Select the **Correction ID** data object in the table (Data 2 in this example) and right-click.
13. From the menu select **Properties**.
14. Select the columns tab.
15. Click Next Column.
Column 2 is displayed at the top of the dialog.
16. Ensure Enables column to expand is selected.
17. Select **Correction Samples** from the drop-down list.
18. Click **OK**.
19. Select the Corrections Samples Raw Spectrum data object.
20. Click and drag to there required size.
The size of the object determines the size of the graph in the report.

Correction Name	Correction ID	Full Sample ID
Name (Data1)	ID (Data2)	Full ID (Data3)



To display all the correction spectra for a particular correction on one graph, you must repeat on the object frame based on correction samples.

21. Right-click on the object frame and select **Properties**.
The Object Frame Properties dialog is displayed.
22. Select the Sequence tab.
23. Ensure **Repeat based on** is selected, and then select **Correction Samples** from the drop-down list.



24. Click **OK**.

To display a table and graph for every sample, a section must be created around the table and graph.

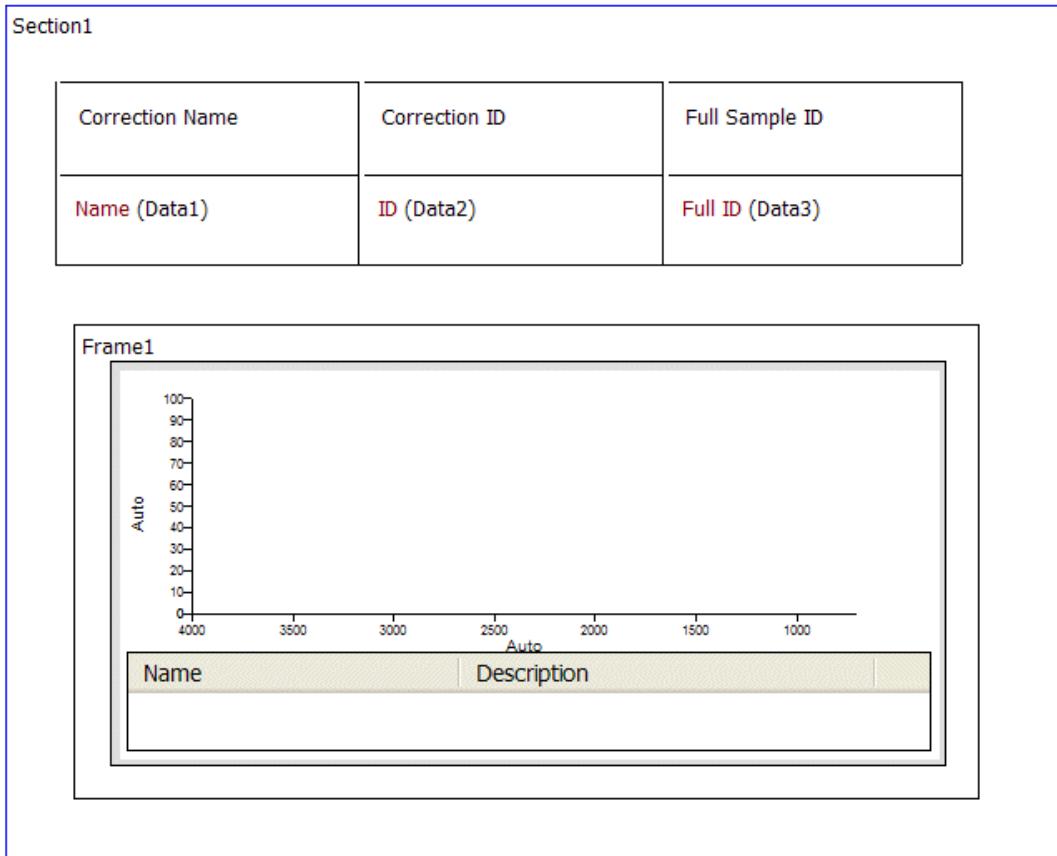


25. Select the Section Layout tool



The mouse pointer changes to

26. Drag the mouse around the table and graph:



and the Section Properties dialog is displayed.

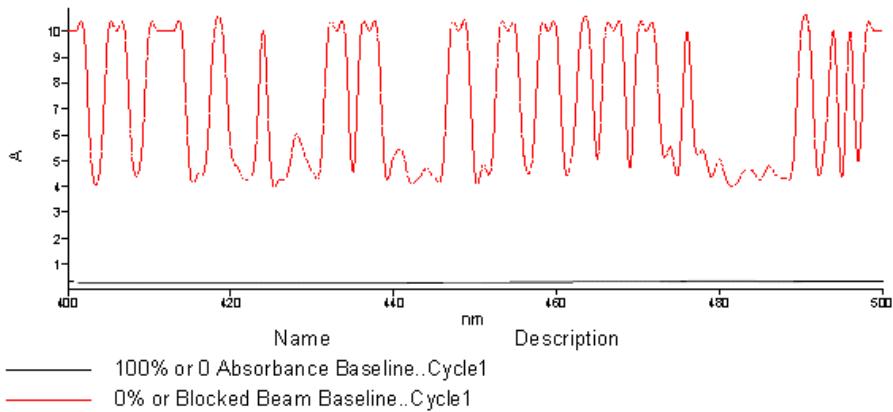
27. Ensure **Repeat section based on** is selected, and then select **Task Samples** from the drop-down list.
28. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

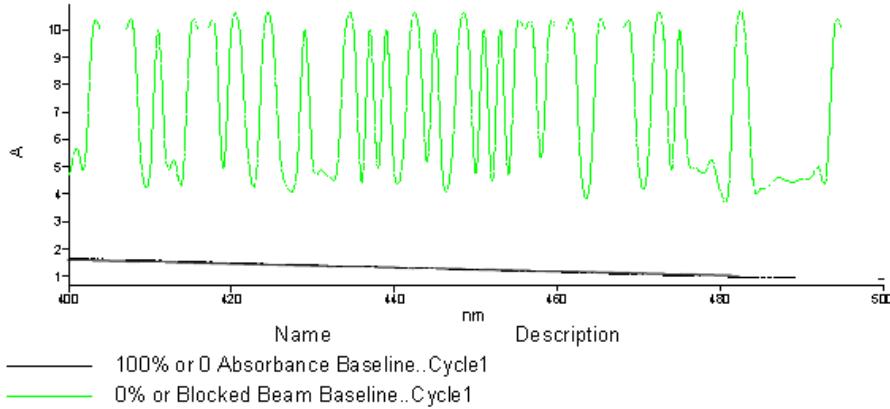
29. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Correction Name	Correction ID	Correction ID	Full Sample ID
100% and 0% baseline corrections	100% or 0 Absorbance Baseline	0% or Blocked Beam Baseline	Sample44.Sample



Correction Name	Correction ID	Correction ID	Full Sample ID
100% and 0% baseline corrections	100% or 0 Absorbance Baseline	0% or Blocked Beam Baseline	Sample45.Sample

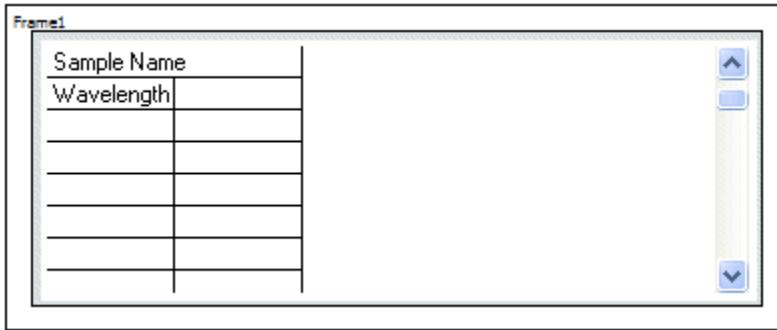


The table could also include a column for the Date Analyzed data object if required. Follow the steps above but create a table with an additional column. Ensure that the Correction ID column is still set to expand.

How do I include a wavelength table for all corrections for each sample in my report template?

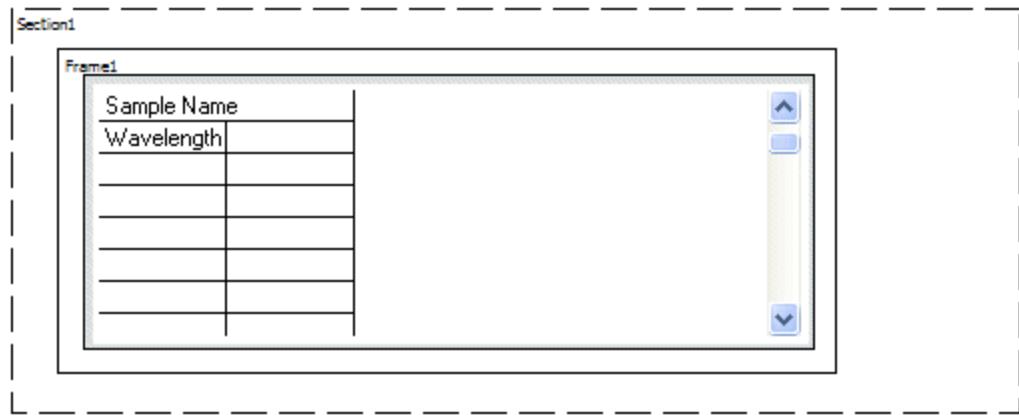
In this example the report template will include the name of the sample, the name of the correction and the wavelength table for the correction (for all samples).

1. Select the **Wavelength Table** data object (beneath Corrections) within the Task List.
2. Click the mouse on the report template and drag to create the object:



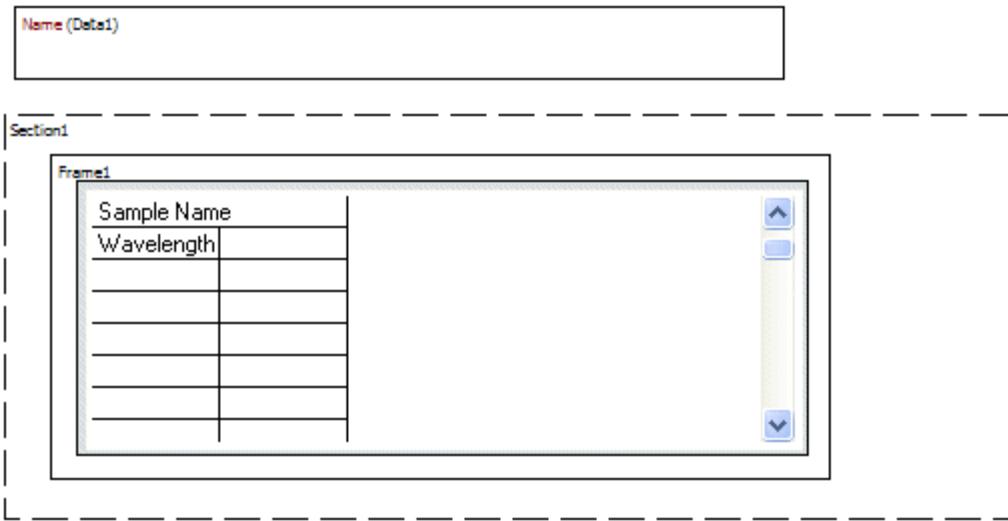
To display a table for all correction samples for a particular correction (for example 100% or 0 Absorbance Baseline and 0% or Blocked beam baseline for 100% and 0% baseline corrections), you need to create a section around the Wavelength Table.

3. Select the Section layout tool.
4. Click and draw a section around the Wavelength Table:

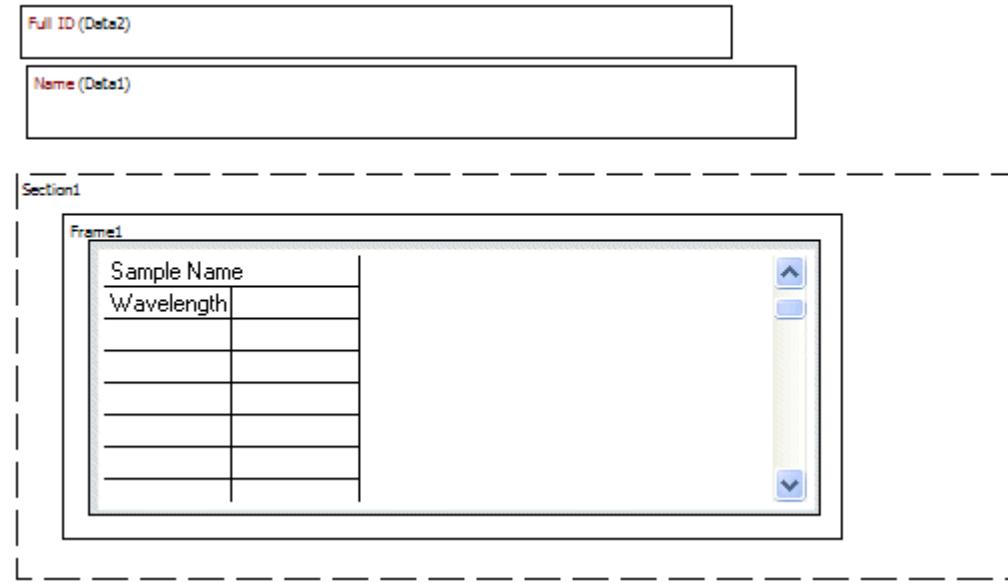


The Section Properties dialog is displayed.

5. Ensure **Repeat section based on** is selected, and select **Correction Samples** from the drop-down list.
6. Click **OK**.
7. Select the **Corrections Name** data object.
8. Place the object above the section.



9. Select the **Full ID** data object from the Sample List.
10. Place the object above the Corrections Name data object.



To obtain the information for all samples, a section is needed around all this information.

11. Select the Section layout tool.
12. Click and draw a section around all the objects placed on the template in the above steps.



The Section Properties dialog is displayed.

13. Ensure **Repeat section based on** is selected, and select **Task Samples** from the drop-down list.
14. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

15. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

PAGlassRM-19.Sample

100% and 0% baseline corrections

100% or 0 Absorbance Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	-0.0516375
546	-0.051752
635	-0.0549886
1700	-0.0143021
2300	-0.0175179

0% or Blocked Beam Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	10
546	10
635	10
1700	10
2300	2.53566

PAGlassRM-20.Sample

100% and 0% baseline corrections

100% or 0 Absorbance Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	-0.0516375
546	-0.051752
635	-0.0549886
1700	-0.0143021
2300	-0.0175179

0% or Blocked Beam Baseline..Cycle1	
Wavelength (nm)	Value (A)
440	10
546	10
635	10
1700	10
2300	2.53566

Data Points

How do I include a table of data points and the associated ordinate values in my report template?

The Raw Points and Processed Points data objects allow you to display abscissa and ordinate value for raw and processed data. You can display all points, every 2 points, every 5 points and/or every 10 points for raw and processed data. Each of the objects has associated abscissa and ordinate objects.

These data objects can be used for all data collection types – scan, timedrive, polarization scan, and wavelength programming.

The example below shows how to create a table of every 5th raw data point and the associated ordinate value. For all other raw point / processed points, follow the steps below and substitute the required data objects.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree



Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The example below has 2 columns and 2 rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



Table1



Text2

and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.
In the example below '**Abscissa (every 5 points)**' has been entered.

Abscissa (every 5 points)	

8. Repeat steps 5–7 to enter text for **Ordinate**:

Abscissa (every 5 points)	Ordinate

9. Click on the Raw Points Every 5 Abscissa data object.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field below **Abscissa (every 5 points)**, and click.

The table now looks something like:

Abscissa (every 5 points)	Ordinate
Abscissa (Data1)	

11. Repeat steps 9 and 10 for the Ordinate data object.

Abscissa (every 5 points)	Ordinate
Abscissa (Data1)	Ordinate (Data2)

To display every 5th data point, row 2 must be set to expand.

12. Select the table and right-click.
13. From the menu select **Properties**.
14. Select the rows tab.
15. Click Next Row.
Row 2 is displayed at the top of the dialog.
16. Ensure Enables row to expand is selected.

17. Select **Raw Points every 5** from the drop-down list.

NOTE: The option selected from the drop-down list must correspond to the raw/processed abscissa/ordinate objects in the table. In this example Raw points every 5 Abscissa and Ordinate data objects have been used so **Raw Points every 5** must be selected for the expansion.

18. Click **OK**.

To display a table for every sample, a section must be created around the table.

NOTE: You may wish to include another column in your table for Sample ID so that you can identify which sample the data points relate to. (This is not shown here)



19. Select the Section Layout tool



The mouse pointer changes to

20. Drag the mouse around the table.

A section is created around the table -

Section1	
Abscissa (every 5 points)	Ordinate
Abscissa (Data1)	Ordinate (Data2)

and the Section Properties dialog is displayed.

21. Ensure **Repeat section based on** is selected, and then select **Sample List** from the drop-down list.

22. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

23. Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Abscissa (every 5 points)	Ordinate
3300	0.638283140485891
3295	0.643445481285867
3290	0.632384085238443
3285	0.632167030853172
3280	0.631976126218767
3275	0.627198864356991
3270	0.62439913507893
3265	0.621620903689229
3260	0.620542520296119
3255	0.618989191884605
3250	0.616728690737875
3245	0.612557868365017
3240	0.611219488747158
3235	0.608352278702349

How do I obtain the ordinate value at a specific abscissa value for all samples and include this in my report?

Rather than obtain all raw or processed points or every 2, 5 or 10 points (see How do I include a table of data points and the associated ordinate values in my report template ?), you can display the ordinate at a single specified value. This example shows how to create a table to report the ordinate value at 445 nm for all samples.



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.
The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The example below has 2 columns and 2 rows.



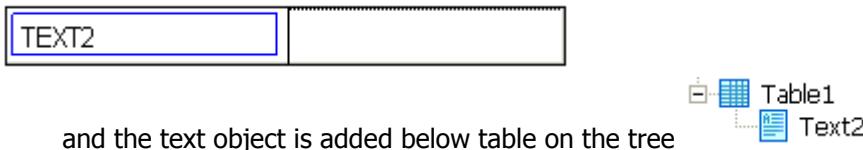
5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



7. Click inside the blue box in the table and edit the text as required.

In the example below 'Sample' has been entered.

Sample	

8. Repeat steps 5–7 to enter text for **Ordinate value at 445nm**:

Sample	Ordinate value at 445 nm

- Select the Sample List Full ID data object , and click in the second row of the first column of the table.

Sample	Ordinate value at 445 nm
Full ID (Data1)	

- Click on the Sample List Raw Points Ordinate data object.

The mouse pointer changes to .

- Position the mouse pointer in the empty field and click.
The table now looks something like:

Sample	Ordinate value at 445 nm
Full ID (Data1)	Ordinate (Data3)

To display the Ordinate value at 445 nm you need to use Indexing on the Ordinate data object.

- Right-click on the Ordinate data object in the table and select **Indexing**.

The Data Object Indexing dialog is displayed.

You need the Raw Points row in the dialog

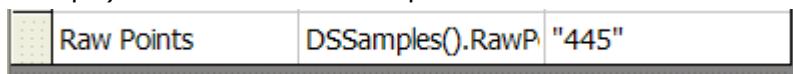


The **Index is Current** by default.

- Click in the Raw Points Index field.

The entry turns blue and a drop-down arrow is displayed.

- Enter the Abscissa value whose Ordinate value you wish to display (445 in this example). The value MUST be in quote marks:



- Click **OK**.

To display the ordinate value at the specified abscissa value for each sample, you need to repeat the second row based on task samples.

- Right-click on the table and select **Properties**.

The Table Properties dialog is displayed.

17. Select the Rows tab.
18. Click Next Row.
Row 2 is displayed at the top of the dialog.
19. Ensure Enables row to expand is selected.
20. Select **Sample List** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

21. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed.

Sample	Ordinate value at 445 nm
Sample1.Sample	1.72816264808361
Sample2.Sample	1.7203010528572

NOTE: To display the ordinate value of a processed data point, follow the procedure described above, but use the **Processed Points Ordinate** data object instead.

User List

How do I add a list of all user names and IDs to my report template?



1. Select the Table Layout tool



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter **2** columns and **2** rows.



5. Select the Text Block layout tool



The mouse pointer changes to

6. Click the mouse inside a field in the first row of the table.

A text object is placed in the field

TEXT2	
-------	--



and the text object is added below table on the tree

7. Click inside the blue box in the table and edit the text as required.

In the example below '**UserName**' has been entered.

User Name	

8. Repeat steps 5–7 for '**User ID**' headings.

User Name	User ID

NOTE: The text must be column headings and not row headings as the repeat (described below) is repeated on rows. If you only want to list Names or IDs, create a table that has 1 column and 2 rows and proceed in the same way.

- Click on the **Name** data object in the Data Object list to select it.



The mouse pointer changes to .

- Position the mouse pointer in the empty field in the table and drag to the required size.
The table now looks something like:

User Name	User ID
Name (DATA23)	

- Repeat steps 9 and 10 for the **User ID** data object.

The table should now look like this:

User Name	User ID
Name (DATA23)	ID (DATA24)



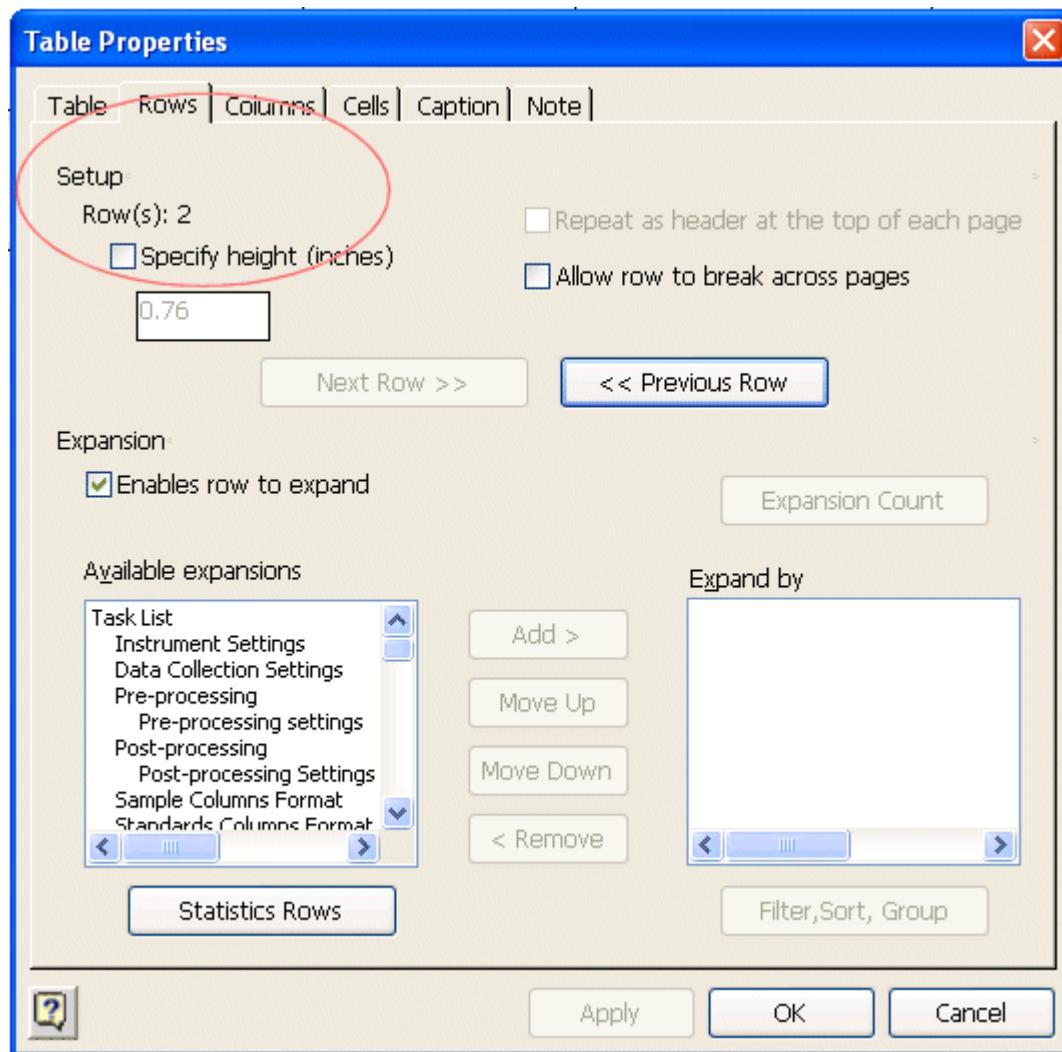
- Click on TABLE in the tree to select the table

- Right-click on the table and select **Properties**.

The Table Properties dialog is displayed.

- Click Next Row.

Row 2 is now specified at the top of the dialog.



15. Select **Enables row to expand**.
16. From the list of Available expansions, select **User List**, and then click **Add**.
Instrument Settings moves from the Available expansions list to the Expand by list.
17. Click **OK**.

To view what will actually appear when the report is printed you need to print preview the report.

18. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

User Name	User ID
PerkinElmer Method Developer	PEDeveloper
Administrator	Administrator
tracey	tracey

Instrument List

The instrument list is used to provide information about all the available instruments. If you wish to include instrument settings used for a particular task, you should use the Data Collection data objects, which are available within the Task List.

How do I add the name, type, serial number, IPV status and the date the instrument information was last modified to my template?

The instructions below describe how to put this information in a table as the individual values may be meaningless on the page.

Information for all instruments can be displayed.



1. Select the Table Layout tool



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.
Table is added to the tree.
3. To format the size of the table, right click on the table and select **Properties**.
The format table dialog is displayed.
4. Select the Table tab and enter the number of columns and rows.
The size will depend on how many options you wish to include. The example has 2 columns and 5 rows.

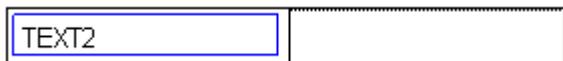


5. Select the Text Block layout tool



The mouse pointer changes to .

6. Click the mouse inside a field of the table.
A text object is placed in the field



and the text object is added below table on the tree.

7. Click inside the blue box in the table and edit the text as required.
 In the example below '**Instrument Name**' has been entered.

Instrument name	

8. Repeat steps 5–7 to enter text for the other settings:

Instrument name	
Instrument type	
Serial number	
IPV status	
Last modified	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Instrument name	Name (DATA31)
Instrument type	
Serial number	
IPV status	
Last modified	

and the data object is added to the tree.

11. Repeat steps 9 and 10 for the other output settings data objects to complete your table:

Instrument name	Name (DATA31)
Instrument type	Type (DATA32)
Serial number	Serial No. (DATA33)
IPV status	IPV Status (DATA34)
Last modified	Last Modified (DATA35)

If you have more than one instrument installed you will need to create a section that will then repeat for all instruments. If you do not do this, the report will give details of the last instrument that was added to the database. This may not be your default instrument if you have more than one instrument installed.



12. Within the Layout Tools click **Section**.



The mouse pointer changes to .

13. Drag the mouse around the table.

SECTION2	
Instrument name	Name (DATA31)
Instrument type	Type (DATA32)
Serial number	Serial No. (DATA33)
IPV status	IPV Status (DATA34)
Last modified	Last Modified (DATA35)

The Format Section dialog is displayed.

14. Select **Repeat section based on**, and then select **Instrument List** from the drop-down list.

To view what will actually appear when the report is printed you need to print preview the report.

15. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Instrument name	Sim45
Instrument type	Lambda 45
Serial number	Simulation
IPV status	None
Last modified	16 June 2003 09:23 AM BST

Instrument name	Lambda 800
Instrument type	Lambda 800
Serial number	Simulation
IPV status	None
Last modified	11 September 2003 09:19 AM BST

NOTE: The IPV data objects are listed but they are only ever populated in templates that are associated with IPV tests. It is not possible to change the default templates that are associated with the IPV tests. It is possible to edit the default templates but WE STRONGLY RECOMMEND THAT YOU DO NOT DO THIS. Therefore, these data objects will not be discussed in the Help.

Query Results List

NOTE: All the data objects in the Query Results List should only be used on templates that will be associated with Query Results. The data objects are not populated (that is, they will appear blank on the report template) unless they are used on a report template associated with a query.

NOTE: Do not use the Calibration Results data object from this list.

How do I use the Query Results Table data object?

The Query Results Table data object displays a table of all the settings specified when generating a query.

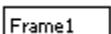
1. Select the **Query Results Table** from the Query Results data objects .



The mouse pointer changes to .

2. Click the mouse on the page.

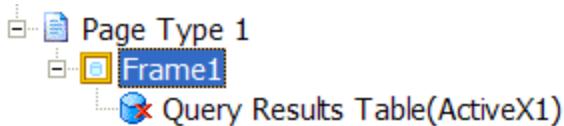
A frame is added to the page.



3. Drag the frame to enlarge it:



The frame object is added to the tree:



4. Save the template.

5. Create a Query.

6. Highlight the results you wish to include in the table in the report.

If only one result is selected, default templates are used. You must select at least two results in order to be able to specify the template to be used for printing.

7. From the File menu select **Print Preview**.

The Print Query dialog is displayed.

8. Select Single report using template.

NOTE: You must select **Single report using template**. If not, the table will not populate and the report will appear blank.

9. Select the template containing the Query Results Table data object from the drop-down list of available templates.
10. Click **OK**.

The report is displayed in the Communiqué Print Preview window:

Task Name	Sample ID	Concentration	Residual	Ordinate
Scanning quant - 1 -	Sample1.Sample	2.0000	0.2457	1.4990
Scanning quant - 1 -	Standard2.Sample	2.0000	0.2245	1.5217
Scanning quant - 1 -	Standard3.Sample	1.0000	-0.1122	0.8124

How do I use the Query Results Chart data object?

NOTE: When the Query Results Chart data object is used, the chart displayed on the report will be the chart that is currently displayed in the Display Pane of the Query window.

The Query Summary data object displays a table of all the settings specified when generating a query.

1. Select the **Query Results Chart** from the Query Results data objects .



The mouse pointer changes to

2. Click the mouse on the page.

A frame is added to the page.

 Frame1

3. Drag the frame to enlarge it:



The frame object is added to the tree.

4. Save the template.

5. Create a Query.

6. Highlight at least two results and create a chart.

If only one result is selected, default templates are used. You must select at least two results in order to be able to specify the template to be used for printing. See Queries for further information about creating charts.

7. From the File menu select **Print Preview**.

The Print Query dialog is displayed.

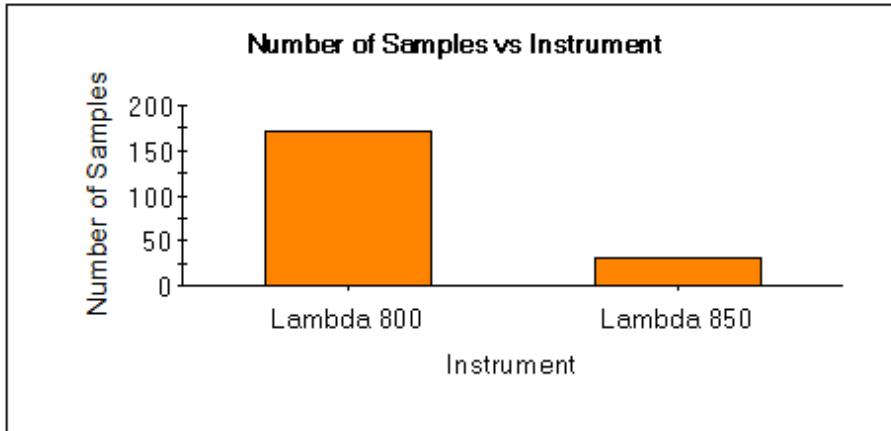
8. Select Single report using template.

NOTE: You must select **Single report using template**. If not, the Chart will not populate and the report will appear blank.

9. Select the template containing the Query Results Table data object from the drop-down list of available templates.

10. Click **OK**.

The report is displayed in the Communiqué Print Preview window. The chart displayed on the report is the chart that is currently displayed in the Display Pane of the Query window. For example:



How do I use the Query Summary data object?

The Query Summary data object displays a table of all the settings specified when generating a query.

1. Select the **Query Summary** from the Query Results data objects .

The mouse pointer changes to

2. Click the mouse on the page.

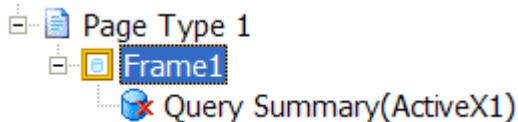
A frame is added to the page.



3. Drag the frame to enlarge it:



The frame object is added to the tree:



4. Save the template.

5. Create a Query.
6. Highlight at least two results.
If only one result is selected, default templates are used. You must select at least two results in order to be able to specify the template to be used for printing.
7. From the File menu select **Print Preview**.
The Print Query dialog is displayed.
8. Select Single report using template.

NOTE: You must select **Single report using template**. If not, the Query Summary will not populate and the report will appear blank.

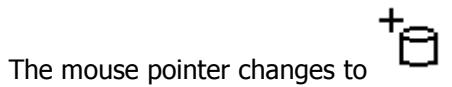
9. Select the template containing the Query Summary data object from the drop-down list of available templates.
10. Click **OK**.
The report is displayed in the Communiqué Print Preview window. The report contains a summary of all the settings used to generate the query. The information from the Sort By tab is not included as this only determines how the results are displayed in the Results Table, and not what data is found as a result of the query parameters.

Method	Method of type Scan
Sample	Samples of type Sample
Person	All Analysts
Date	All Dates
Result	Results with Status Complete
Instrument	All Instruments

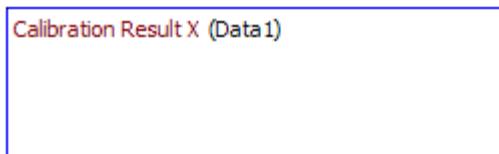
How do I use the Calibration Result X data object?

This data object must only be used on report templates that will be used for printing the results of calibration Queries. It is used to generate a printed calibration report for all selected results in the query. (The calibration report is the report that is seen when a calibration result is selected in the Results Table, and **Report** is selected on the Results Tree.)

1. Select the **Calibration Result X** data object.



2. Click on the report template and drag to the required size.



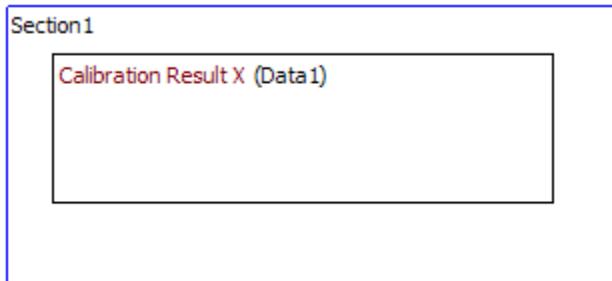
NOTE: A section must be used with this data object so that the calibration report will print for all selected results. If a section is not used, the report template will not be populated.



3. From the Layout Tools list, select

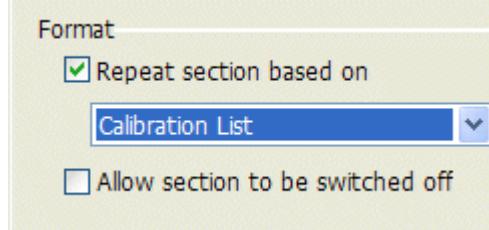
The mouse pointer changes to

4. Drag the mouse around the object frame to create a section containing the Calibration Result X object:



The Section Properties dialog is displayed.

5. Select Repeat section based on **Calibration List**



6. Click **Apply** and then click **OK**.
 7. Save the report template.
 8. Create a Calibration Query.
 9. Highlight the results you wish to obtain report for.
 10. From the File menu select **Print Preview**.
 The Print Query dialog is displayed.
 11. Select Multiple reports using template.

NOTE: You must select Multiple reports using template.

12. Select the template containing the Calibration Result X data object from the drop-down list of available templates.
 13. Click **OK**.
 The report is displayed in the Communiqué Print Preview window. The report contains a text calibration report for each of the highlighted results in the Query Results Table.

Pages 1 of 4 GoTo 100%

1.....2.....3.....4.....5.....6.....7.....8.....9.....10.....11.....

Calibration Time: 23/01/2004 14:11
User Full Name: a

Component Name: Analyte
Component Units: mgml-1

Calibration: Calibration Curve - Linear ($y=mx+c$)
Ordinate Mode: Maximum Height
Baseline Correction: None

Settings (nm): Start:800.00 End:300.00

Force through Zero: No

Calibration Coefficients:
 $a_0 = -0.377038$
 $a_1 = 1.069421$

Specified Correlation Coefficient: 0.980000
Calculated Correlation Coefficient: 0.981622

Index	StandardID	Specified	Calculated	Residual
-------	------------	-----------	------------	----------

Calibration Time: 23/01/2004 14:15
User Full Name: a

Component Name: Analyte
Component Units: mgml-1

Calibration: Calibration Curve - Linear ($y=mx+c$)
Baseline Correction: None

Settings (nm): Start:350.00 End:500.00

System Data

How do I place the data object in a table with text to explain what the object is?



1. Select the Table Layout tool:



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

In the example below there are two columns and one row.



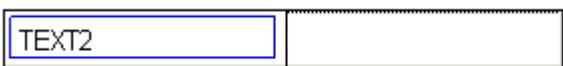
5. Select the Text Block layout tool:



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:

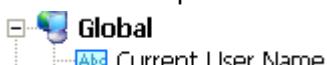


7. Click inside the blue box in the table and edit the text as required.

In the example below 'Current User Name' has been entered.



8. Click on the required data object in the Data Object list to select it. For example:



The mouse pointer changes to

9. Position the mouse pointer in the empty field in the table and drag to the required size.
The table now looks something like

Current User Name	Current User Name (Data2)
-------------------	---------------------------

and the data object is added to the tree.

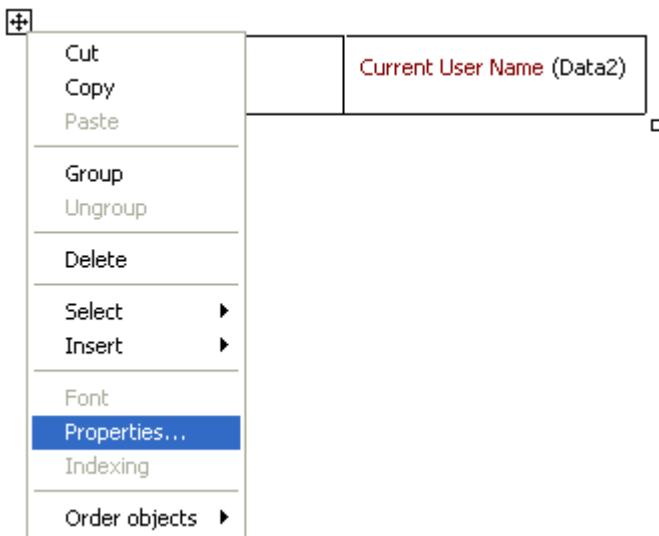
To view what will actually appear when the report is printed you need to print preview the report.

10. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Current User Name	tracey darling
-------------------	----------------

11. If you wish to remove the table borders, right click on the table in the template editor (not print preview), and select **Properties**.



12. In the Table Properties dialog, remove all the table lines:

The table will then appear without any lines in the Print Preview.

Current User Name tracey darling

Miscellaneous

How do I add page X of Y to my template?

NOTE: Page X of Y can only be added to the footer section of the report template.

1. Select **Default Footer** from the Tree on the left hand side of the screen.
2. Click on **Page X of Y** in the Data Object list to select it.

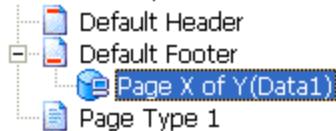


3. The mouse pointer changes to
4. Position the mouse pointer within the footer and drag to display.

The following object is added to the template

Page X of Y (Data1)

and the Page X of Y object is added to the tree on the left hand side of the template:



To view what will actually appear when the report is printed you need to print preview the report.

5. Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Page 1 of 1

How do I add the page number to my template?

NOTE: The **Page Number** can only be added to the footer of the report template.

1. Select **Default Footer** from the Tree on the left hand side of the screen.
2. Click on **Page No.** in the Data Object list to select it.

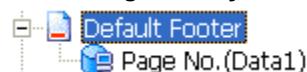


3. The mouse pointer changes to
4. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Page No. (Data1)

and the Page No object is added to the tree on the left hand side of the template:



To view what will actually appear when the report is printed you need to print preview the report.

5. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See **How do I place the data object in a table with text to explain what the object is ?**

How do I add the number of pages to my report template?

NOTE: The number of pages(**Num Pages**)can only added to the footer of the report template.

1. Select **Default Footer** from the Tree on the left hand side of the screen.
2. Click on **Num Pages** in the Data Object list to select it.



3. The mouse pointer changes to .
4. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Num Pages (Data1)

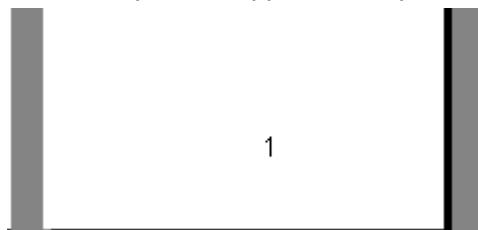
and the Num Pages object is added to the tree on the left hand side of the template:



To view what will actually appear when the report is printed you need to print preview the report.

5. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



NOTE: **Num Pages** will not automatically appear on all pages of the report. It must be added to each page of the template.

NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. In the example above the number could be the page number or the number of pages. You can create a table where you can enter text in one field and the data object in other field. See *How do I place the data object in a table with text to explain what the object is?*

How do I add the version number of the Communiqué Software to my report template?

1. Click on **Communiqué Software Version** in the Data Object list to select it.



2. The mouse pointer changes to

3. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Communiqué Software Version (Data1)

and the Communiqué Software Version object is added to the tree on the left hand side of the template:



To view what will actually appear when the report is printed you need to print preview the report.

4. Click

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See *How do I place the data object in a table with text to explain what the object is?*

How do I add the name of the user who is logged into the computer to my template?

1. Click on **Operating System User Name** in the Data Object list to select it.

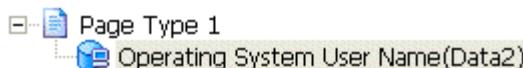


2. The mouse pointer changes to .
3. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Operating System User Name (Data2)

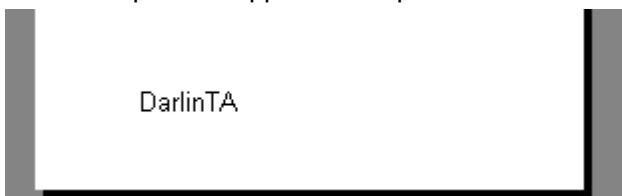
and the Operating System User Name object is added to the tree on the left hand side of the template:



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See *How do I place the data object in a table with text to explain what the object is?*

How do I add the name of operating system and the software version to my template?

1. Click on **Operating System And Version** in the Data Object list to select it.

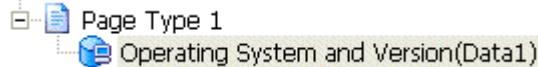


2. The mouse pointer changes to .
3. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Operating System and Version (Data1)

and the OperatingSystemAndVersion object is added to the tree on the left hand side of the template:



To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



Microsoft Windows XP Version: 5.1 Service Pack 1

NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See *How do I place the data object in a table with text to explain what the object is?*

How do I add the name of the computer to my template?

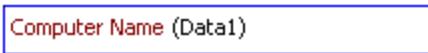
1. Click on **Computer Name** in the Data Object list to select it.



2. The mouse pointer changes to .

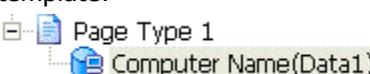
3. Position the mouse pointer on the template and drag to display.

The following object is added to the template



Computer Name (Data1)

and the Computer Name object is added to the tree on the left hand side of the template:

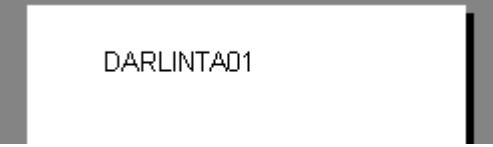


Page Type 1
Computer Name(Data1)

To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



DARLINTA01

NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See *How do I place the data object in a table with text to explain what the object is?*

How do I add the date and time the report was generated to my template?

This date and time is when the report is generated.

1. Click on **Date Time Report** in the Data Object list to select it.



2. The mouse pointer changes to .
3. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Date Time Report (Data1)

and the Date Time Report object is added to the tree on the left hand side of the template:

 Page Type 1
 Date Time Report(Data1)

To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See *How do I place the data object in a table with text to explain what the object is?*

How do I add the PC clock time to my report?

This is the time on the computer's clock when the report is printed.

1. Click on **Date Time Local PC** in the Data Object list to select it.



2. The mouse pointer changes to .

3. Position the mouse pointer on the template and drag to display.

The following object is added to the template

Date Time Local PC (Data1)

and the Date Time Local PC object is added to the tree on the left hand side of the template:

Page Type 1
 Date Time Local PC(Data1)

To view what will actually appear when the report is printed you need to print preview the report.

4. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:



NOTE: It may not be particularly useful to have just the object on the page as it is not clear what it relates to. You can create a table where you can enter text in one field and the data object in other field. See

Templates

How do I add the name of the template, the template creation date, the template author, the date the template was last edited, the name of the person who last edited the template, the template description, current revision, previous revision, previous name, and data model version to my report template?

The instructions below describe how to put this information in a table as the individual values such as **1** for **Template Previous Revision** would be meaningless on the page.



1. Select the Table Layout tool:



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 10 rows.



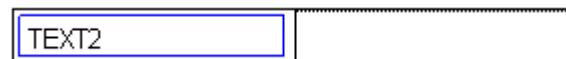
5. Select the Text Block layout tool:



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Template Name**' has been entered:

Template Name	

8. Repeat steps 5–7 to enter text for the other template objects. For example:

Template Name	
Template Creation Date	
Author	
Edited Date	
Editor	
Description	
Current Revision	
Previous Revision	
Previous Name	
Data Model Version	

9. Click on the required data object in the Data Object list to select it.



The mouse pointer changes to .

10. Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Template Name	Template Name (Data1)
Template Creation Date	
Author	
Edited Date	
Editor	
Description	
Current Revision	
Previous Revision	
Previous Name	
Data Model Version	

11. Repeat steps 9 and 10 for the other output settings data objects EXCEPT THE CREATION DATE AND EDITED DATE:

Template Name	Template Name (Data1)
Template Creation Date	
Author	Template Author (Data3)
Edited Date	
Editor	Template Editor (Data5)
Description	Template Description (Data6)
Current Revision	Template Current Revision (Data7)
Previous Revision	Template Previous Revision (Data8)
Previous Name	Template Previous Name (Data9)
Data Model Version	Template Data Model Version (Data10)

If a date object is placed in a field which is too small, when you print preview you will see ##### instead of the date. Rather than enlarging the field (which may not always be possible if you have a large table), you can create a text block and then place the date object inside this block. The text block will handle any wrapping or expanding that is required.

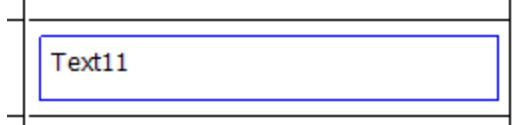


12. Select the Text Block from the layout tools:



The mouse pointer changes to

13. Drag the mouse on the report template to create a Text Block inside the date field:



14. Click inside the Text Block and remove the default text.

15. Select the Template Creation Date data object and click inside the text field:



16. Repeat for the Template Edited Date data object.

The table should now look like:

Template Name	Template Name (Data1)
Template Creation Date	[D11:Template Creation Date]
Author	Template Author (Data3)
Edited Date	[D12:Template Edited Date]
Editor	Template Editor (Data5)
Description	Template Description (Data6)
Current Revision	Template Current Revision (Data7)
Previous Revision	Template Previous Revision (Data8)
Previous Name	Template Previous Name (Data9)
Data Model Version	Template Data Model Version (Data10)

To view what will actually appear when the report is printed you need to print preview the report.

17. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Template Name	A1
Template Creation Date	02/07/2004 04:48:12 PM GMT Standard Time
Author	PerkinElmer Method Developer
Edited Date	02/07/2004 04:48:12 PM GMT Standard Time
Editor	PerkinElmer Method Developer
Description	Default template for short scans.
Current Revision	1
Previous Revision	0
Previous Name	template for short scans.
Data Model Version	UV WinLab Main 1.0

How do I add the Template Status to my report template?



1. Select the Table Layout tool:  Table



The mouse pointer changes to .

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree  Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

In the example below there are two columns and one row.





5. Select the Text Block layout tool:



The mouse pointer changes to .

6. Click the mouse inside a field of the table.

A text object is placed in the field

Text1	
-------	--

and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

In the example below '**Template Status**' has been entered.

Template Status	
-----------------	--

8. Click on the **Template Status** Data Object list to select it.



The mouse pointer changes to .

9. Position the mouse pointer in the empty field in the table and drag to the required size.

The table now looks something like

Template Status	Template Status (Data1)
-----------------	-------------------------

and the data object is added to the tree.

To view what will actually appear when the report is printed you need to print preview the report.

10. Click .

Template Status	UN-APPROVED
-----------------	-------------

How do I add a table of all the template signatures to my report template?

The Templates Signatures data object will create a table of all the signatures that have been applied to the report template – when the report template has been saved, reviewed and approved.

1. Select the Templates Signatures data object.



The mouse pointer changes to (as the object is added as a table).

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1 .

A table header and table column headers are automatically created:

Template Signature

Name	Reason	Comments	Date/Time	Status

To view what will actually appear when the report is printed you need to print preview the report.

3. Click .

Template Signature

Name	Reason	Comments	Date/Time	Status
tracey	Saved		04/02/2004 12:02:45 PM GMT Standard Time	TemplateEdit
Reviewer	Reviewed for technical content		04/02/2004 12:03:04 PM GMT Standard Time	TemplateReview
Approver	Approved		04/02/2004 12:03:15 PM GMT Standard Time	TemplateApprove

Reports

How do I add the report name, ID, template name and revision, data model version, and report created date to my report template?

The instructions below describe how to put this information in a table as the individual values such as 1 for **Report Template Revision** would be meaningless on the page.



1. Select the Table Layout tool:



The mouse pointer changes to

2. Position the mouse pointer on the template and drag to create a table.

Table is added to the tree Table1

3. To format the size of the table, right click on the table and select **Properties**.

The Table Properties dialog is displayed.

4. Select the Table tab and enter the number of columns and rows.

The size will depend on how many options you wish to include. The example below has 2 columns and 6 rows.



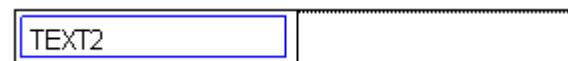
5. Select the Text Block layout tool:



The mouse pointer changes to

6. Click the mouse inside a field of the table.

A text object is placed in the field



and the text object is added below table on the tree:



7. Click inside the blue box in the table and edit the text as required.

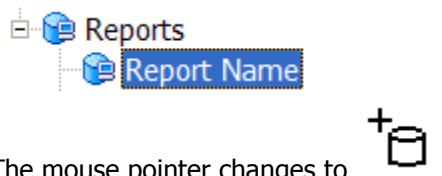
In the example below '**Report Name**' has been entered.

Report Name	

8. Repeat steps 5–7 to enter text for the other template objects:

Report Name	
Report ID	
Report Template Name	
Report Template Revision	
Report Data Model Version	
Report Created Date	

9. Click on the required data object in the Data Object list to select it:



- Position the mouse pointer in the empty field in the table next to the corresponding text and drag to the required size.

The table now looks something like:

Report Name	Report Name (Data1)
Report ID	
Report Template Name	
Report Template Revision	
Report Data Model Version	
Report Created Date	

- Repeat steps 9 and 10 for the other output settings data objects EXCEPT THE REPORT CREATED DATE:

Report Name	Report Name (Data1)
Report ID	Report ID (Data2)
Report Template Name	Report Template Name (Data3)
Report Template Revision	Report Template Revision (Data4)
Report Data Model Version	Report Data Model Version (Data5)
Report Created Date	

If a date object is placed in a field which is too small, when you print preview you will see ##### instead of the date. Rather than enlarging the field (which may not always be possible if you have a large table), you can create a text block and then place the date object inside this block. The text block will handle any wrapping or expanding that is required.



- Select the Text Block from the layout tools



The mouse pointer changes to

- Drag the mouse on the report template to create a Text Block inside the date field.

14. Click inside the Text Block and remove the default text.
15. Select the **Report Created Date** data object and click inside the text field:

[D6:Report Created Date]

The table should now look like:

Report Name	Report Name (Data1)
Report ID	Report ID (Data2)
Report Template Name	Report Template Name (Data3)
Report Template Revision	Report Template Revision (Data4)
Report Data Model Version	Report Data Model Version (Data5)
Report Created Date	[D6:Report Created Date]

To view what will actually appear when the report is printed you need to print preview the report.

16. Click .

The report is displayed and the data objects are populated with the information. This is how the report will appear when printed:

Report Name	A1
Report ID	[Report ID]
Report Template Name	A1
Report Template Revision	1
Report Data Model Version	UV WinLab Main 1.0
Report Created Date	

NOTE: In print preview the Report ID and Created date are not populated as the report has not been generated.

Communiqué FAQ

Why is the date object displayed as ##### when I print preview the report?

This means that the object is too big for the field. The field will not expand to fit the information. You can enlarge the field although this is not always possible depending on the space available on your report template. Or, you can create a text block and then place the date object inside this block. The text block will handle any wrapping or expanding that is required.



1. Select the Text Block from the layout tools



The mouse pointer changes to .

2. Drag the mouse on the report template to create a Text Block.

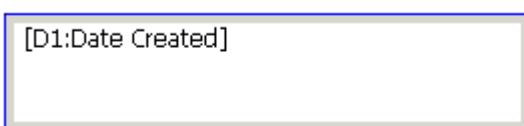


3. Click inside the Text Block and remove the default text.

4. Select the Date object from the Data Objects list.

5. Click inside the Text Block.

The object is placed inside the Text Block.



To view what will actually appear when the report is printed you need to print preview the report.

6. Click .

11/12/2003 01:20:46 PM GMT
Standard Time

NOTE: This also applies to a text block that is within a table, which prevents the date column from being very long.

Why is the footer on my report not printing out fully?

Older Deskjet printers (for example, Deskjet 660) seem to have internal margins. You will have to modify the template to place the footer information into the main page. If you create a section with a page break, this will allow you to repeat the footer information on each page.

Can I amend and re-save a template that has been approved?

No, once the template is approved no further modifications are possible as it becomes 'read-only'. However, you can save it with a new name and this version will be fully editable.

How can I set the number of significant figures or decimal places for the values displayed in my report?

If the 'numeric' data objects (for example, an ordinate value) are within a table, you can right-click on the table and select **Properties**. Select the Cells tab and define the numeric formatting for the table.

If the object is not in a table, right-click on the object and select **Properties**. The Numeric Data Object Properties dialog is displayed. You can select **Significant figures** or **Decimal places** and specify the number of Significant figures or Decimal places from the appropriate drop-down list.

Does a graph retain the colors set in the Workspace when it is placed in a report template?

Colors are generally retained between the Workspace and Communiqué. However, if you wish, you can change the colors of the graphs within Communiqué.

- Right-click on the graph object in the template and select **Properties**.
The Graph Properties dialog is displayed.

How can I get my text blocks and data fields to line up vertically?

If a data object is placed directly on the report template it can sometimes be difficult to line this up with an associated Text Block. To solve this, you can put the data object in its own text block:



1. Select the Text Block from the layout tools



The mouse pointer changes to

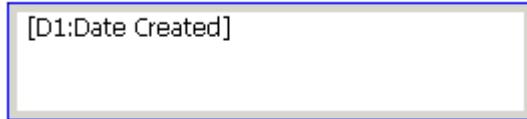
2. Drag the mouse on the report template to create a Text Block.



3. Click inside the Text Block and remove the default text.
4. Select the Data object from the Data Objects list.

5. Click inside the Text Block.

The object is placed inside the Text Block.



You will now be able to align this with another Text Block easily.

I'm having trouble moving my Communiqué objects around on the report template. How should I do it?

Click on the object already on the template to select it (The outline turns blue to show that it is selected and the mouse pointer changes to a four-headed arrow to show that it can be moved). Drag the object to the new position.

Does the object frame determine the size of the spectrum that will be displayed in my report?

Yes. You must drag the object frame to the required size. This allows you to specify how large the spectrum will appear on your report.

What is the difference between ID and Full ID?

The **ID** data object will report the Sample ID as seen in the Sample Table. This is sufficient if you have a fairly simple Sample Table, for example just 5 samples:

ID
Sample38
Sample37
Sample39
Sample40
Sample41

However, this does not report the full sample name which includes the extension (as seen in the Results Table). The **Full ID** is particularly useful if you are using Replicates or Measurements. If, for example, you have 1 sample with 2 replicates, and you create a table in the report template using **ID**, you will see the Sample ID twice but it will not show that these are replicates:

ID
Sample1
Sample1

If you use the **Full ID** data object, the full name including the extension will be shown:

Full ID
Sample1.Replicate1
Sample1.Replicate2

Why do some of the data objects have a red cross through them?

A red cross through an object, for example  **Query Results Table**, signifies that this is an Active X object. This is simply how the software provides the data for the report. It does not mean that the object is broken or unavailable.

Why do the examples of the data objects in the Help have '1' after them in the Tree View but I always have a different number?

The Data Object is named (Object Name) DataX where X is numerically incremented (Object Name)Data1, etc. for each new Data Object added to the template. You can rename the Data Object if you wish. As long as you have followed the example, you should see the same information in your report. The incremental number is not important.

Why is there a browse for folder option within the Communiqué print dialog but my reports are not stored within folders?

Communiqué reporting is available to other PerkinElmer applications, and so the Print dialog has to cater for all these applications. Folders are not available for storing reports within UV WinLab.

Why is my column in a table not populating when I have used Indexing for the column header?

If you have created a custom column in a table within UV WinLab, you may wish to include this column in a table within your report template. (see 'How do I create a table that contains all the information in my results table and will continue over several pages when printed ?' on the Sample List page of the Help).

When you use Indexing to populate the column, you must put the name of the column within quotes but you must not include any units that are present in the column header.

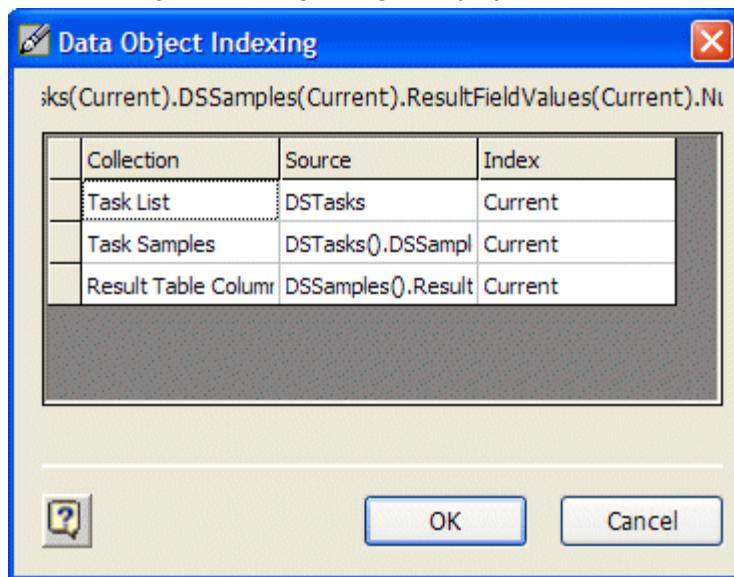
You have a Results Table column that you have created called Ordinate (that has the units A)

	Sample ID	Ordinate (A)
✓	1 Sample2.Replicate1	0.11058
✓	2 Sample3.Replicate1	0.26624
✓	3 Sample4.Replicate1	0.07767
✓	4 Sample2.Replicate2	0.11048

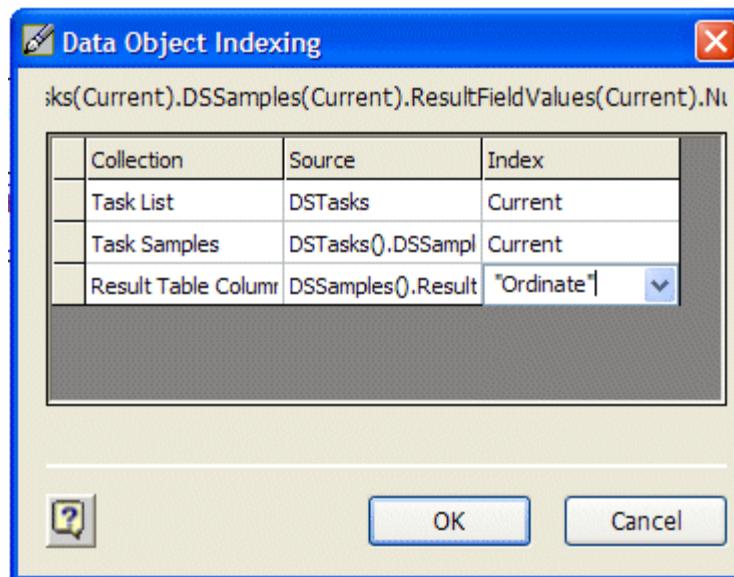
1. Create a table within the report template.
2. Enter the Heading of the column, for example, **Ordinate**.
3. Select the Number data object in Table Columns (within the Sample List).

4. Right-click inside the data object beneath the **Ordinate** header and select **Indexing**.

The Data Object Indexing dialog is displayed:



5. Select the Index field for the Result Table columns.
6. Within quotes (" "), enter the name of the column as it appears in the results table -



If the name is not **exactly** correct (including upper and lower case letters), the table will not populate with data.

NOTE: Do not enter the units that were in the original column header. In the above example, (A) is included in the header in the Results Table, but if this is included in the Index, the column will not populate.

When you print preview, the column will now populate with all the data from the Ordinate column in the Results Table.

Why is the Autosampler tray layout (E, F, or G) not reported in my report?

The tray layout is reported as part of the Data Collection settings. However, a value of 0, 1, or 2 – corresponding to E, F, or G respectively is reported rather than the letter of the tray.

Why does the standards table in the Default Scanning Quant report not show all the information from the Calibration page of the task?

The size of the table on the report is limited. The report will fit as many columns as possible in the table in the order in which they are displayed within the task. You can use the Table Builder dialog to choose which columns to hide and show, and the order of these columns within the table and this will be reflected in the table printed in the report. See How do I format the Standard Table (Columns tab).

How can I transfer my UV data tables into Excel?

This can be achieved in Communiqué by setting up a table. It is best to keep the report as concise as possible (that is, as little other information as possible including headers and footers). It is important to ensure that the table is fully populated. If you are using fields from the Sample Table such as description, make sure that these are set to mandatory in the method otherwise mis-assignment of the data fields will result.

Within the Output page of the task, select **Print to file**. Excel (*.csv) is one of the options within the settings. The resulting file (which is stored under the task name unless it is renamed) will be placed in:

- For Windows XP, C:\Documents and Settings\Administrator\My Documents.
- For Windows 7, C:\Users\Administrator\Documents.

The 'print to database' option is grayed. How can I send my reports to the database?

Only reports that use approved templates can be sent to the database. You need to approve your template. This is done from the print preview window in Communiqué.

Further Information

Further Information

See the following for further information on topics related to UV Spectroscopy.

[Recommended Practices](#)

[Handling Calibration Standards](#)

[Choice of Solvent](#)

[IPV References](#)

[UV Spectroscopy](#)

[Sources of Degradation of Spectrophotometric Performance](#)

[FAQs](#)

Recommended Practices

Following these guidelines will help ensure good instrument performance:

- Operate the instrument in a clean laboratory environment.
- Do not operate the instrument in a high temperature or high humidity environment.
- Plug the instrument into a proper power line free from interference.
- Do not drop the instrument and do not place instrument on an uneven benchtop.
- Do not block air flow to the instrument or partially cover the instrument during operation.
- Do not put fingerprints or marks on the radiation source or optical surfaces.
- Handle the sources by the base and always wipe off the envelope before installation.
- Do not get fingerprints on cells or allow reagent to run down the sides of the cells. Wipe the cell clean before placing it into the instrument.
- Change source lamp(s) when performance begins to degrade instead of waiting until the lamp(s) burn out. Tungsten halide lamps provide a continuous energy output throughout lamp life, and no degradation of instrument performance with lamp age.
- Increase the lamp life by turning off the lamp when not in use and keeping the instrument power on.
- Always refer to the operating directions for details on each individual instrument, as well as for manufacturer's practices for optimum performance.

Handling Calibration Standards

Calibration standards are to be treated with special care if they are to retain their validity. Scratches, dirt etc on the optical surfaces can easily introduce substantial errors.

Observe the following rules when handling calibration standards:

- Take special care not to touch or scratch the optical surfaces when inserting a calibration standard into the cell holder in the spectrometer.
- Do not use calibration standards in corrosive or dusty atmospheres.
- After use, do not place calibration standards on the laboratory bench, but return them immediately to their storage container.
- Always keep the storage container closed in a safe place where dust cannot accumulate on it.
- Do not clean the optical surfaces of calibration standards unless absolutely essential. If you need to clean the optical surfaces, for example to remove a fingerprint, very carefully press the glass filter out of the spring mount with a soft wooden probe. Take care to hold the filter only by the edges. Clean the optical surfaces by very carefully wiping them with a soft, lint-free cloth moistened with ethanol. Return the filter to the spring mount with the same orientation and press it into place with a soft wooden probe.

Choice of Solvent

The table below gives a list of some common solvents and the minimum wavelength from which they may be used in a 1 cm cell.

Solvent	Minimum Wavelength (nm)
acetonitrile	190
water	191
cyclohexane	195
hexane	195
methanol	201
ethanol	204
ether	215
methylene chloride	220
chloroform	237
carbon tetrachloride	257

IPV References

- National Bureau of Standards, (1983) *Accuracy in Analytical Spectrophotometry*, NBS Special Publication, 260-82, National Bureau of Standards, Washington, D.C.
- Alman, D.H., Billmeyer, F.W., (1975) "A Review of Wavelength Calibration Methods for Visible-Range Photoelectric Spectrophotometers", *J. Chem Ed.* 52(5), A28, 53(6), A325.
- Beeler, M.F., (1974) "Laboratory performance of spectrophotometers: Results of a national survey", *Am. J. Clin. Pathol.*, 61, pp. 789 - 798.
- Bohme, W., Steinwand, M., (1983) "UV/VIS Spectrometers as Detectors in Modern Liquid Chromatography", *Applied UV Spectroscopy*, 9E, Bodenseewerk PerkinElmer GmbH, Überlingen Germany.
- Burgess, C., Knowles, A., (1986) *Standards in Absorption Spectrometry*, Chapman and Hall Ltd., London/New York.
- Chamran, M., Keiser, R., "Maintaining Optimum Spectrophotometer Performance", PerkinElmer Application Data Bulletin ADS 109.
- Demey, R.C., Sinclair R, (1987) *Visible and Ultraviolet Spectroscopy*, John Wiley and Sons, New York.
- Denny R.C., (1973) *A dictionary of spectroscopy*, MacMillan Press Ltd., London, pp. 1-161.
- Coleman Instruments Division, (1972) *Directions for Didymium Calibration Standard*. Oak Brook, IL: Coleman Instruments Division, No 6-400, D241R.
- Drews, U.W., Chamran, M., "Limitations in Using Photometric Accuracy Standards", PerkinElmer Applications Data Bulletin ADS 107.
- Eckerle, K.L., Sutter, E., et al., (1990) "International Comparison of Regular Transmittance Scales", *Metrologia*.
- Lange, B., Vejdelek, Z.K., (1980) "Photometrische Analyse", *Verlag Chemie*, Weinheim.
- Macdonald, R.P., (1964) "Uses for a holmium oxide filter in spectrophotometry", *Clin. Chem.*, 10, pp. 1117-1120.
- Rand, R.N., (1972) "The Role of Spectrophotometric Standards in the Clinical Chemistry Laboratory", NBS Special Publication 378, National Bureau of Standards, Washington, D.C.
- Slavin, W., (1963) "Stray light in ultraviolet, visible and near-infra-red spectrophotometry", *Anal. Chem.*, 35, pp. 561-566.
- American Society of Testing Materials Committee E13 (1969) "Standard definitions of terms and symbols relating to molecular spectroscopy (E131-71)", *Manual on Recommended Practices in Spectroscopy*, American Society of Testing Materials Committee E13, Philadelphia, PA, pp. 10-34.

- American Society of Testing Materials Committee E13 (1974) "Standard method of estimating stray radiant energy (E387-72), *Analytical Methods of Spectroscopy*, American Society of Testing Materials Committee E13, Philadelphia, PA, Annual Book of ASTM Standards, pp. 291-302.
- American Society of Testing Materials Committee E13 (1974) "Standard recommended practice for describing and measuring performance of spectrophotometers", *Analytical Methods of Spectroscopy*, American Society of Testing Materials Committee E13, Philadelphia, PA, Annual Book of ASTM Standards, pp. 193-212.
- Strobel, H.A., (1973) *Chemical Instrumentation, A Systematic Approach*, Addison-Wesley Publishing Company, Readings, USA.

UV Spectroscopy

Ultraviolet and visible (UV/Vis) absorption spectroscopy is the measurement of the attenuation of a beam of light after it passes through a sample. Absorption measurements can be made over a wavelength range or at single wavelengths.

UV/Vis spectra have broad features that are of limited use for sample identification, but the spectra are very useful for quantitative measurements. The concentration of an analyte in solution can be determined by measuring the absorbance at a particular wavelength and applying the Beer–Lambert Law (Beer's Law).

The UV/Vis spectral range is defined as approximately 190 to 1100 nm. The short-wavelength limit is determined by the absorption of spectral gases. If a spectrometer is purged with nitrogen, this lower limit can be extended to 175 nm. Beyond 175 nm, a vacuum spectrometer and suitable source are required. Some high-end UV/Vis spectrometers are able to extend their working range into the NIR region as far as 3300 nm.

Electronic Transitions

Ultraviolet and visible light are energetic enough to promote outer electrons to higher energy levels, and UV/Vis spectroscopy is usually applied to molecules or complexes in solution. There are three types of electronic transition to consider:

1. Transitions involving π , σ , and η electrons.
2. Transitions involving charge-transfer electrons.
3. Transitions involving d and f electrons. (These will not be discussed here).

When an atom or molecule absorbs energy, electrons are promoted from their ground state to an excited state.

Transitions involving π , σ , and η electrons

There are several types of electronic transitions available including:

- σ to σ^* (alkanes) – An electron in a bonding σ orbital is excited to the corresponding antibonding orbital. The energy required for this transition is large. This transition is not seen in typical UV spectra between 200 and 700 nm. Methane shows an absorbance maximum at 125 nm due to the σ to σ^* transition
- η to π^* (carbonyl compounds)
- π to π^* (alkenes, carbonyl compounds, alkynes, azo compounds).

Most absorption spectroscopy of organic compounds is based on transitions of η or π electrons to the π^* excited state. The absorption peaks for these transitions fall in the 200–700 nm region of the spectrum. These transitions need an unsaturated group in the molecule to provide the π electrons. Molar absorptivities from η to π^* transitions are relatively low, and range from 10 to 100 L mol⁻¹ cm⁻¹. π to π^* transitions normally give molar absorptivities between 1000 and 10000 L mol⁻¹ cm⁻¹.

The solvent in which the absorbing species is dissolved also has an effect on the spectrum of the species. Peaks resulting from η to π^* transitions are shifted to shorter wavelengths (blue shift) with increasing solvent polarity. This arises from increased solvation of the lone pair, which lowers the energy of the η orbital. Often, the reverse (red shift) is seen for π to π^* transitions. This is caused by attractive polarization forces between the solvent and the absorber, which lowers the energy levels of the excited and unexcited states.

- η to σ^* (oxygen, nitrogen, sulfur, and halogen compounds) - This transition is possible for compounds with lone pairs of electrons. The transition needs less energy than σ to σ^* .
- σ to π^* (carbonyl compounds)

Transitions involving charge-transfer electrons

Many inorganic species show charge-transfer absorption and are therefore called charge-transfer complexes. These species must have one component that has electron donating properties and one component that is able to accept electrons. Absorption of radiation then involves the transfer of an electron from the donor to an orbital associated with the acceptor.

Molar absorptivities from charge-transfer absorption are greater than $10000 \text{ L mol}^{-1} \text{ cm}^{-1}$.

Sources of Degradation of Spectrophotometric Performance

Environment

An unclean laboratory environment, including instrument exposure to volatile organic solvents, hydrochloric acid, nitric acid, ammonium chloride, and other fuming or volatile chemicals or cigarette smoke will degrade instrument performance. Smoke and volatile chemicals coat the sample compartment windows, optics and source lamps in a sealed instrument. Volatile organic solvents often absorb in the UV region, causing increased noise as well as possibly reducing sensitivity and causing sample interference.

High humidity and temperature may cause water condensation on the optical surfaces, which results in performance degradation. In extreme cases it may also affect some electronic components as well, resulting in early failure of parts.

Line Power Supply

Excessive power fluctuations may cause instability of the instrument and thus performance degradation. The source of power fluctuation may be caused by inadequate, old AC power lines, or overloading the AC power line with heavy current-drawing equipment.

Aging Lamp

An aging lamp causes a reduction in energy, which lowers performance and increases noise and stray radiation.

Alignment

Improper lamp alignment after lamp installation causes loss of energy and results in noise and degradation of performance. Improper alignment of the cell holder in the radiation beam also causes loss of energy and poor performance. When micro-cells or micro flowcells are used, proper alignment is critical.

Warm-up Time

If the instrument has not reached its specified warm-up time, it may not perform according to specification.

Sample Handling

Poor chemistry, improper procedure or method, and scratched or unclean cells can cause degradation of performance.

Aging of Instrument

Optical degradation with time, unclean environment and humidity all cause loss of energy and result in degradation of performance.

Electrical component aging may cause a calibration change and results in degradation of performance.

Photodetector aging or exposure of the detector to room light may cause failure or poor performance. High humidity may cause drift and photometric error. Photodiodes are less sensitive to these influences.

Proper Care of the Instrument

The manufacturer's recommendations must be followed in caring for the instrument if proper performance is to be maintained. To keep the instrument at optimum performance level, daily care should be exercised by the laboratory staff. In case of instrument failure, the instrument should be serviced only by qualified persons.

Frequently Asked Questions (FAQs)

Administration

I'm set up as an Administrator and I want to use the system for the first time but I can't access methods or the instrument – why?

The Administrator has been likened to the King in a game of chess – the most important person, but with limited rights. As an Administrator, you can make yourself a member of another group that will then give you access to all areas of the software. If you make yourself a member of the default groups provided, you will have access to everything. The software is designed like this as Software Administrators may not possess an analytical chemistry background and therefore may not be qualified to run an analysis.

I am the only system Administrator and I have forgotten my password. What should I do?

This is very serious as there is not a 'back door' into the software. This is to ensure 21 CFR part 11 compliance.

We strongly recommend that at least one more Administrator is created. A very good idea is to create a dummy administrator whose log-in is stored in a secure location (such as a safe).

I am concerned that it looks like I am able to clear the security audit trail. Is this possible?

The Security Audit Trail can be deleted only once it has been exported. You will not be allowed to proceed unless the information has been first saved as a file. Once the data has been exported, it becomes your responsibility.

Databases and File Structure

What databases are used by the software?

Three databases are used:

- The security (Users.mdb) is the smallest and contains all users and log-in information. This database MUST remain resident on the PC and cannot be run on a network.
- The second database, UVWinLab.mdb, stores all the methods and tasks (data) associated with those methods as well as the IPV (Instrument Performance Verification) information.
- The third database, Communiqué.mdb, is concerned with all aspects of the Communiqué reporting tool.

Please refer to the Administrator's Guide (provided with UV WinLab ES) for further information.

Can I put data directly onto a network?

The UV WinLab and Communiqué databases can be placed on a network although it is the responsibility of the customer to ensure that their network is reliable and secure. UV WinLab has not been designed as a multi-access 'client/server' system but it will allow the data to be stored remotely from the PC. The security database (users.mdb) MUST remain local to the PC.

Does the system create a backup of my databases automatically?

No. Two of the databases are too large for this to happen after every writing to the database event. The system does make a backup of the security database for its own internal purposes but, as this is still resident on the hard drive, it cannot be regarded as a backup for archive purposes, as data will be lost in the event of a catastrophic hard disk failure.

- ...\\PerkinElmer\\Security System\\Users.mdb
- ...\\PerkinElmer\\Security System\\Backup\\Users.bak (this is a backup of users.mdb)
- ...\\PerkinElmer\\UVWinLab\\Communique.mdb
- ...\\PerkinElmer\\UVWinLab\\UVWinLab.mdb

Where "..." represents:

For Windows XP, C:\\Documents and Settings\\All Users\\Application Data

For Windows 7, C:\\Program Data

How should I archive my data?

We do not make specific recommendations on how to do this, but it is the client's responsibility to ensure that this is done.

How often should I archive my data?

It depends on use, but a suggestion would be to perform this daily.

How secure are my databases against tampering?

There are three levels of security within the main UV WinLab database:

- Data is written to the database as a Binary Large Object (BLOB) which makes it impossible to view the raw data in, for example, a text editor.
- The data is then encrypted using a proprietary encryption (RSA encryption). This is about as secure as encryption is allowed to be for civilian applications.
- Finally, the records of the database are checksummed to make them tamper-evident.

Some aspects of my system have started to run more slowly, why is this?

Assuming that your PC is of the correct specification, it could be that your Communiqué or UV WinLab databases have become very large. You should try compacting the database (using Database Tools). If this doesn't improve the situation, consider archiving your old database and starting a new one (you will need to export methods and templates from the old database(s) to the new one(s)).

Can I accidentally delete the databases?

The system insists of the file system being configured as NTFS (NT File System) during installation. This means that only Windows Administrators can delete databases. Normal users will not be able to accidentally delete protected files.

Is it possible to check my databases for areas of invalid data?

Yes. In Database Tools there is a Check Database option that allows you to do this.

If I have a checksum failure on a data record, it means that I've permanently lost that piece of data. Is that correct?

Yes. Similarly, if your data was on the hard disk and you had a catastrophic failure of the hard drive, you would also have lost your data. UV WinLab ES is technically compliant with 21 CFR part 11, but, the system administrator also has obligations to make sure that the software is being used in a compliant manner. Regular backup of the databases is part of your obligation to the FDA.

Is there a user's manual?

Yes, there is. It is provided as a .pdf file. On the toolbar of this Help file is a button called User's Guide. Click this and it will access a .pdf version of this Help file.

Methods and Tasks

The software won't let me approve my method, why is this?

Assuming you have the correct permissions to review and approve methods, you need to lock the method first.

When is the method audit trail switched on?

When the method is first locked. If you wish to revise a method, it can be unlocked, edited and locked again (this will be reflected in the audit trail).

How many people can review and approve methods?

As many as you like.

How do I calculate a mean value?

You need to set up replicates in the sample table (design). If you want to average an absorbance at 450 nm (for example), you should set up an equation:

Yval(SampleID.Replicates,450)

If this was Equation 1, then you should set up Equation 2 as follows:

Mean(Equation1)

This will then create a separate table (replicates table) that will display the mean values.

Why does the software not appear to respond when I make changes?

If you have made any changes that affect the instrument settings, the software relays these changes to the instrument. This may take several seconds, during which time any other changes to the software may not be seen. The software will update correctly when the communication with the instrument is complete.

Why do I get a straight line at 100.00 % T when I try to scan?

This indicates a detector overload. In addition, an exclamation mark is displayed next to the ordinate value in the Instrument Status Bar and an 'Invalid data: overflow reference' tooltip is also displayed. Alter the slit and/or gain settings on the Data Collection page of the Workspace.

Why can't I access the instrument properties page from manual control?

If you cannot access the Instrument Properties page from Manual Control it is likely that the lamp usage is beyond the specified limit. You will see the progress bar on the screen and a lamp warning prompt beneath the progress bar. If the lamp warning is confirmed, the Instrument Properties page is displayed but the progress bar remains.

This can be resolved by resetting the lamp usage. See [Instruments](#) for further information about changing and resetting lamps.

Why can't I edit a locked method when I have 'create and edit methods and IPV set-ups' permission?

Once a method has been locked it cannot be edited. If you wish to edit the method, you must open it by clicking the right mouse button on the Method name in the Explorer and selecting **View**, and then use the **Save As - Method** command from the File menu to save the method with a new name.

Why is sample position one used for sampling when I have added a sample to the Sample Table after my previous samples have been run, even when I have specified a different sample position?

When all the samples in the Sample Table have been run, and then a further sample is added, the default sample position is 1. You can select another position from the drop-down list. However, you must then click outside of this cell (containing the new sample position) for the software to recognize the change. If you click Run without moving out of the cell, the software will automatically use position 1.

Why can't I select substrate concentration within End Point Analysis (Rate) when I reprocess the task?

By definition, the substrate concentration calculation needs a column to be present in the Sample Table. However, it is not possible to edit the Sample Table when reprocessing a task.

What correction spectra are collected when I set the sample beam attenuator to automatic (High performance instruments only, excluding the Lambda 650 and 750 spectrometers) and then press Autozero ?

The software will collect three spectra:

- 100%T baseline – Sample beam and reference beam set to 100%
- 0%T baseline – Sample beam 0% and reference beam 100%
- Attenuator spectrum – Sample beam set to the value defined for the reference beam attenuator on the Data Collection page (for example, 1% or 10%) and reference beam set to 100%.

How do I save the attenuator correction spectra as part of a method but collect other correction spectra (baselines) when I run a task?

NOTE: This applies to High performance instruments only.

1. Create a method.
2. On the Data Collection page select automatic attenuator for the sample beam and select the value for the reference beam.
3. Select the Corrections page.
4. Select the Frequency as As required at task start or As required before next measurement.
5. Ensure all baselines are turned off.
6. Click  .
The attenuator data (correction spectra) are collected.

7. When the spectra have been collected, select the required baseline options and set the required expiry time for the attenuator corrections.

NOTE: You may also wish to stop the software from invalidating the attenuator corrections when instrument settings (for example, slit width) are changed, by selecting **Do not invalidate attenuator corrections**.

8. Save the Method with corrections.

Ensure the Save corrections check box is selected on the Save dialog.

In future when the method is run, only the baselines selected will be collected and not the attenuator correction spectra.

Why is the residual column on the results page not populated when I run a Quant method?

The residual column can only be calculated for samples in the Sample Table in the 'expected' concentration has been entered. To do this, you must add the Concentration column to the Sample Table and enter the value for all samples. The residual is this 'expected' concentration minus the analyte concentration.

Why can't I reprocess my task?

This could be due to one of two reasons:

- If you are using the Enhanced Security version of UV WinLab you may not have the necessary permission to reprocess tasks. You should contact your UV WinLab Administrator for further information about your permissions
- If the method used to create the task is locked, reviewed or approved, you will not be able to reprocess the task.

IPV (Instrument Performance Verification)

What does 'stop tests on failure' mean. Does it mean that I can't use the instrument any more?

No, it means that if you have selected more than one test as part of your IPV, it will abort further tests. This just saves you unnecessary effort.

If an instrument fails its IPV calibration, can it be put out of use by my analysts?

This depends on how you set up the permissions. If an IPV is due, it will prompt you when you run a method. If you do not have permission to perform an IPV or are logged on as an analyst, it will not let you proceed. If you are a method developer, you will be able to postpone the IPV to a more convenient time.

Why are there two identical sample and two identical block prompts in the summary for the IPV Stray light with sodium nitrite test?

In the test setup there is one sample prompt row and one block row only where the message text can be edited. However, the test is performed at two wavelengths, so the same sample and block messages are used for each wavelength. In the summary, Pre-Sample-Prompt-2 and Pre-Sample-Prompt-4 are the sample message text. Pre-Sample-Prompt-1 and Pre-Sample-Prompt-3 are the block message text.

Why do I get a full report and summary report for my IPV test(s)?

If **Report summary only** is selected, only the report summary is printed. However, if **Report summary only** is not selected, both the full report and summary report are printed by default.

How can I review and approve IPV results?

IPV results can be reviewed and approved from the Results Browser. Create an IPV query to search for the results you wish to review/approve. From the list of results displayed, select the required result(s) and from the Tools menu select **Review** or **Approve**.

Why are there empty fields in my IPV report?

For tests involving standards, the IPV report has 4 boxes for standard details regardless of the number of samples used. However, if the particular test (for example, Photometric accuracy with K₂Cr₂O₇ solution) has only one standard, 3 of the fields will be empty.

Printing

Why does my graph print badly when I have copied it from the clipboard into a report?

When you select Copy to Clipboard, 2 copies of the graph are placed on the clipboard - a bitmap and an enhanced metafile.

A bitmap of the graph is the same dimension as the current graph window. This is an EXACT copy of what is shown on the screen. It will include the information page. The graph will have the same resolution as the screen. This means, the bigger the graph is shown in UV WinLab, the bigger the bitmap and the better the resolution. This format should be used for screen reports.

The enhanced metafile (EMF) is the same physical dimension as the screen but it is produced for the default printer rather than the screen and is therefore of much higher resolution. This is the format that should be used if you wish to print the report as the curve will appear much smoother than if the bitmap is used. There is a disadvantage with the metafile. If you stretch the image, you must keep the same aspect ratio as the original image otherwise you will get overlapping characters if the image is stretched in the vertical direction but not the horizontal direction. If you perform a non-proportional stretch with a bitmap, the characters just become stretched on one direction (rather than overlapping) but the curve starts to become 'blocky'.

You need to use the **Paste Special** command within Word to see the different formats. Word gives the options: **Picture**, **Bitmap**, **Device Independent Bitmap**, and **EMF**. If you use **Paste** rather than **Paste Special**, Word will use the **Picture** format by default.

Why can I not print a report despite selecting the settings on the Output page of the Workspace?

If you are using the Enhanced Security version of UV WinLab is it likely that you do not have permission to print reports. Contact your UV WinLab Administrator for information about your permissions.