Multimode Detection Software

User's Manual

Beckman Coulter PN 987958
Revision AA
May 2004

Beckman Coulter, Inc
4300 N. Harbor Boulevard, Fullerton, CA 92834-3100

Except as provided in writing signed by an officer to Beckman Coulter, Inc., this system and any related documentation are provided “as is” without warranty of any kind, expressed or implied, including that the system is “error free.” This information is presented in good faith, but Beckman Coulter, Inc. does not warrant, guarantee, or make any representations regarding the use or the results of the use of this system and related documentation in terms of correctness, accuracy, reliability, currentness, omissions, or otherwise. The entire risk as to the use, results, and performance of this system and related documentation is assumed by the user.

Except as expressly provided herein, Beckman Coulter, Inc. makes no other warranty, whether oral or written, expressed or implied, as to any matter whatsoever, including but not limited to those concerning merchantability and fitness for a particular purpose, nor is freedom from any patent owned by Beckman Coulter, Inc. or by others to be inferred.

LIMITATIONS OF LIABILITY

Beckman Coulter, Inc. shall not be liable, to any extent whatsoever, for any damages resulting from or arising out of the use or performance of this system and related documentation or the procedures specified in this manual, regardless of foreseeability or the form of action, whether in contract, tort (including negligence), breach of warranty, strict liability or otherwise, and including but not limited to damages resulting from loss of data, loss of anticipated profits, or any special, indirect, incidental or consequential damages. In no event shall Beckman Coulter, Inc.’s liability to the user exceed the amount paid by the user to Beckman Coulter, Inc. hereunder. The user assumes full responsibility for the results obtained from the use of this system and related documentation and for application of such results.

Beckman Coulter, SAGIAN Operations
Indianapolis, Indiana
www.beckmancoulter.com

Warranty and Returned Goods Requirements

All standard Beckman Coulter, Inc. policies governing returned goods apply to this product. Subject to the exceptions and upon the conditions stated below, the Company warrants that the products sold under this sales agreement shall be free from defects in workmanship and materials for one year after delivery of the products to the original Purchaser by the Company, and if any such product should prove to be defective within such one year period, the Company agrees, at its option, either (1) to correct by repair or at the Company's election by replacement, any such defective product provided that investigation and factory inspection discloses that such defect developed under normal and proper use, or (2) to refund the purchase price. The exceptions and conditions mentioned above are as follows:

a. Components or accessories manufactured by the Company which by their nature are not intended to and will not function for one year are warranted only to reasonable service for a reasonable time. What constitutes a reasonable time and a reasonable service shall be determined solely by the Company. A complete list of such components and accessories is maintained at the factory.

b. The Company makes no warranty with respect to components or accessories not manufactured by it. In the event of defect in any such component or accessory, the Company will give reasonable assistance to Purchaser in obtaining from the manufacturer's own warranty.

c. Any product claimed to be defective must, if required by the Company, be returned to the factory, transportation charges prepaid, and will be returned to Purchaser with transportation charges collect unless the product is found to be defective, in which case the product must be properly decontaminated of any chemical, biological, or radioactive hazardous material.

d. The Company shall be released from all obligations under all warranties, either expressed or implied, if any product covered hereby is repaired or modified by persons other than its own authorized service personnel, unless such repair by others is made with the written consent of the Company.

e. If the product is a reagent or the like, it is warranted only to conform to the quantity and content and for the period (but not in excess of one year) stated on the label at the time of delivery.

It is expressly agreed that the above warranty shall be in lieu of all warranties of fitness and of the warranty of merchantability, and that the company shall have no liability for special or consequential damages of any kind or from any cause whatsoever arising out of the manufacture, use, sale, handling, repair, maintenance, or replacement of any of the products sold under the sales agreement.

Representatives and warranties made by any person, including dealers and representatives of the Company, which are consistent or in conflict with the terms of this warranty, shall not be binding upon the Company unless reduced in writing and approved by an expressly authorized officer of the Company.

Parts replaced during the warranty period are warranted to the end of the instrument warranty.

Note: Performance characteristics and specifications are only warranted when Beckman Coulter replacement parts are used.

Safety Information

All Warnings and Cautions in this document include an exclamation point, a lightning bolt, or a light burst symbol framed within a triangle. Please pay special attention to the specific safety information associated with these symbols.

WARNING: If the equipment is used in a manner not specified by Beckman Coulter, Inc., the protection provided by the equipment may be impaired.

Warning and Caution Definitions

The exclamation point symbol is an international symbol which serves as a reminder that all safety instructions should be read and understood before installation, use, maintenance, and servicing is attempted.

When this symbol is displayed in this manual, pay special attention to the specific safety information associated with the symbol.

WARNING

A WARNING calls attention to a condition or possible situation that could cause injury to the operator.

CAUTION

A CAUTION calls attention to a condition or possible situation that could damage or destroy the product or the operator's work.

Warnings and Cautions Found in this Manual

Please read and observe all cautions and instructions. Remember, the most important key to safety is to use Multimode Detection Software with care.

The WARNINGs and CAUTIONs found within this document are listed below.

WARNING: If the equipment is used in a manner not specified by Beckman Coulter, Inc., the protection provided by the equipment may be impaired.

CAUTION: Shake low density plates, such as 6- or 48-well plates, at low speed only. Shaking low density plates at higher speeds may cause liquid in wells to spill.

CAUTION: The plate height configured must not be less than that of the actual plate. Doing so may cause the DTX 880 optics transport to collide with the plate during a Read Height Optimization.

Table of Contents

2 Setting Up and Controlling Instruments

3 Creating and Editing Detection Methods

4 Creating and Editing Labware

5 Defining, Editing, and Running Protocols

List of Figures

1 Installing, Setting Up, and Using the Software

1.1 Overview

Multimode Detection Software configures and controls all measurement protocols and operations performed by the DTX 800 and DTX 880 Multimode Detectors. Absorbance, glow luminescence, fluorescence intensity (top and bottom), fluorescence polarization, and time-resolved fluorescence (TRF) measurements are all supported. The measurement methods available to users depend on the capabilities of the instrument being controlled. Measurement results are easily exported and may be evaluated with an external application such as Microsoft® Excel.

The simple interface divides the main window into three basic sections, which provide easy access to system functionality. Most operations use a wizard-type interface to efficiently guide the task being performed.

This user's manual provides comprehensive coverage of Multimode Detection Software functionality, including:

- *Setting Up and Controlling Instruments* (Chapter 2).
- *Creating and Editing Detection Methods* (Chapter 3).
- *Creating and Editing Labware* (Chapter 4).
- *Defining, Editing, and Running Protocols* (Chapter 5).
- *Viewing Measurement Results* (Chapter 6).

This chapter introduces users to the software and provides instructions for:

- *Using Multimode Detection Software* (Section 1.2).
- *Installing Multimode Detection Software* (Section 1.3).
- *Setting Up Multimode Detection Software* (Section 1.4).

1.2 Using Multimode Detection Software

Multimode Detection Software uses a simple interface that divides the main window into three basic sections: navigation pane, toolbar, and selection and configuration pane (Figure 1-1). The interface includes easy access to the five modules that provide system functionality and a comprehensive, context-sensitive online help system.

This section covers:

- *Launching the Software* (Section 1.2.1).
- *Using the Software Interface* (Section 1.2.2).
- *Accessing Online Help* (Section 1.2.3).

1.2.1 Launching the Software

To launch Multimode Detection Software:

From the Windows® Start menu, choose **Programs>Beckman Coulter, Inc>Multimode Detection Software>Multimode Detection Software**. Multimode Detection Software appears (Figure 1-1).

Note: If Multimode Detection Software is not found in the Start menu, the software may have been installed for a single user account on the system instead of all accounts. Check with the site system administrator or login to the user account with permission to access the software. Refer to Section 1.3.2, *Using the Setup Program to Install the Software*, for more information about installing the software for a single or multiple user accounts.

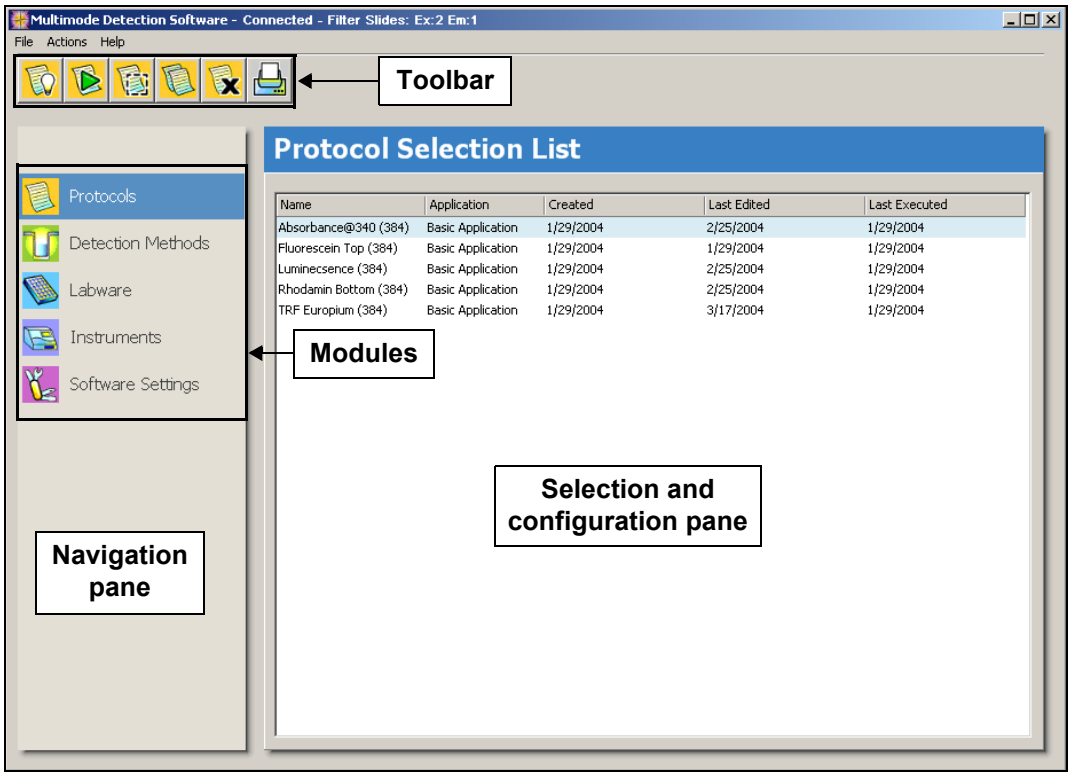


Figure 1-1. Multimode Detection Software main window

1.2.2 Using the Software Interface

Multimode Detection Software uses a simple interface that is divided into three basic sections:

- Navigation pane (refer to Section 1.2.2.1, *About the Navigation Pane*).
- Toolbar (refer to Section 1.2.2.2, *About the Toolbar*).
- Selection and configuration pane (refer to Section 1.2.2.3, *About the Selection and Configuration Pane*).

The navigation pane provides access to the five modules that provide the majority of the functionality built into the software. The options available in the toolbar and selection and configuration pane are determined by the module currently selected in the navigation pane.

1.2.2.1 About the Navigation Pane

The navigation pane is the narrow pane on the left of the Multimode Detection Software window (Figure 1-1). Use the navigation pane to switch between modules:

- **Protocols** — opens the Protocol Selection List and provides the ability to define, run, edit, copy, delete, and print measurement protocols (refer to Chapter 5, *Defining, Editing, and Running Protocols*).
- **Detection Methods** — opens the Detection Method Selection List and provides the ability to create, edit, copy, and delete detection methods (refer to Chapter 3, *Creating and Editing Detection Methods*).
- **Labware** — opens the Labware Selection List and provides the ability to create, edit, optimize, copy, and delete labware types (refer to Chapter 4, *Creating and Editing Labware*).
- **Instruments** — opens the Instrument Selection List and provides the ability to manually control instrument functions, such as shaking, and configure communications settings and filter slides (refer to Chapter 2, *Setting Up and Controlling Instruments*).
- **Software Settings** — configures software settings, including system language, print options, default simulated data files, and the directory where measurement results are saved (refer to Section 1.4.2, *Configuring Software Settings*).

1.2.2.2 About the Toolbar

The toolbar provides easy access to common software functions. Functions available in the toolbar are specific to the module chosen in the navigation pane. For example, Optimize Labware is only available when the Labware module is active.

1.2.2.3 About the Selection and Configuration Pane

The selection and configuration pane is the large pane to the right of the navigation pane. Options available in this pane change depending on which module is currently selected in the navigation pane. For example, when Protocols is selected, the Protocol Selection List is displayed, which provides access to configured protocols and functionality specific to the Protocols module.

1.2.3 Accessing Online Help

Multimode Detection Software contains a detailed online help system that covers defining and editing labware, detection methods, and protocols, performing measurements, and exporting measurement results. The online help is context sensitive, which provides instant access to help for the active screen.

To access online help:

Press **F1** at any time to display online help for the active screen (Figure 1-2).

OR

From the Help menu, choose **Help** to display the table of contents.

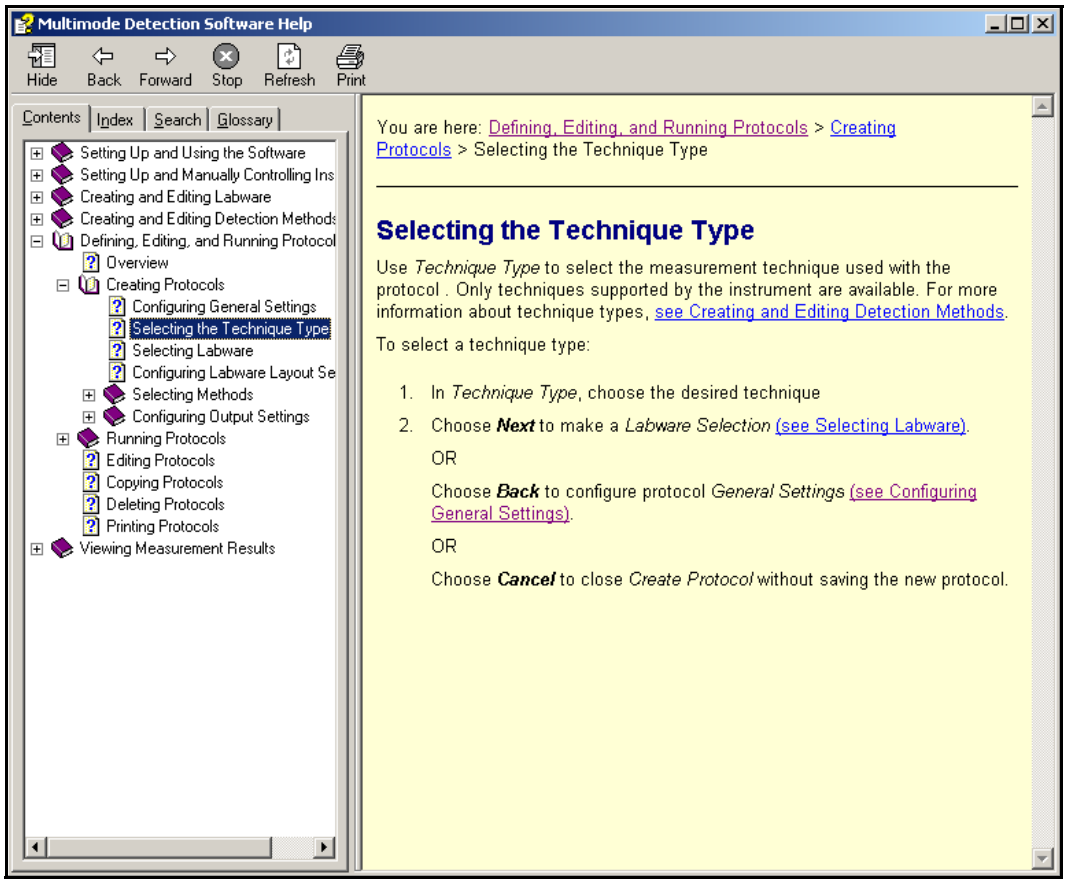


Figure 1-2. Using online help

1.3 Installing Multimode Detection Software

Installing Multimode Detection Software requires:

- *Meeting System Requirements* (Section 1.3.1).
- *Using the Setup Program to Install the Software* (Section 1.3.2).

1.3.1 Meeting System Requirements

To install and use Multimode Detection Software successfully, the host computer must meet the minimum system requirements listed in Table 1-1. Where relevant, Table 1-1 also lists recommended specifications.

Table 1-1. Host computer System Requirements

Component	Minimum Requirements
CPU	Pentium® II 300 Mhz
RAM	128 MB minimum 256 MB recommended
Hard Drive	150 MB free space
CD-ROM Drive	4X
Monitor	640x480 resolution
Keyboard	101 key
Mouse	IBM® compatible
Serial Port	1 free serial port
Operating Systems	Windows NT® 4 (Service Pack 6a) Windows® 2000 Windows® XP
Operating System Languages	English (U.S.), German
Web Browser	Internet Explorer 5.01 or later

1.3.2 Using the Setup Program to Install the Software

The setup program on the installation CD installs all of the components required to run Multimode Detection Software.

Note: The setup program also provides the ability to repair the software installation or remove (uninstall) the software from the host computer (refer to Section 1.3.3, *Repairing or Removing the Multimode Detection Software Installation*).

To install Multimode Detection Software:

1. Exit all open Windows programs before running the setup program.
2. Make sure the current user account has Administrator privileges. Accounts with Standard or Restricted access are not permitted to run the setup program. Contact the site system administrator for more information about account privileges.

3. Insert the Multimode Detection Software installation CD into the CD-ROM drive. After a few seconds, Language Options appears (Figure 1-3).

Note: If Language Options does not appear automatically within a few seconds of inserting the CD into the drive, use Windows Explorer to browse to the CD-ROM drive and run MultimodeDetectionSoftwareInstall.exe.

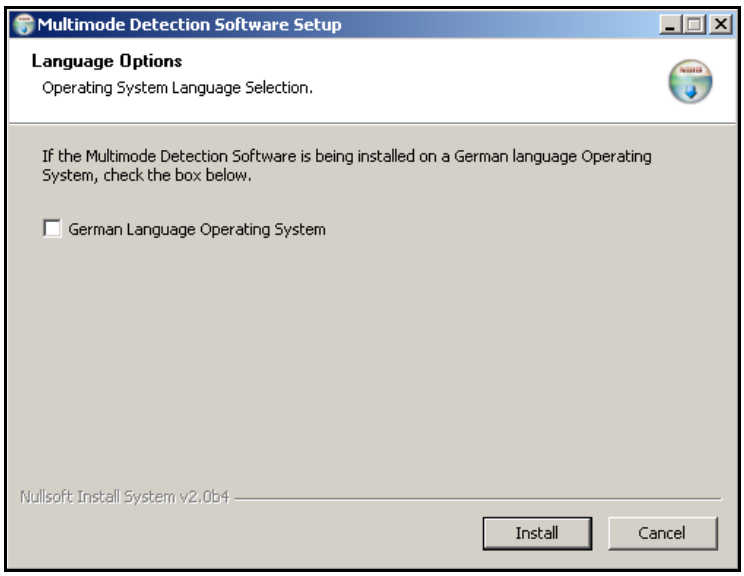


Figure 1-3. Installer Language Options

4. If installing the software on a German language version of Windows, choose German Language Operating System. This ensures that all software screens are displayed correctly when German is set as the system language used in the Multimode Detection Software interface.
5. Choose **Install** to start the installation. Multimode Detection Software Installation appears (Figure 1-4).

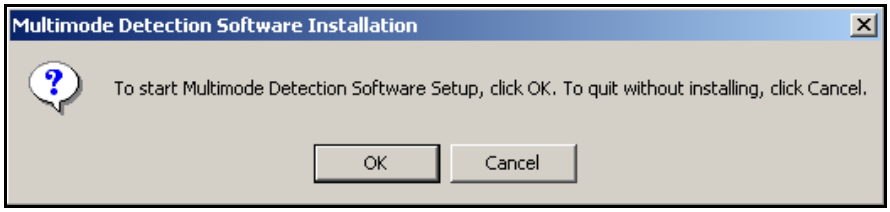


Figure 1-4. Starting the software installation

6. Choose **OK** to start the installation. The Multimode Detection Software setup wizard appears (Figure 1-5).

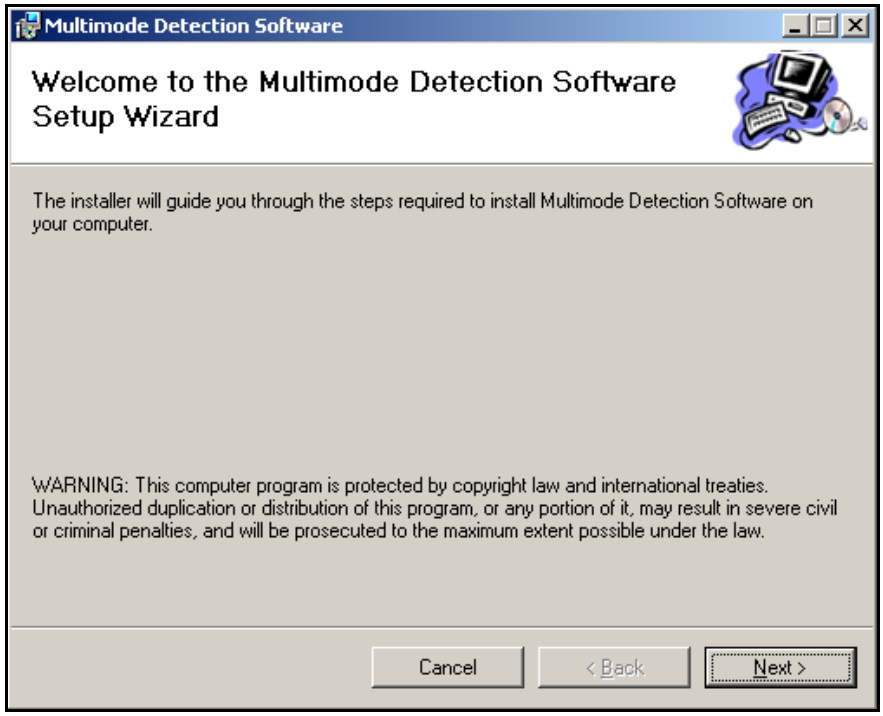


Figure 1-5. Setup wizard

7. Choose **Next** to continue the installation. Select Installation Folder appears (Figure 1-6).

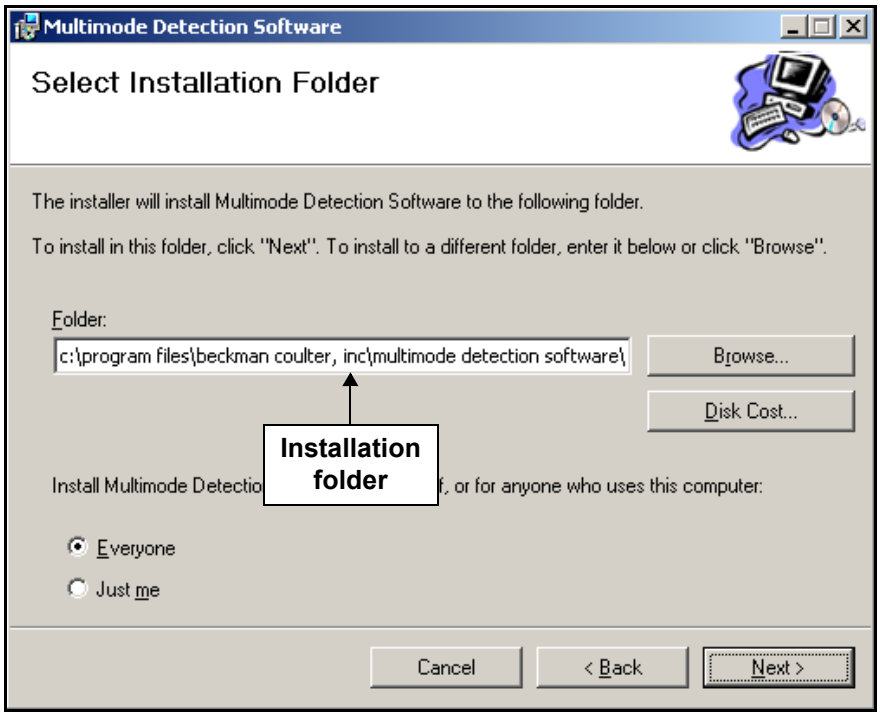


Figure 1-6. Selecting installation options

8. Choose **Browse** to select a different installation folder for the software, if desired.
9. Choose **Disk Cost** to view the free space available on all hard drives installed in the computer, if desired.
10. Select **Everyone** to allow all user accounts configured on the computer to access and use the software. Everyone should be selected when more than one user will be using the software to perform measurements.

    OR

    Select **Just Me** to allow only the user account currently logged in to access and use the software.

    Note: Only user accounts with Administrator or Standard access may use Multimode Detection Software. User accounts with Restricted access do not have access to the software.

11. Choose **Next** to continue the installation.

    OR

    Choose **Back** to return to the previous screen in the setup wizard.

    OR

    Choose **Cancel** to close the setup wizard without installing the software.
12. Follow the remaining instructions in the setup wizard to complete the software installation. When installation is complete, communications and system settings may be configured (refer to Section 1.4, *Setting Up Multimode Detection Software*).

1.3.3 Repairing or Removing the Multimode Detection Software Installation

Along with performing the initial Multimode Detection Software installation, the setup program also provides the ability to:

- Repair the installation if required software components or files are missing or damaged. Try repairing the installation if the software will not open or does not run correctly.
- Remove (uninstall) Multimode Detection Software from the host computer.

Note: Multimode Detection Software may also be removed from the computer using the Add/Remove Programs control panel in Windows®.

To repair or remove the software:

1. Exit all open Windows programs before running the setup program.
2. Make sure the current user account has Administrator privileges. Accounts with Standard or Restricted access are not permitted to run Setup. Contact the site system administrator for more information about account privileges.
3. Insert the Multimode Detection Software installation CD into the CD-ROM drive. After a few seconds, Language Options appears.

   Note: If Language Options does not appear automatically within a few seconds of inserting the CD into the drive, use Windows Explorer to browse to the CD-ROM drive and run MultimodeDetectionSoftwareInstall.exe.

4. If repairing software installed on a German language version of Windows, choose German Language Operating System. This ensures that all software screens are displayed correctly when German is set as the system language used in the Multimode Detection Software interface.
5. In Language Options, choose **Install** to start the installation. Multimode Detection Software Installation appears.
6. Choose **OK** to continue. The Multimode Detection Software setup wizard appears.
7. Choose **Next** to continue. Options to repair or remove the software appear (Figure 1-7).

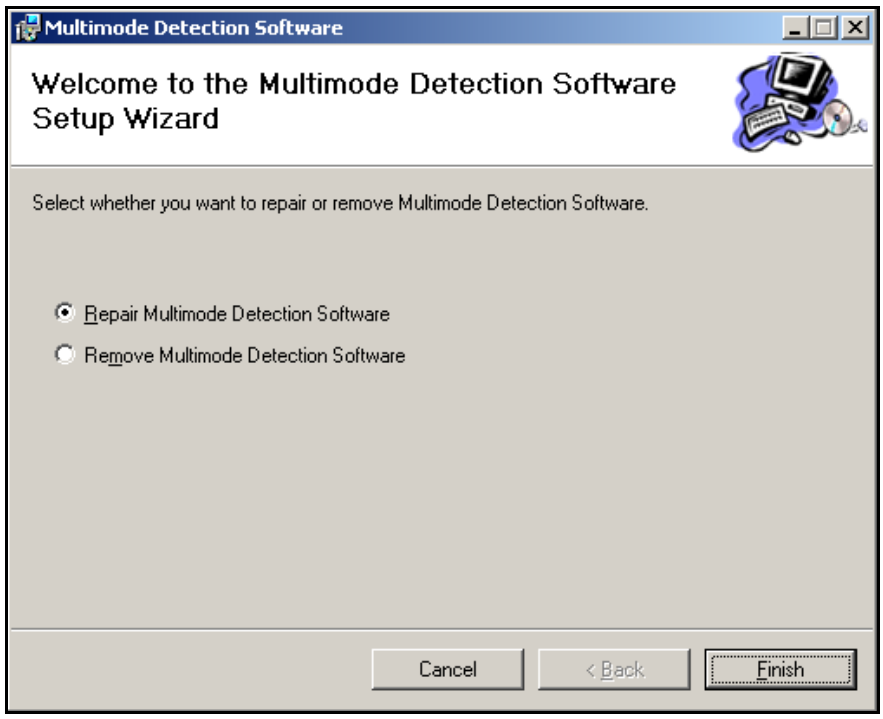


Figure 1-7. Setup wizard — repair or remove the software installation

8. Select **Repair Multimode Detection Software** to search for and automatically reinstall any missing or damaged files required to run the software.

   OR

   Select **Remove Multimode Detection Software** to remove the application and support files from the host computer.

   Note: When removing the software, only files installed by the setup program during the initial installation are removed. Files created after the installation, such as exported measurement results and labware, detection method, and protocol definitions, are not removed. Measurement results files may still be viewed after the software is uninstalled. All file types may be copied, archived, or manually deleted.

9. Choose **Finish** to perform the selected action.

   OR

   Choose **Cancel** to close the setup wizard without repairing or removing the software.

1.4 Setting Up Multimode Detection Software

After installing the software and physically connecting the instrument to a serial port on the host computer, communications and software settings must be set up.

Setting up the software includes:

- *Configuring Communication Settings* (Section 1.4.1).
- *Configuring Software Settings* (Section 1.4.2).

To set up Multimode Detection Software:

1. From the Windows® Start menu, choose **Programs>Beckman Coulter, Inc>Multimode Detection Software>Multimode Detection Software**. Multimode Detection Software appears (Figure 1-8).

Note: If Multimode Detection Software is not found in the Start menu, the software may have been installed for a single user account on the system instead of all accounts. Check with the site system administrator or login to the user account with permission to access the software. Refer to Section 1.3.2, *Using the Setup Program to Install the Software*, for more information about installing the software for a single or multiple user accounts.

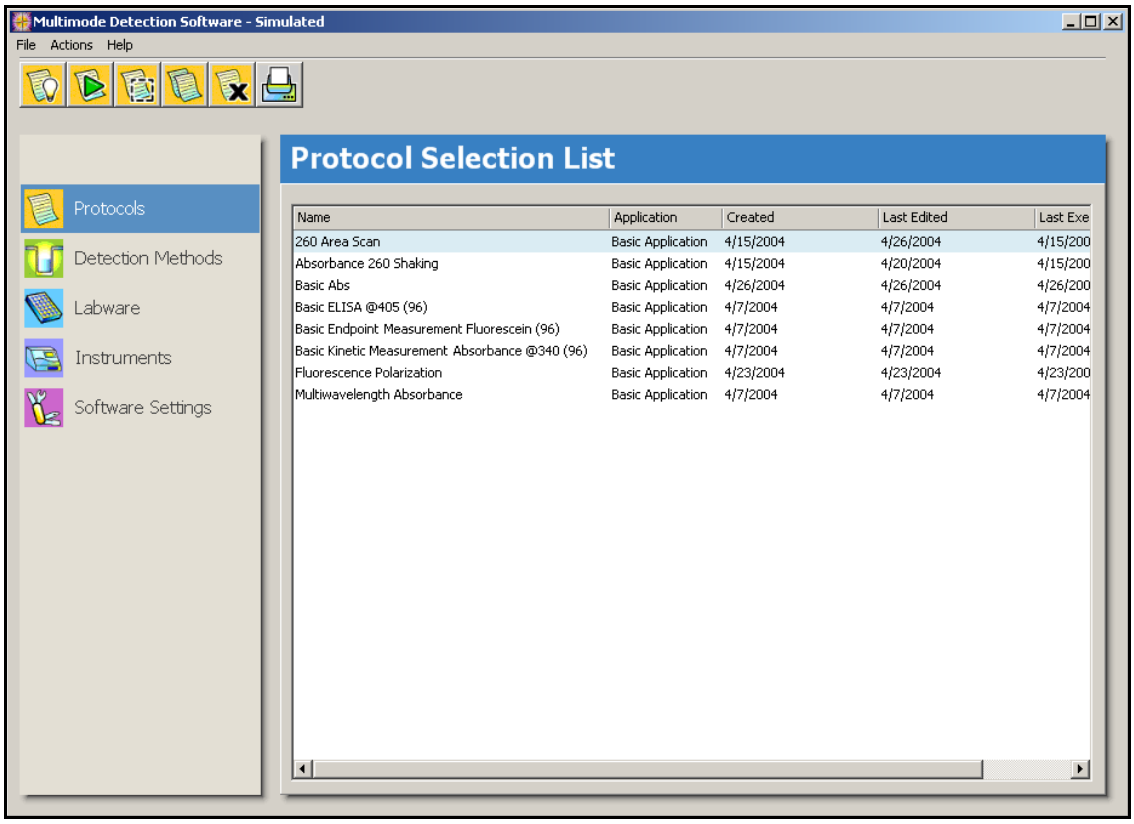


Figure 1-8. Multimode Detection Software main screen

2. If the Protocol Selection List (Figure 1-8) appears immediately, the instrument was automatically detected by the software. Proceed to Section 1.4.2, *Configuring Software Settings*, to configure system settings.

   OR

   If Warning appears (Figure 1-9), the instrument was not detected by the software. Choose **OK** to work in simulation mode until the communications settings are configured properly.

Figure 1-9. Warning — Instrument is not connected

3. If the instrument was not detected, configure the communications settings following the steps in Section 1.4.1, *Configuring Communication Settings*.
4. Configure system settings, such as language and printing options, following the steps in Section 1.4.2, *Configuring Software Settings*.

1.4.1 Configuring Communication Settings

After physically connecting the instrument to a serial communications port on the host computer, the selected serial port and communications speed must be configured correctly before Multimode Detection Software can control instrument operations. Communications settings are configured in Instrument Settings.

Before communications with an instrument are configured, a simulated instrument (Serial # 0000) appears in the Instrument Selection List (Figure 1-10). When communications are configured correctly and the instrument is detected by the software, the simulated instrument is replaced in the list by the connected instrument.

To configure communications settings:

1. From the navigation pane, choose **Instruments**. The Instrument Selection List appears (Figure 1-10).

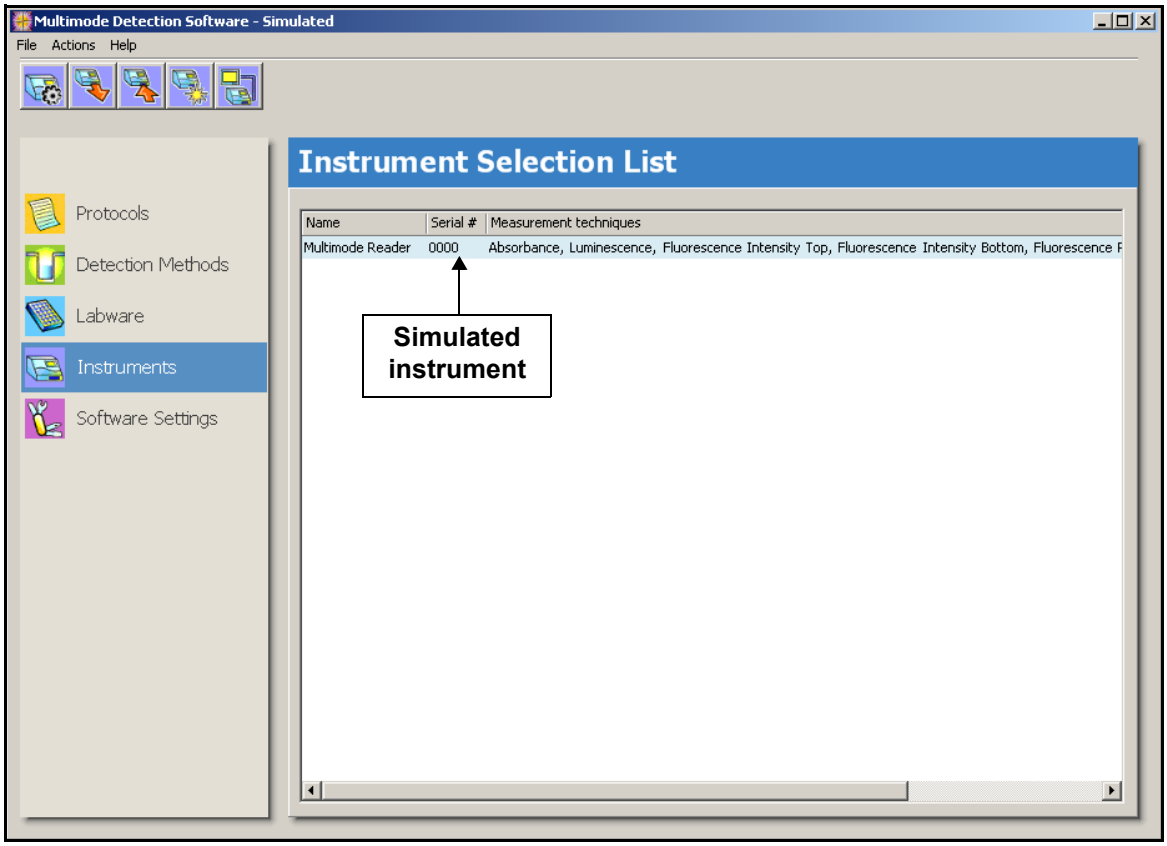


Figure 1-10. Instrument Selection List with simulated instrument selected

2. From the toolbar, choose **Instrument Settings**. Instrument Settings appears (Figure 1-11).

   OR

   From the Actions menu, choose **Instrument Settings**.

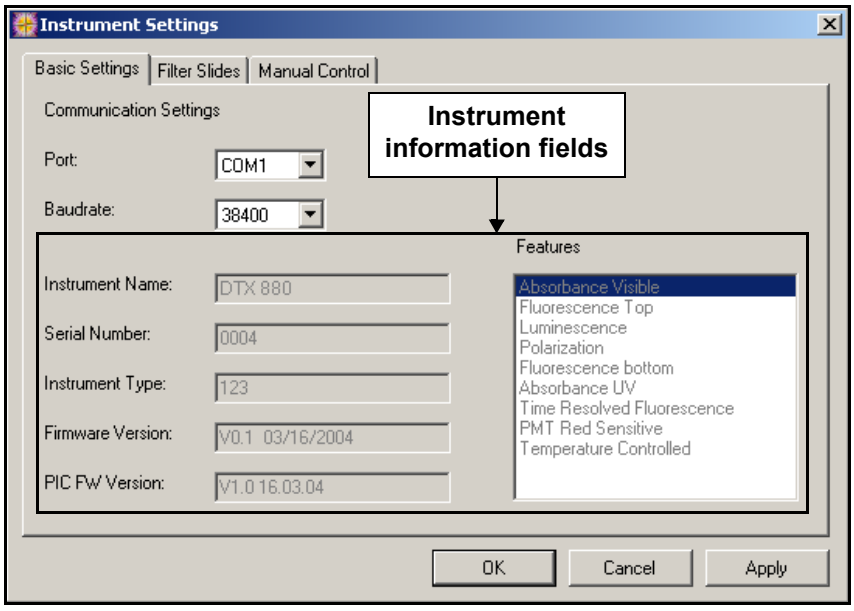


Figure 1-11. Configuring communications settings

3. Select **Basic Settings** to display the Basic Settings tab, if necessary.
4. In Port, select the serial communications port on the host computer that connects to the instrument.
5. In Baudrate, select **38400**.
6. Choose **Apply** to detect the instrument. When the instrument is detected, the instrument information fields are autopopulated with information about the connected instrument. Refer to Table 2-1 for more information about each field.
7. Choose **OK** to close Instrument Settings.

1.4.2 Configuring Software Settings

Multimode Detection Software can be customized using the options available in Software Settings (Figure 1-12). Use the tabs in Software Settings to configure the system language, printout options, default simulated data files, and directory where measurement results are stored.

To configure Software Settings:

1. From the navigation pane, choose **Software Settings**. Software Settings appears (Figure 1-12).

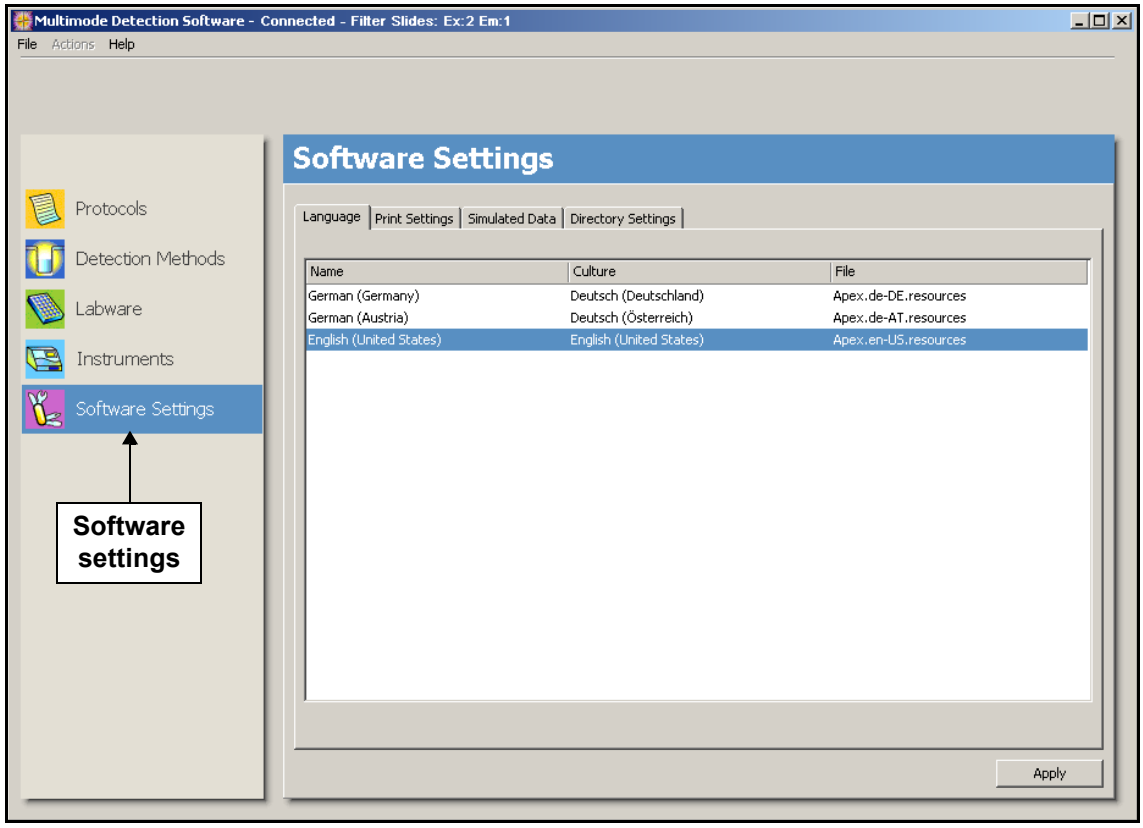


Figure 1-12. Configuring Software Settings

2. Configure the settings in the desired tab(s):
   - *Setting the System Language* (Section 1.4.2.1).
   - *Configuring Print Settings* (Section 1.4.2.2).
   - *Choosing Simulated Data Files* (Section 1.4.2.3).
   - *Choosing the Directory Where Measurement Results are Saved* (Section 1.4.2.4).

1.4.2.1 Setting the System Language

English is set as the default system language when Multimode Detection Software is installed, even when German Language Operating System is selected during installation (refer to Section 1.3.2, *Using the Setup Program to Install the Software*). The system language may be changed in the Language tab (Figure 1-13).

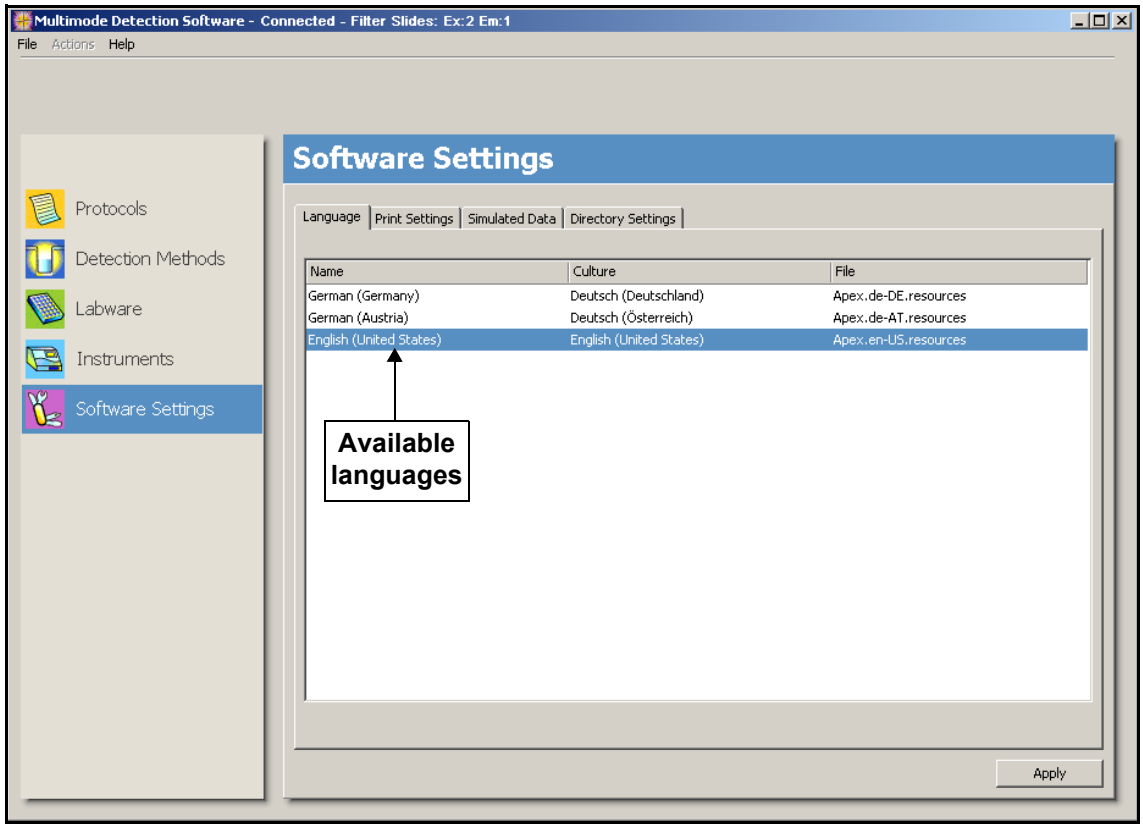


Figure 1-13. Choosing the software interface language

To change the system language:

1. Choose the **Language** tab. All available languages are listed, with the current system language highlighted (Figure 1-13).
2. Select the desired language.
3. Choose **Apply**. The system language changes to the language selected.

1.4.2.2 Configuring Print Settings

Measurement results and protocol configurations may be printed. Printing parameters, such as headers and footers are configured in the Print Settings tab (Figure 1-14).

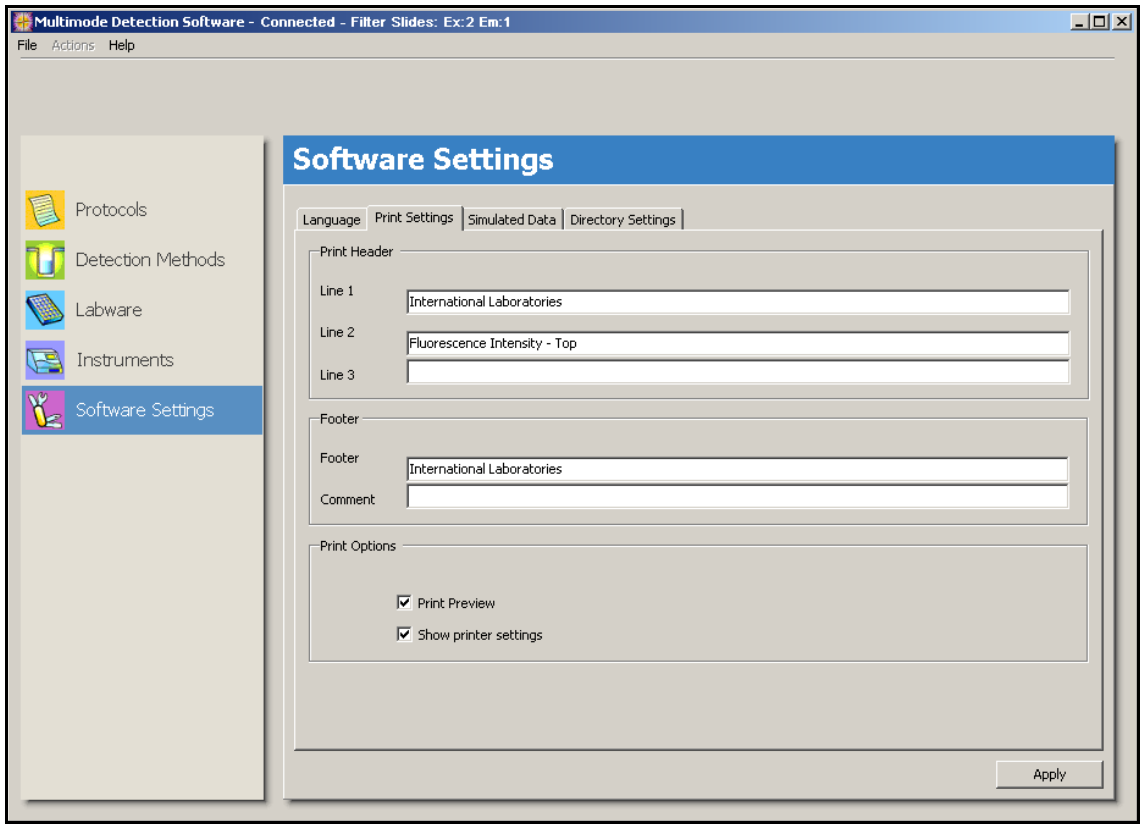


Figure 1-14. Configuring Print Settings

To configure print settings:

1. Choose the **Print Settings** tab. Print Settings appears (Figure 1-14).
2. In Print Header, enter text for each header **Line**, as desired. Header lines may be left blank.
3. In Footer, enter text for the **Footer** and **Comment**, as desired. The comment appears on printed pages below the footer. The footer and comment may be left blank.
4. In Print Options, select **Print Preview** to preview the page layout onscreen each time a protocol or measurement results are printed.
5. In Print Options, choose **Show printer settings** to display printing options each time a protocol or measurement results are printed.
6. Choose **Apply** to save the new print settings.

1.4.2.3 Choosing Simulated Data Files

Protocols may be run in simulation mode, which allows the protocol configuration to be tested using simulated data before performing the protocol on actual samples. In simulation mode, all features for the instrument type currently selected in the Instrument Selection List are available, but measurement results are either randomly generated by the software or read from a data file.

Use Simulated Data to choose the default data files for simulated absorbance, luminescence, and fluorescence measurements (Figure 1-15).

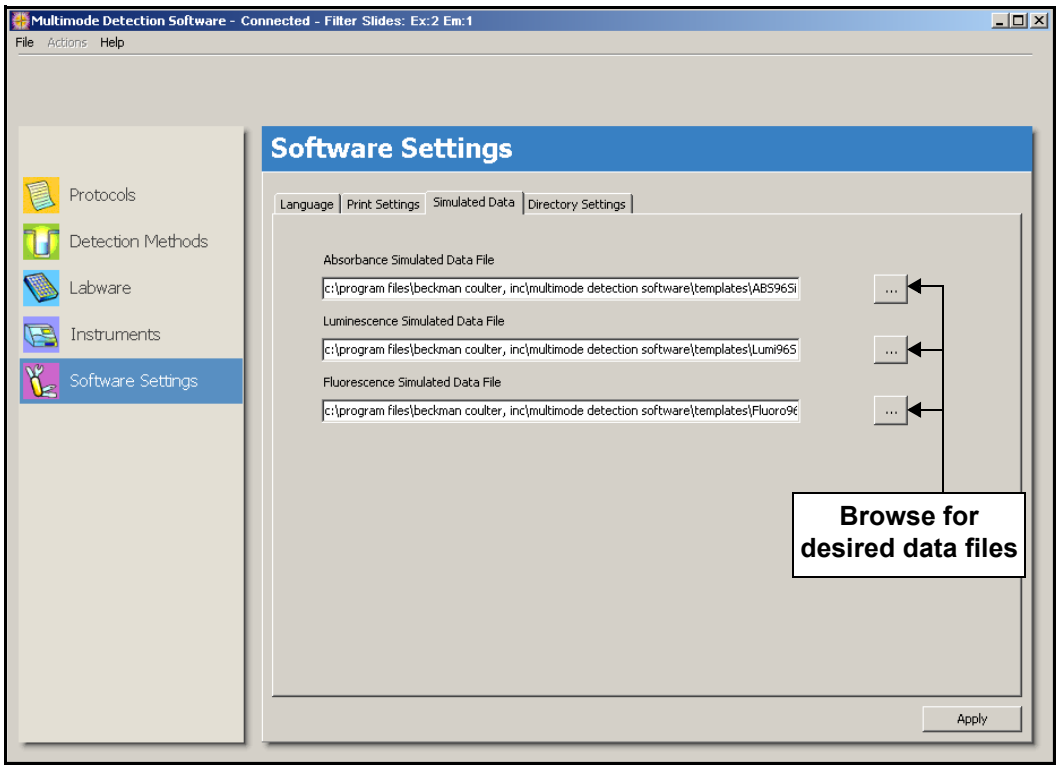


Figure 1-15. Choosing the Simulated Data files

To choose different simulated data files:

1. Choose the **Simulated Data** tab. Simulated Data appears (Figure 1-15).
2. In the desired field, enter the full path to the new simulated data file; for example, **c:\program files\beckman coulter, inc\multimode detection software\templates\DefaultDataFile.dat**. Any data file with the .dat extension may be selected, including prior measurement results. Proceed to step 5.

   OR

   Choose the **Browse** button next to the desired measurement type. Open appears.

   Note: Simulated data files are used when the number of measurement points in the simulated protocol run is the same as those present in the data file. When the number of measurement points is different, the software generates random data.

3. In Open, browse to desired data file. Any data file with the .dat extension may be selected, including prior measurement results.

4. Choose **Open** to select the data file and return to Software Settings.

   OR

   Choose **Cancel** to close Open without selecting a different data file.

5. Repeat steps 2 – 4 for each simulated data file desired to change.
6. Choose **Apply** to set the new default data file(s).

1.4.2.4 Choosing the Directory Where Measurement Results are Saved

All exported measurement results files, regardless of format, are saved in a single directory. The default storage directory is c:\program files\beckman coulter, inc\multimode detection software\data\. Use Directory Settings to change the storage directory, if desired (Figure 1-16).

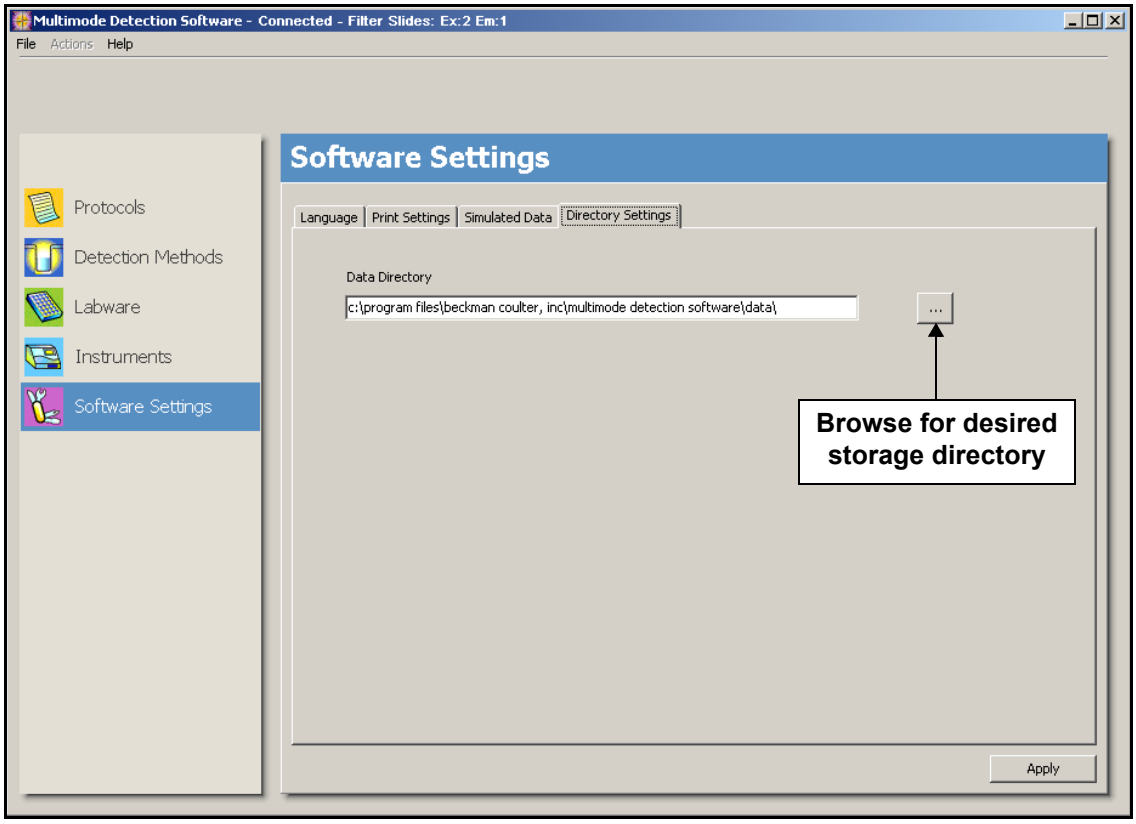


Figure 1-16. Choosing the directory where data is saved

To choose a different storage directory:

1. In Data Directory, enter the complete path of the desired storage directory; for example, **C:\documents\measurement results\DTX 880\MyResults**. Proceed to step 4.

   Choose **Browse for a directory**. Open appears.

2. In Open, browse to desired directory.

3. Choose **Open** to select the directory and return to Output Settings.

   OR

   Choose **Cancel** to close Open without selecting a new storage directory.

4. Choose **Apply** to set the new storage directory.

2 Setting Up and Controlling Instruments

2.1 Overview

Before defining measurement protocols, detection methods, and labware, or running protocols on an instrument using Multimode Detection Software, the instrument must be set up and configured. All configured instruments are listed separately in the Instrument Selection List.

The Instrument Selection List contains quick access to control common instrument functions, such as loading and ejecting the plate carrier, and to configure instrument settings.

Use the Instrument Selection List for:

- *Controlling Instrument Functions* (Section 2.2).
- *Configuring Instrument Settings* (Section 2.3).

To configure and manually control instruments:

From the navigation pane, choose **Instrument**. The Instrument Selection List appears (Figure 2-1).

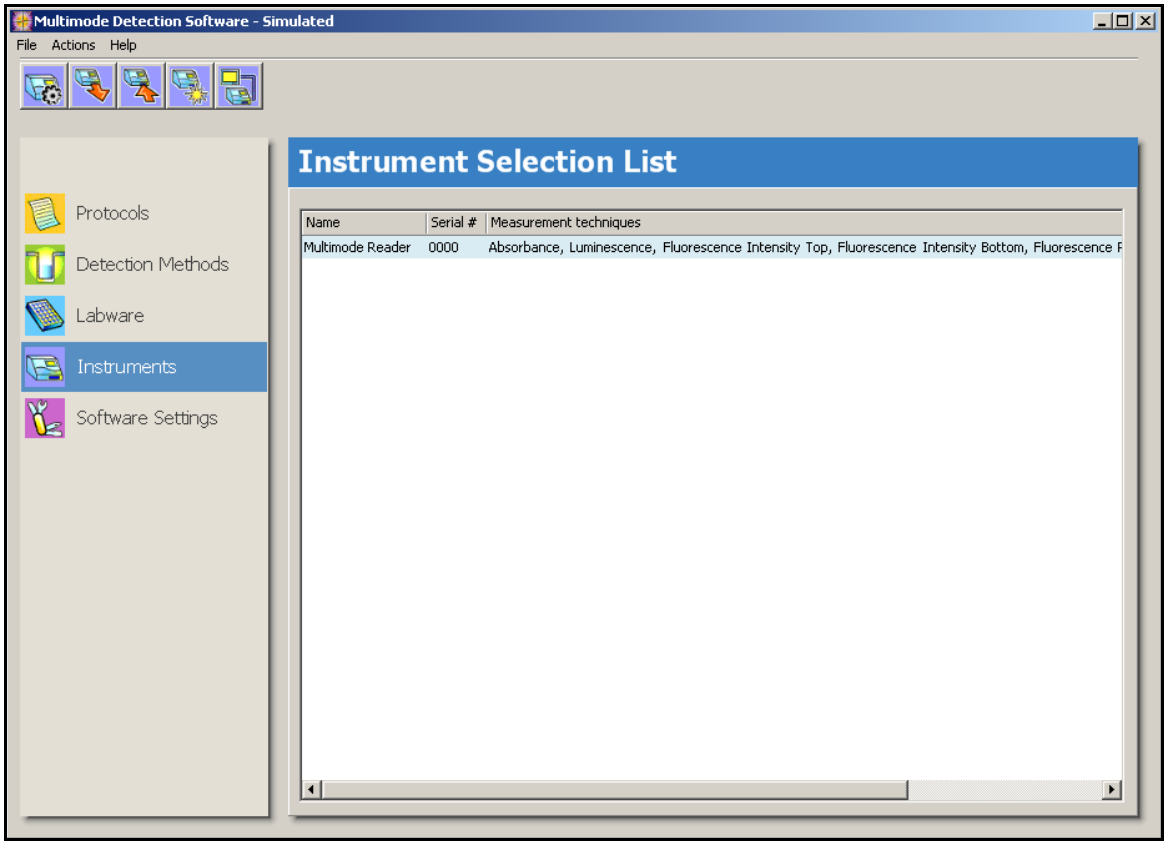


Figure 2-1. Instrument Selection List

All instruments that have been connected to the host computer and configured in the software are displayed in the Instrument Selection List. When an instrument not currently connected to the computer is selected, the software automatically enters simulation mode (refer to Section 2.2.5, *Enabling Simulation Mode*). This allows protocols, detection methods, and labware to be defined, edited, and tested for the selected instrument even though it is not physically connected to the host computer.

2.2 Controlling Instrument Functions

Many common instrument functions, such as ejecting and loading the plate carrier and initializing the instrument, can be performed directly from the Instrument Selection List using the buttons on the toolbar.

Controlling instrument functions includes:

- *Ejecting the Plate Carrier* (Section 2.2.1).
- *Loading the Plate Carrier* (Section 2.2.2).
- *Initializing the Instrument* (Section 2.2.3).
- *Connecting to the Instrument* (Section 2.2.4).
- *Enabling Simulation Mode* (Section 2.2.5).

2.2.1 Ejecting the Plate Carrier

Eject the plate carrier moves the plate carrier outside the instrument to allow access for placement or removal of a microplate.

To eject the plate carrier:

1. In the Instrument Selection List, select the desired instrument (Figure 2-1).

2. From the toolbar, choose **Eject the plate carrier**.

   OR

   From the Actions menu, choose **Eject the plate carrier**.

   OR

   Right-click on the desired instrument and choose **Eject the plate carrier** from the menu that appears.

2.2.2 Loading the Plate Carrier

Load the plate carrier retracts the plate carrier and microplate back inside the instrument in preparation of performing a measurement.

To load the plate carrier:

1. In the Instrument Selection List, select the desired instrument (Figure 2-1).

2. From the toolbar, choose **Load the plate carrier**.

   OR

   From the Actions menu, choose **Load the plate carrier**.

   OR

   Right-click on the desired instrument and choose **Load the plate carrier** from the menu that appears.

2.2.3 Initializing the Instrument

Initializing the instrument moves the optics and microplate transports to home positions. Each time the instrument is turned on, an initialization procedure is automatically performed. If necessary, the instrument may be initialized manually; for example, after an emergency stop has been performed (refer to the *DTX 800/880 Multimode Detectors User's Manual,* Section 1.3.1.1, *Performing an Emergency Stop*).

Note: When a hardware error occurs, turning the instrument off and on is recommended. The instrument initializes automatically on power up, and will not require manually initializing the instrument using the software.

To manually initialize the instrument:

1. In the Instrument Selection List, select the desired instrument (Figure 2-1).
2. From the toolbar, choose **Initialize the instrument**.

OR

From the Actions menu, choose **Initialize the instrument**.

OR

Right-click on the desired instrument and choose **Initialize the instrument** from the menu that appears.

2.2.4 Connecting to the Instrument

When Multimode Detection Software is launched, it automatically connects to the instrument or starts in simulation mode if no instrument is detected. The connection to the instrument may be established manually after physically connecting a different instrument to the computer, or when switching from simulation mode (refer to Section 2.2.5, *Enabling Simulation Mode*).

To connect to the instrument:

1. In the Instrument Selection List, select the desired instrument (Figure 2-1).
2. From the toolbar, choose **Connect to the instrument**. The button remains depressed while the instrument is connected and not in simulation mode.

OR

From the Actions menu, choose **Connect to the instrument**.

OR

Right-click on the desired instrument and choose **Connect to the instrument** from the menu that appears.

2.2.5 Enabling Simulation Mode

Multimode Detection Software can operate in simulation mode whether or not an instrument is connected. Simulation mode enables all features supported by the instrument currently selected in the Instrument Selection List, but measurement results are randomly generated by the software or read from a file. Refer to Section 1.4.2.3, *Choosing Simulated Data Files*, for more information about selecting simulated data files.

To enable simulation mode:

1. In the Instrument Selection List, choose the desired instrument.

2. From the toolbar, choose **Simulate the current instrument**. The button remains depressed while the instrument is in simulation mode.

   OR

   From the Actions menu, choose **Simulate the current instrument**.

   OR

   Right-click on the desired instrument and choose **Simulate the current instrument** from the menu that appears.

2.3 Configuring Instrument Settings

Communication settings and filter slides for the instrument are configured in Instrument Settings. Configuring the instrument in Instrument Settings informs Multimode Detection Software about the instrument, such as the communication speed and port used, and the configuration of filter slides and individual filters. Instrument features, such as microplate shaking, may also be controlled manually.

To configure instrument settings:

1. Select the desired instrument in Instrument Selection List.

2. From the toolbar, choose **Instrument Settings**. Instrument Settings appears.

   OR

   From the Actions menu, choose **Instrument Settings**.

   OR

   Right-click on the desired instrument and choose **Instrument Settings** from the menu that appears.

3. Configure instrument settings on the three tabs as described in the following sections:

   - *Configuring Basic Instrument Settings* (Section 2.3.1).
   - *Defining and Editing Filter Slides* (Section 2.3.2).
   - *Manually Controlling the Instrument* (Section 2.3.3).

2.3.1 Configuring Basic Instrument Settings

The Basic Settings tab (Figure 2-2) contains information about the connection from the instrument to the operating computer, as well as identifying information about the instrument. The Port and Baudrate must be configured correctly to enable Multimode Detection Software to communicate with the instrument.

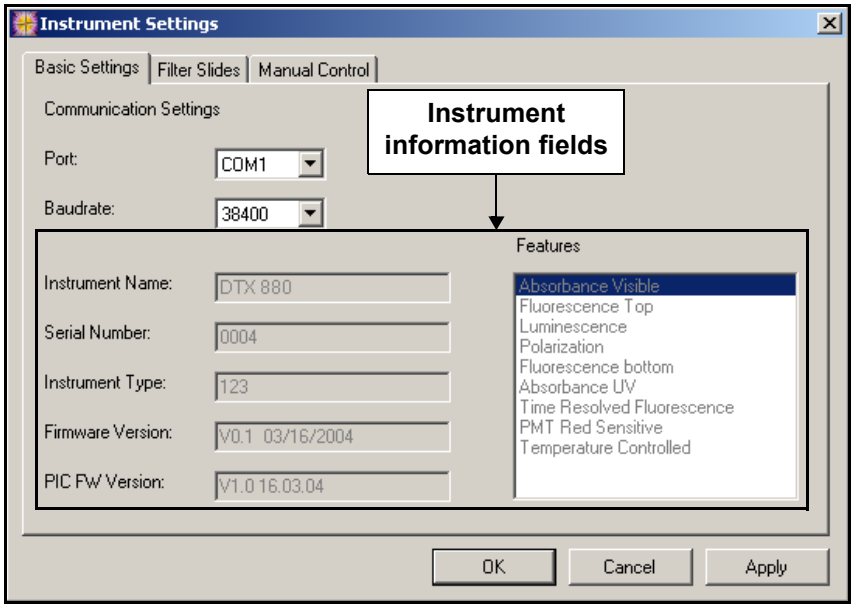


Figure 2-2. Instrument Settings — Basic Settings

To configure basic instrument settings:

1. In Instrument Settings, select **Basic Settings** to display the Basic Settings tab, if necessary.
2. In Port, select the serial communications port on the host computer that connects to the instrument.
3. In Baudrate, select **38400**.
4. Choose **Apply** to detect the instrument. When the instrument is detected, the instrument information fields are autopopulated with information about the connected instrument. Refer to Table 2-1 for more information about each field.

Table 2-1. Basic Settings Read-Only Fields

Field	Description
Instrument Name	The model of the instrument.
Serial Number	The serial number of the instrument.
Instrument Type	A numerical code used by the software to identify the instrument type.
Firmware Version	The version of firmware loaded for the instrument.
PIC FW Version	The version of firmware loaded for the PIC processor of the instrument.
Features	The types of measurements the instrument is capable of performing.

2.3.2 Defining and Editing Filter Slides

The Filter Slides tab (Figure 2-3) is used to add, remove, and configure filter slides and the filters installed on a filter slide.

Filters used to perform measurements are mounted on two types of interchangeable slides. One slide is reserved for excitation filters used in absorbance and fluorescence measurements; the other is used for emission filters used in fluorescence and some luminescence measurements. Excitation and emission filter slides are different sizes to prevent them from being installed in the incorrect position. Each slide can hold up to six filters.

When a slide is exchanged, an identification code built into the slide allows the Multimode Detection Software to recognize the new slide and filter configuration. When a slide with a new configuration is inserted, or the filters on a slide change, the slide must be configured in the Filter Slides tab. Up to 31 excitation filter slides and 31 emission filter slides may be stored in Multimode Detection Software at one time.

Note: Refer to the *DTX 800/880 Multimode Detectors User's Manual,* Section 1.3.2, *Exchanging Filter Slides and Individual Filters*, for information on installing filters and filter slides.

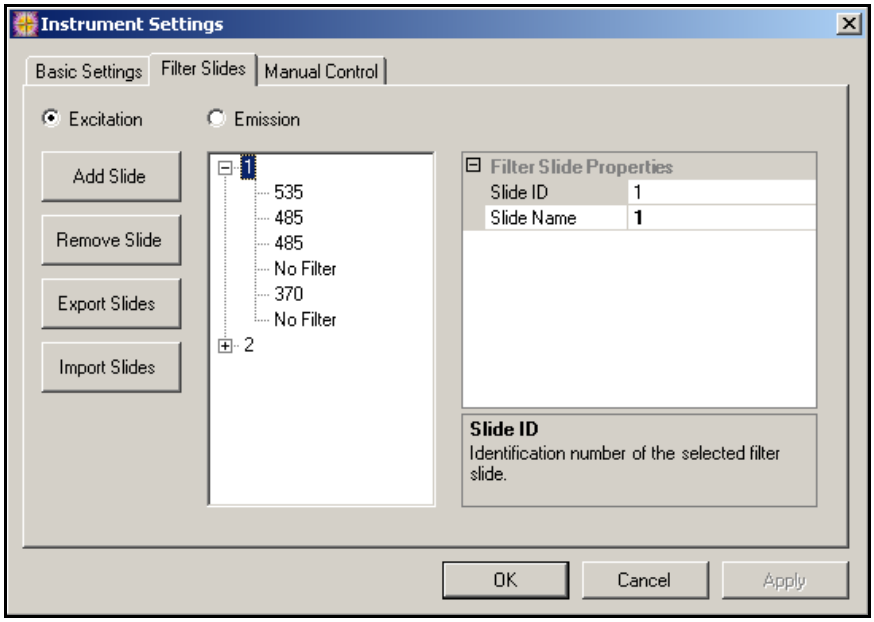


Figure 2-3. Instrument Settings — Filter Slides

2.3.2.1 Adding Filter Slides

When a new filter slide is used with the instrument for the first time, it must be added so Multimode Detection Software can identify the slide and filter configuration.

To add filter slides:

1. Select the type of filter slide to add: **Excitation** or **Emission**. The list of filter slides displays all slides of the selected type currently stored in memory.
2. Choose **Add Slide**. A new filter slide is added to the list of filter slides.
3. Configure the slide following the steps in Section 2.3.2.2, *Configuring Filter Slides*.

2.3.2.2 Configuring Filter Slides

The filter slide configuration includes a name and ID for the slide and information about the filters installed on the slide. When a new slide is added, or the filter configuration on a slide changes, the slide must be configured.

To configure a filter slide:

1. Select the type of filter slide to configure: **Excitation** or **Emission**. The list of filter slides displays all configured slides of the selected type.
2. Select the desired filter slide to configure from the list. Filter Slide Properties displays information about the selected slide (Figure 2-3).
3. In Slide ID, enter the identification number printed on the slide.
4. In Slide Name, enter a name to identify the filter slide.
5. In the list of filter slides, click the **+** to the left of the filter slide name to display the list of filters installed on the slide.
6. Select a filter to configure. Filter Properties for the selected filter is displayed on the right side of the screen (Figure 2-4).

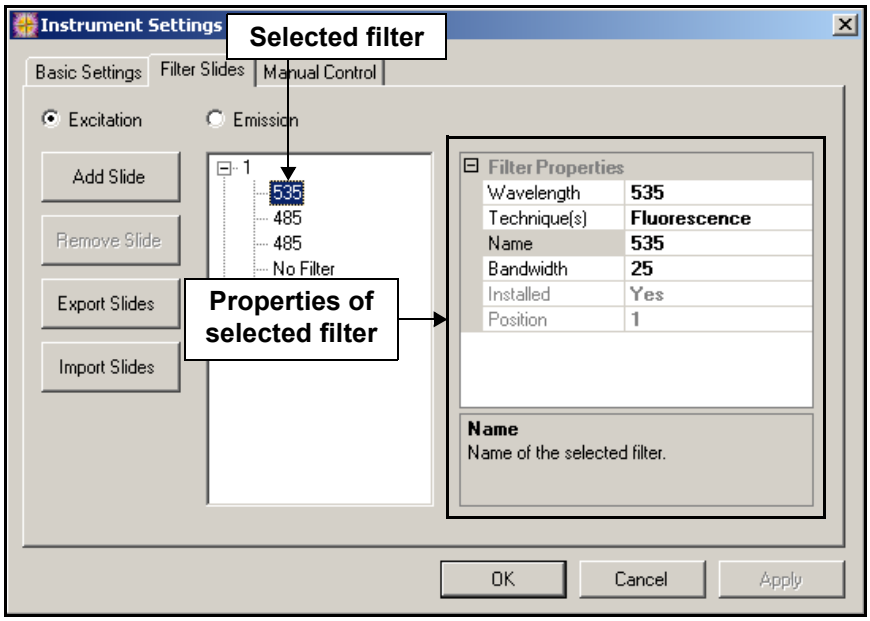


Figure 2-4. Configuring filter properties

7. In Wavelength, enter the wavelength of the filter.
8. Click in the **Technique(s)** field and then click the down arrow to display a list of the available techniques.
9. Select all techniques for which the filter applies. The filter can be used only for measurements of the selected technique type(s). When techniques are selected, the read-only Installed field displays Yes. When No is displayed, no techniques are selected and the filter may not be used in any techniques.
   - ***DTX 880*** — Select Polarization only for filter positions where a polarization filter is installed.

10. In Name, enter a name for the selected filter. Filter names default to the wavelength entered, but may be renamed as desired.
11. In Bandwidth, enter the bandwidth in nanometers of the selected filter.
12. Repeat steps 6 to 11 to configure additional filters on the slide.

2.3.2.3 Removing Filter Slides

If a filter slide is no longer used with an instrument, it can be removed from Multimode Detection Software.

To remove a filter slide:

1. Select the type of filter slide to remove: **Excitation** or **Emission**. The list of filter slides displays all slides of the selected type.
2. Select the desired filter slide to remove from the list. Filter Slide Properties displays information about the selected slide.
3. Choose **Remove Slide**. The selected filter slide is removed from the list.

2.3.2.4 Exporting and Importing Filter Slides

Information for all excitation and emission filter slides configured for the instrument can be exported to an XML file and imported to restore that configuration or share the filter slide configuration with another instrument. Importing the filter slide configuration from an XML file replaces the current configuration for *all* filter slides with the configuration from the file.

To export all filter slides:

1. Choose **Export Slides**. Save As appears.
2. In Save As, select the desired directory and enter a file name.
3. Choose **Save**. Slide information is saved as an .xml file with the specified path and file name.

To import all filter slides from a previously exported file:

1. Choose **Import Slides**. Open appears.
2. In Open, browse to and select the desired .xml file to import.
3. Choose **Open**. The filter slides defined in the .xml file are imported into the filter slide list and replace all existing filter slides.

2.3.3 Manually Controlling the Instrument

The Manual Control tab (Figure 2-5) provides options to control functions of the connected instrument, such as shaking microplates and ejecting or loading filter slides.

Manual Control is divided into five subsections:

- *Temperature Control (880 only)* (Section 2.3.3.1).
- *Shake Control* (Section 2.3.3.2).
- *Plate Control* (Section 2.3.3.3).
- *Excitation and Emission Filter Slide Control* (Section 2.3.3.4).

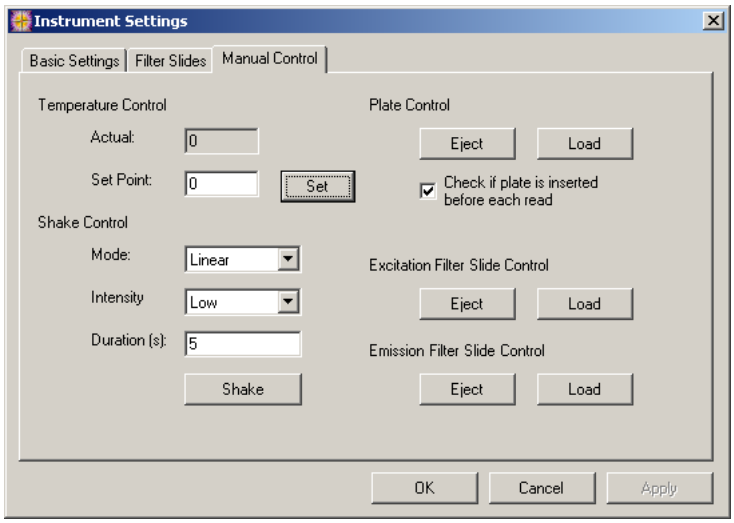


Figure 2-5. Instrument Settings — Manual Control

2.3.3.1 Temperature Control (880 only)

Temperature Control is used to set the microplate chamber temperature. The temperature is set by heating the microplate chamber; cooling the chamber is not supported. Depending on the measurement wavelength and light source used in the protocol, the temperature may range from 3°C (5.4°F) or 4°C (7.2°F) above ambient to 45°C (113°F). Actual displays the current temperature inside the instrument.

To set the temperature:

1. In Set Point, enter the desired temperature in Celsius.
2. Choose **Set**. Temperature control is activated for the instrument and begins to heat to the desired temperature. The set temperature is maintained until it is changed or the instrument is powered off.

Note: It takes a minimum of 30 minutes for the instrument to reach the desired temperature. The actual time required depends on the relative change in temperature.

To turn off temperature control:

In Set Point, enter **0**.

OR

Turn power to the instrument off and on.

2.3.3.2 Shake Control

Shake Control is used to manually shake a 96- or 384-well microplate in the plate carrier.

To manually perform a shaking operation:

CAUTION: Shake low density plates, such as 6- or 48-well plates, at low speed only. Shaking low density plates at higher speeds may cause liquid in wells to spill.

1. In Mode, select the desired shaking mode:
    - **Linear** — shakes from side to side.
    - **Orbital** — shakes in a circular pattern.
    - **Squared** — shakes in a square pattern, moving at right angles.
2. In Intensity, select the desired shaking intensity: **Low**, **Medium**, or **High**.
3. In Duration, enter the length of time to shake in seconds.
4. Choose **Shake**. The instrument shakes the microplate according to the configured settings.

2.3.3.3 Plate Control

Plate Control provides options to eject or load the plate carrier. It also features an option to sense that a microplate is in the plate carrier before starting a measurement.

To manually control the plate carrier:

- Choose **Eject** to extend the plate carrier outside the instrument.
- Choose **Load** to retract the plate carrier inside the instrument.
- Select **Check if plate is inserted before each read** to sense if a microplate is in the plate carrier before starting each measurement.

2.3.3.4 Excitation and Emission Filter Slide Control

Excitation Filter Slide Control and Emission Filter Slide Control are used to manually eject or load the excitation or emission filter slides.

To manually eject or load the excitation or emission filter slide:

- Choose **Eject** from the desired filter slide control section to unload the filter slide from the filter compartment and partially open the compartment door.

  Note: To remove the filter slide, it is still necessary to grasp it by the tab and pull it until it is free of the geared track. Store the removed filter slide in a protected, dust-free area, preferably in the original packaging.

- Choose **Load** from the desired filter slide control section to retract the filter slide into position.

3 Creating and Editing Detection Methods

3.1 Overview

Multimode Detection Software stores measurement configuration parameters in detection methods. Stored parameters include the method technique (for example, absorbance), filter(s) used, and parameters specific to the selected method, such as integration time. Absorbance, luminescence, and fluorescence method techniques are all supported. The method techniques available to the user depend on the capabilities of the instrument being controlled.

Detection methods are created and edited using the Method Editor and stored in the Detection Method Selection List. Once a detection method has been defined, it may be used in a measurement protocol (refer to Chapter 5, *Defining, Editing, and Running Protocols*).

To view available detection methods and access the Method Editor:

From the navigation pane, choose **Detection Methods**. The Detection Method Selection List appears (Figure 3-1).

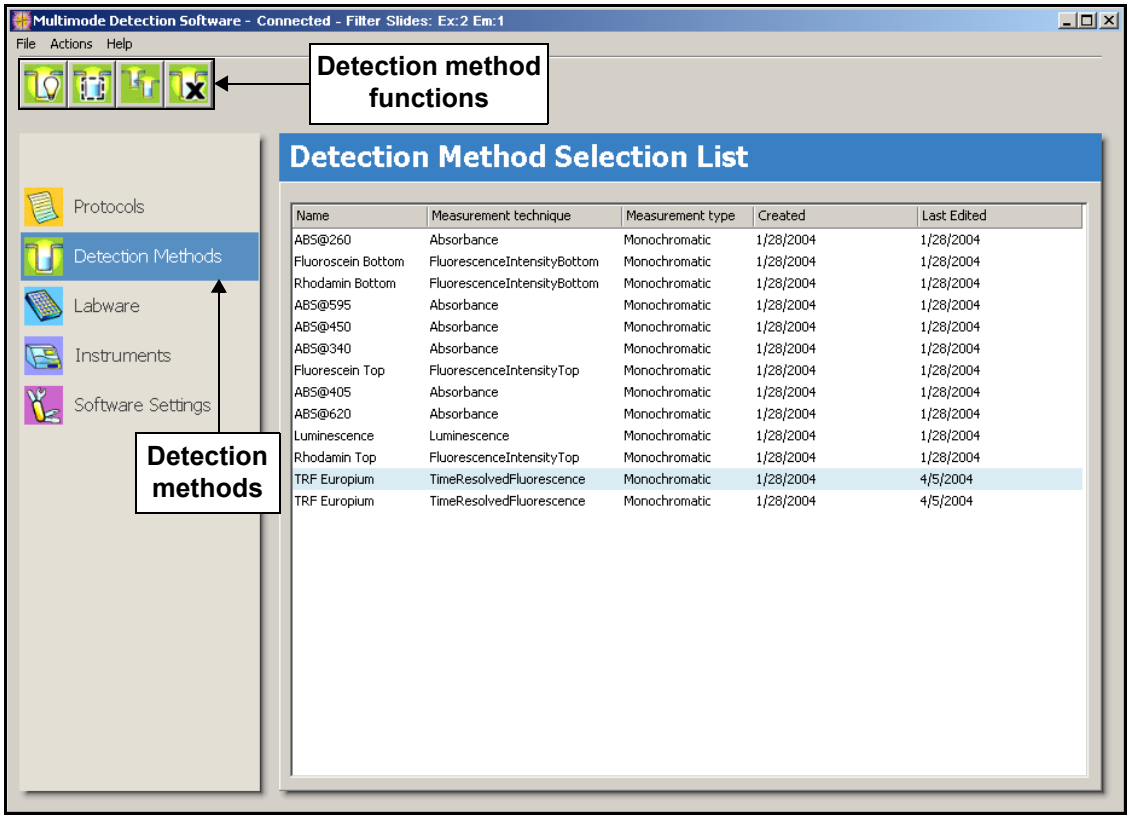


Figure 3-1. Accessing detection method functions

All detection method functions are accessed from the Detection Method Selection List. Available detection method functions are:

- *Creating Detection Methods* (Section 3.2).
- *Editing Detection Methods* (Section 3.3).
- *Copying Detection Methods* (Section 3.4).
- *Deleting Detection Methods* (Section 3.5).
- *Exporting and Importing Detection Methods* (Section 3.6).

3.2 Creating Detection Methods

Detection methods are created in the Method Editor, which guides the creation process with a wizard-type interface. Creating a new detection method requires:

- *Selecting a Method Technique* (Section 3.2.1).
- *Defining Method Parameters* (Section 3.2.3).

Note: When creating absorbance measurements, a third screen, Method Type, must also be configured (refer to Section 3.2.2, *Selecting the Type of Absorbance Method*).

To create and define a new detection method:

From the toolbar, choose **Create a new method**.

OR

From the Actions menu, choose **Create a new method**.

OR

Right-click in the Detection Method Selection List and choose **Create a new method** from the menu. The Method Editor appears (Figure 3-2).

3.2.1 Selecting a Method Technique

In the Method Editor, the type of detection method to create is selected in Method Technique (Figure 3-2). Only techniques supported by the instrument currently selected in the Instrument Selection List are available for configuration (refer to Chapter 2, *Setting Up and Controlling Instruments*).

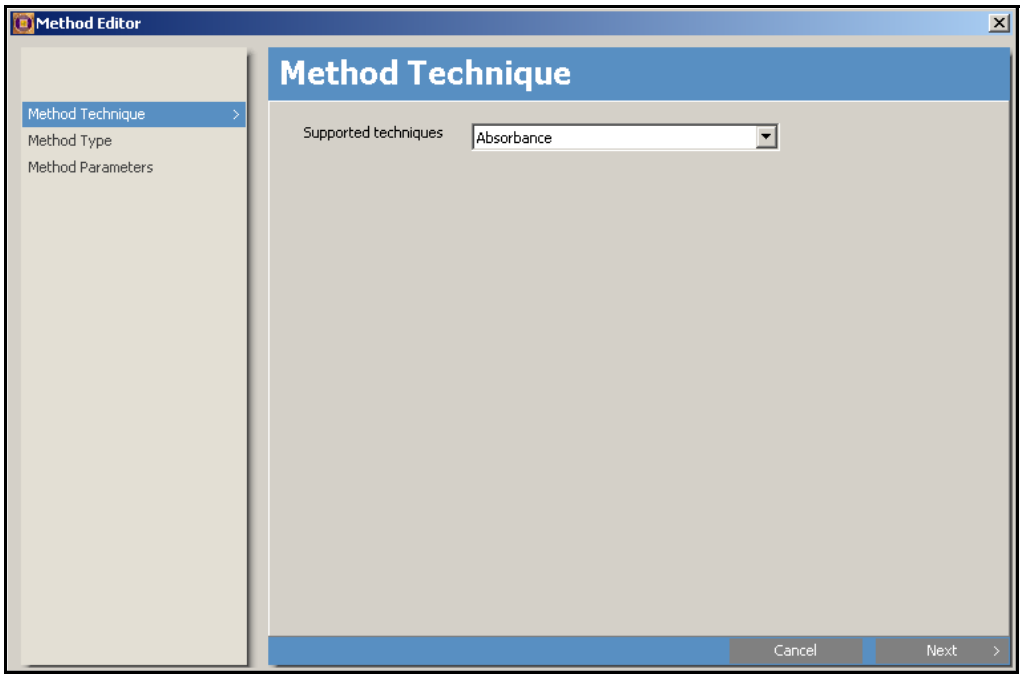


Figure 3-2. Selecting a Method Technique

To select a method technique:

1. In Supported techniques, choose the desired detection method.
2. If defining an absorbance method, choose **Next** to select the Method Type (refer to Section 3.2.2, *Selecting the Type of Absorbance Method*).

   OR

   If defining a luminescence or fluorescence method, choose **Next** to define Method Parameters (refer to Section 3.2.3, *Defining Method Parameters*).

   OR

   Choose **Cancel** to close the Method Editor without defining a new detection method.

3.2.2 Selecting the Type of Absorbance Method

When defining an absorbance detection method, use Method Type to select whether a monochromatic or bichromatic method will be defined (Figure 3-3). Monochromatic methods perform a single-wavelength measurement. Bichromatic methods perform a second measurement at a reference wavelength, which is subtracted from the first to calculate the final result.

Note: Method Type appears only when defining absorbance methods.

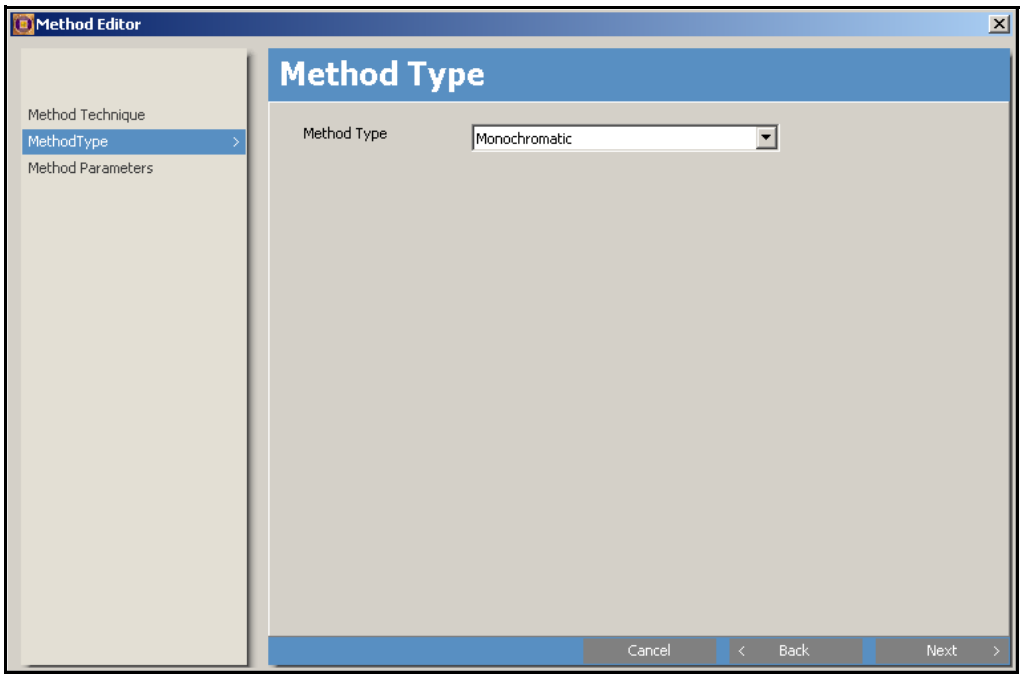


Figure 3-3. Selecting an absorbance Method Type

To select a method type:

1. In Method Type, choose the desired method: **Monochromatic** or **Bichromatic**.
2. Choose **Next** to define Method Parameters (refer to Section 3.2.3, *Defining Method Parameters*).

   OR

   Choose **Back** to select a different Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

   OR

   Choose **Cancel** to close the Method Editor without defining a new detection method.

3.2.3 Defining Method Parameters

Parameters, such as filters and integration time, are defined in Method Parameters. The parameters available for configuration depend on the technique selected in Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

Use Method Parameters for:

- *Defining Absorbance Method Parameters* (Section 3.2.3.1).
- *Defining Luminescence Method Parameters* (Section 3.2.3.2).
- *Defining Fluorescence Intensity Top Method Parameters* (Section 3.2.3.3).
- *Defining Fluorescence Intensity Bottom Method Parameters (880 only)* (Section 3.2.3.4).
- *Defining Fluorescence Polarization Method Parameters (880 only)* (Section 3.2.3.5).
- *Defining Time-Resolved Fluorescence Method Parameters (880 only)* (Section 3.2.3.6).

3.2.3.1 Defining Absorbance Method Parameters

A monochromatic absorbance method performs an absorbance measurement at a single wavelength. A bichromatic method performs a second measurement at a reference wavelength. This measurement is subtracted from the first to calculate the final result.

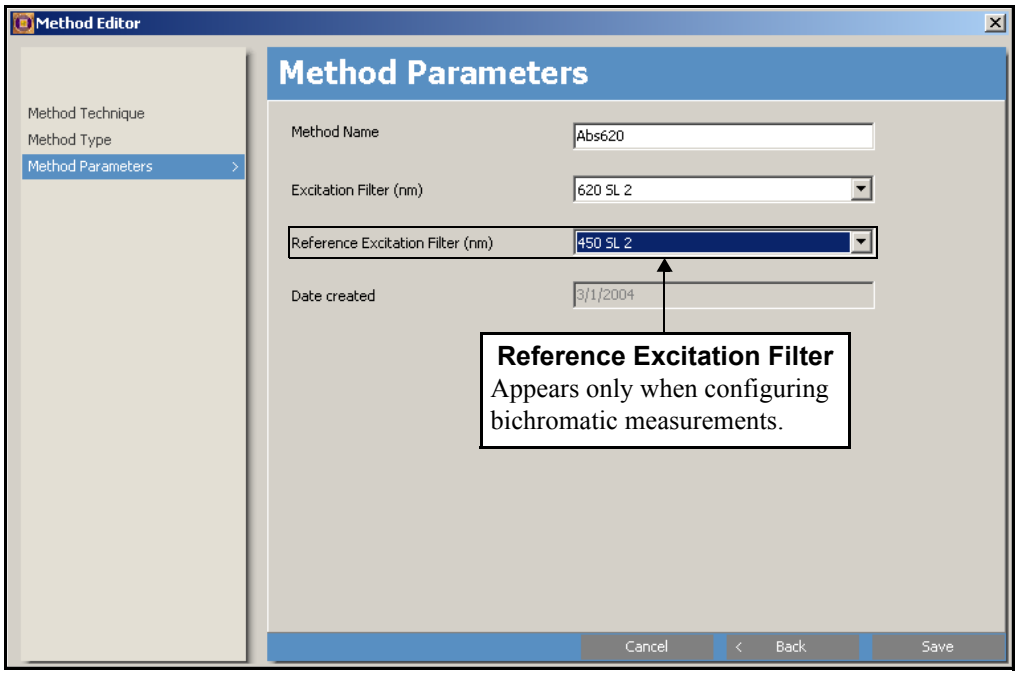


Figure 3-4. Defining absorbance measurement parameters

To define absorbance method parameters:

1. Enter a **Method Name** (Figure 3-4).
2. In Excitation Filter (nm), select the measurement filter.
3. If a bichromatic measurement is being defined, in Reference Excitation Filter (nm), select the reference filter.

   Note: The filters available are those installed on the excitation filter slide loaded in the instrument and configured for absorbance techniques in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

4. Choose **Save** to save the new absorbance detection method. The new method appears in the Detection Method Selection List.

   OR

   Choose **Back** to define a different Method Type (refer to Section 3.2.2, *Selecting the Type of Absorbance Method*).

   OR

   Choose **Cancel** to close the Method Editor without saving the new detection method.

3.2.3.2 Defining Luminescence Method Parameters

A luminescence method performs glow luminescence measurements on samples. Generally, luminescence measurements do not require a filter; however, cutoff filtration using an emission filter may be specified when eliminating photoluminescence generated by the microplate itself is desired.

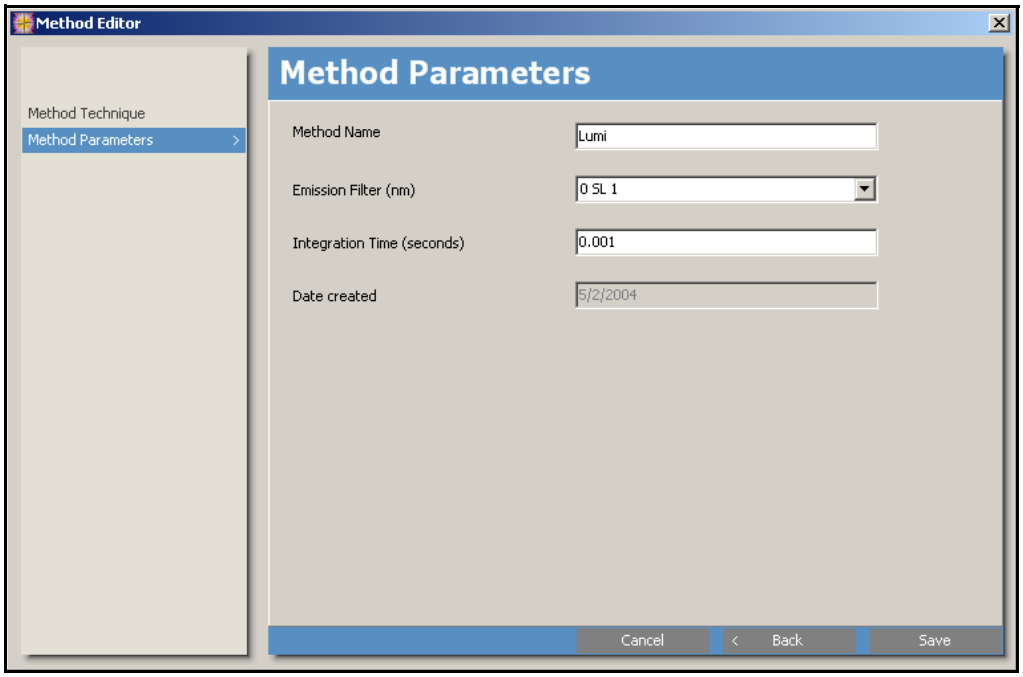


Figure 3-5. Defining luminescence measurement parameters

To define luminescence method parameters:

1. Enter a **Method Name** (Figure 3-5).
2. In Emission Filter (nm), select the cutoff filter.

   Note: Most luminescence measurements do not require cutoff filtration.

   The filters available are those installed on the emission filter slide loaded in the instrument and configured for luminescence techniques in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

3. In Integration Time (seconds), enter the length of time, in seconds, the signal is collected from samples. The integration time may be set within the range of 0.00005 to 3600 seconds.
4. Choose **Save** to save the new luminescence detection method. The new method appears in Detection Method Selection List.

   OR

   Choose **Back** to select a different Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

   OR

   Choose **Cancel** to close the Method Editor without saving the new detection method.

3.2.3.3 Defining Fluorescence Intensity Top Method Parameters

In a fluorescence intensity top method, a light source above the plate is directed through an excitation filter, which passes only the wavelength necessary to excite samples. The resulting fluorescence passes through an emission filter that separates background light from the specific wavelength generated by samples. This signal is read from above the plate by the photo multiplier tube.

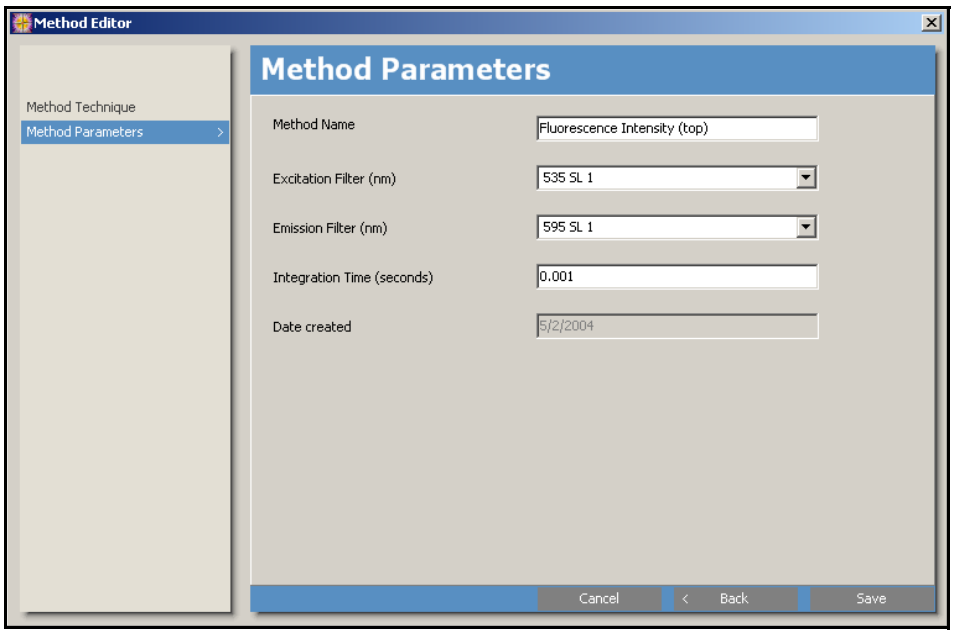


Figure 3-6. Defining fluorescence intensity top method parameters

To define fluorescence intensity top method parameters:

1. Enter a **Method Name** (Figure 3-6).
2. Select the **Excitation Filter (nm)**.
3. Select the **Emission Filter (nm)**.

   Note: The filters available are those installed on the slides loaded in the instrument and configured for fluorescence techniques in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

4. In Integration Time (seconds), enter the length of time, in seconds, the signal is collected from samples. The integration time may be set within the range of 0.00005 to 3600 seconds.
5. Choose **Save** to save the new fluorescence intensity top detection method. The new method appears in the Detection Method Selection List.

   OR

   Choose **Back** to select a different Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

   OR

   Choose **Cancel** to close the Method Editor without saving the new detection method.

3.2.3.4 Defining Fluorescence Intensity Bottom Method Parameters (880 only)

In a fluorescence intensity bottom method, a light source below the plate is directed through an excitation filter, which passes only the wavelength necessary to excite samples. The resulting fluorescence passes through an emission filter that separates background light from the specific wavelength generated by samples. This signal is read from below the plate by the photo multiplier tube.

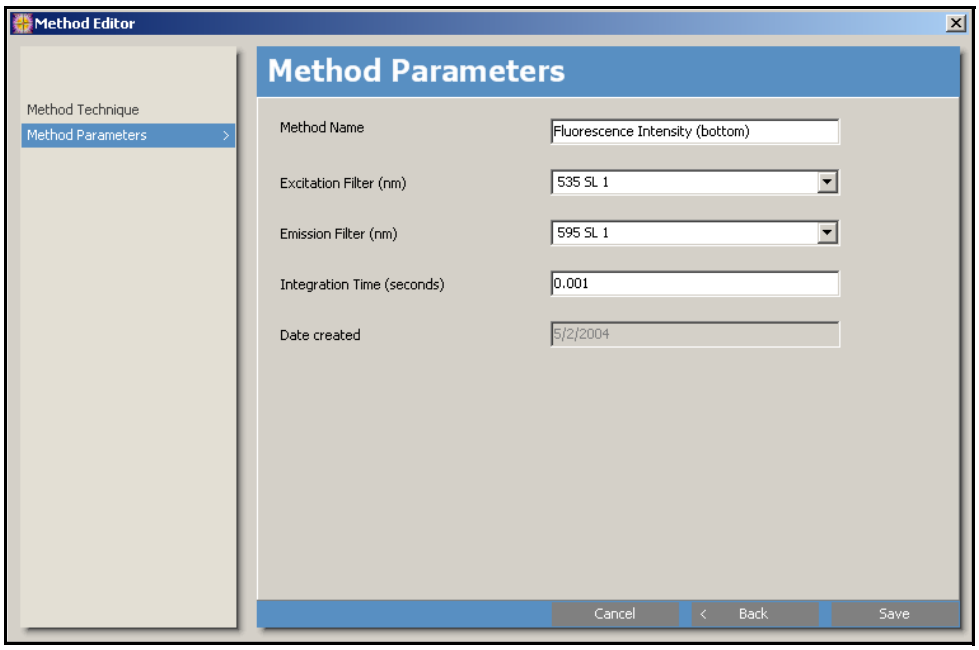


Figure 3-7. Defining fluorescence intensity bottom method parameters

To define fluorescence intensity bottom method parameters:

1. Enter a **Method Name** (Figure 3-7).
2. Select the **Excitation Filter (nm)**.
3. Select the **Emission Filter (nm)**.

   Note: The filters available are those installed on the slides loaded in the instrument and configured for fluorescence techniques in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

4. In Integration Time (seconds), enter the length of time, in seconds, the signal is collected from samples. The integration time may be set within the range of 0.00005 to 3600 seconds.
5. Choose **Save** to save the new fluorescence intensity bottom detection method. The new method appears in the Detection Method Selection List.

   OR

   Choose **Back** to select a different Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

   OR

   Choose **Cancel** to close the Method Editor without saving the new detection method.

3.2.3.5 Defining Fluorescence Polarization Method Parameters (880 only)

A fluorescence polarization method performs a fluorescence intensity measurement where two orthogonal (perpendicular) polarization states are measured.

A light source above the plate passes through an excitation filter, which passes only the wavelength necessary for excitation, and a polarizing filter. The fluorescence resulting from the excitation of the sample is separated by polarization and passed through two emission filters equipped with polarizing filters to ensure both polarization states are measured.

The polarized signal is then read from above the plate by the photo multiplier tube.

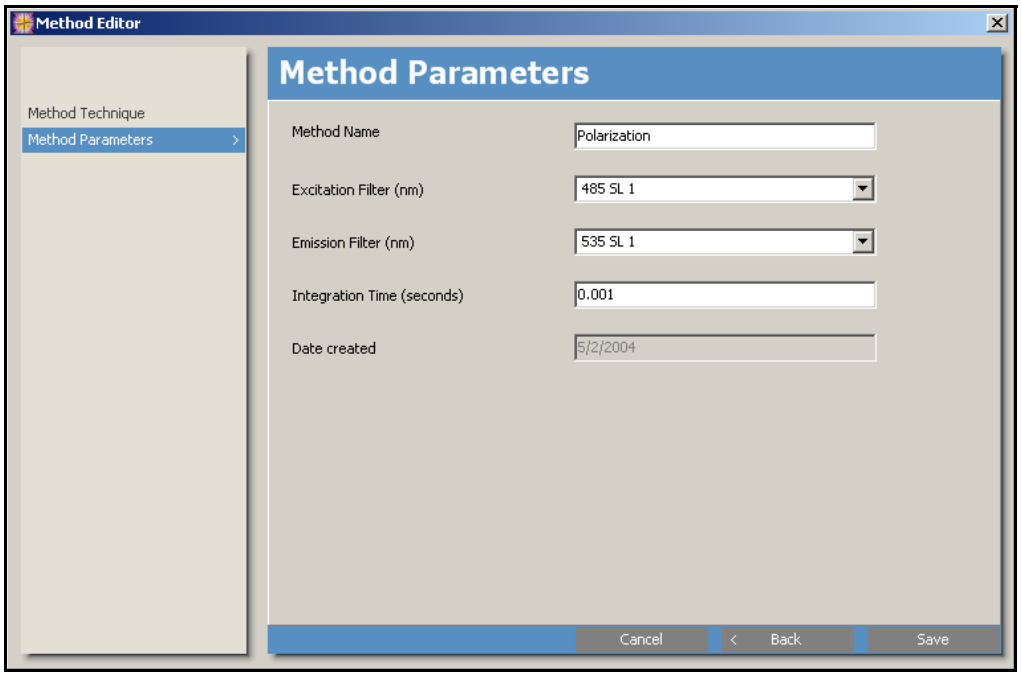


Figure 3-8. Defining fluorescence polarization method parameters

To define fluorescence polarization parameters:

1. Enter a **Method Name** (Figure 3-8).
2. Select the **Excitation Filter (nm)**.
3. Select the **Emission Filter (nm)**.

   Note: The filters available are those installed on the excitation and emission filter slides loaded in the instrument and configured for fluorescence polarization techniques in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

4. In Integration Time (seconds), enter the length of time, in seconds, the signal is collected from samples. The integration time may be set within the range of 0.00005 to 3600 seconds.

5. Choose **Save** to save the new fluorescence polarization detection method. The new method appears in the Detection Method Selection List.

   OR

   Choose **Back** to select a different Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

   OR

   Choose **Cancel** to close the Method Editor without saving the new detection method.

3.2.3.6 Defining Time-Resolved Fluorescence Method Parameters (880 only)

In a time-resolved fluorescence measurement, the excitation light source is turned off and the measurement is performed after a specified delay. Several of these excitation/measurement cycles may be performed on each sample. When multiple excitation/measurement cycles are performed, the results from all cycles are used to calculate a single measurement result for each sample.

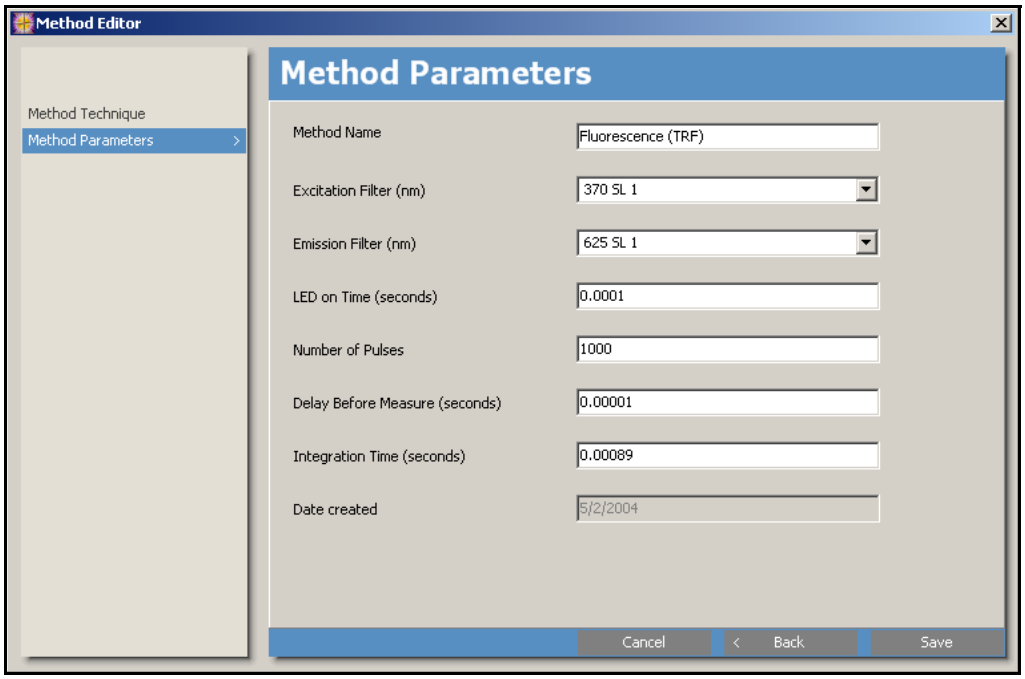


Figure 3-9. Defining time-resolved fluorescence method parameters

To define time-resolved fluorescence parameters:

1. Enter a **Method Name** (Figure 3-9).
2. Select the **Excitation Filter (nm)**.
3. Select the **Emission Filter (nm)**.

   Note: The filters available are those installed on the excitation and emission filter slides loaded in the instrument and configured for time-resolved fluorescence techniques in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

4. In LED on Time (seconds), enter the length of time, in seconds, that the LED light source remains turned on.
5. In Number of Pulses, enter the number of excitation/measurement cycles performed in the measurement.
6. In Delay Before Measure (seconds), enter the interval, in seconds, between switching off the light source and performing the measurement. The delay may be set within the range of 0.000001 to 0.0075 of a second.
7. In Integration Time (seconds), enter the length of time, in seconds, each sample is measured. The integration time may be set within the range of 0.00005 to 0.0075 of a second.
8. Choose **Save** to save the new time-resolved fluorescence detection method. The new method appears in the Detection Method Selection List.

   OR

   Choose **Back** to select a different Method Technique (refer to Section 3.2.1, *Selecting a Method Technique*).

   OR

   Choose **Cancel** to close the Method Editor without saving the new detection method.

3.3 Editing Detection Methods

Parameters configured in saved detection methods may be edited; however, the method technique may not be changed. Default detection methods provided with the software and methods used in protocols may not be renamed.

To edit a detection method:

1. In the Detection Method Selection List, select the detection method to edit.
2. From the toolbar, choose **Edit the selected method**.

OR

From the Actions menu, choose **Edit the selected method**.

OR

Right-click on the selected detection method and choose **Edit the selected method** from the menu. The Method Editor appears (Figure 3-10).

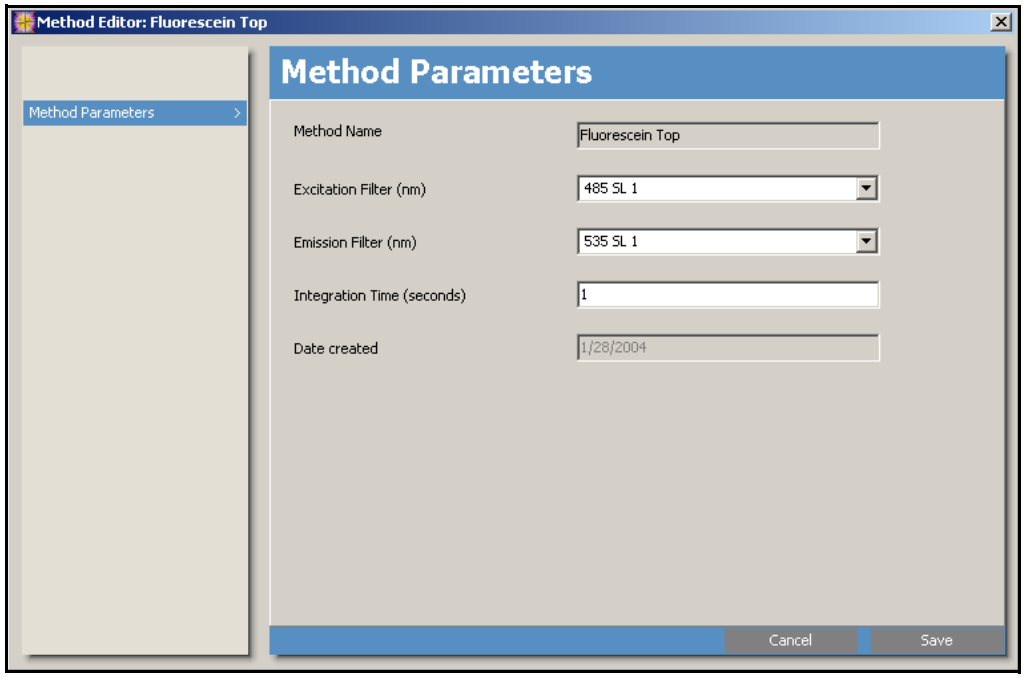


Figure 3-10. Editing a fluorescence intensity top method

3. Edit the method parameters as desired. For more information about the parameters for a specific detection method, refer to the section that covers defining the desired detection method:
    - *Defining Absorbance Method Parameters* (Section 3.2.3.1).
    - *Defining Luminescence Method Parameters* (Section 3.2.3.2).
    - *Defining Fluorescence Intensity Top Method Parameters* (Section 3.2.3.3).
    - *Defining Fluorescence Intensity Bottom Method Parameters (880 only)* (Section 3.2.3.4).
    - *Defining Fluorescence Polarization Method Parameters (880 only)* (Section 3.2.3.5).
    - *Defining Time-Resolved Fluorescence Method Parameters (880 only)* (Section 3.2.3.6).
4. Choose **Save** to close the Method Editor and save the changes.

    OR

    Choose **Cancel** to close the Method Editor without saving changes.

3.4 Copying Detection Methods

Copies may be made of existing detection methods. After a copy has been created, it may be used as a template for a new detection method using the same method technique.

To make a copy of a detection method:

1. In the Detection Method Selection List, select the detection method to copy.
2. From the toolbar, choose **Make a copy of the selected method**.

OR

From the Actions menu, choose **Make a copy of the selected method**.

OR

Right-click on the selected detection method and choose **Make a copy of the selected method** from the menu that appears.

Note: The default name format for copied detection methods is Copy of OriginalName. To change the name, edit the detection method (refer to Section 3.3, *Editing Detection Methods*).

3.5 Deleting Detection Methods

User-created detection methods not used in any measurement protocols may be deleted from the Detection Method Selection List. Default detection methods provided with the software and user-created methods used in protocols may not be deleted.

To delete a detection method:

1. In the Detection Method Selection List, select the detection method to delete.

2. From the toolbar, choose **Delete the selected method**.

   OR

   From the Actions menu, choose **Delete the selected method**.

   OR

   Right-click on the selected detection method and choose **Delete the selected method** from the menu. Message appears (Figure 3-11).

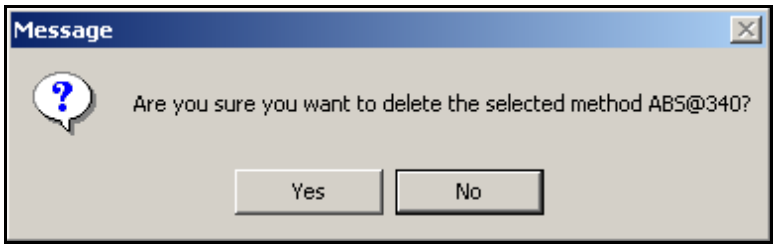


Figure 3-11. Confirming the deletion of a detection method

3. Choose **Yes** to delete the selected detection method.

   OR

   Choose **No** to cancel the deletion.

3.6 Exporting and Importing Detection Methods

User-defined detection methods saved in the Detection Method Selection List can be exported to an XML file and imported later to restore that configuration or share it with a copy of Multimode Detection Software installed on another computer.

Exporting default detection methods included with the software installation is not necessary. Since a default detection method may not be edited, deleted, or overwritten, it is available to users at all times. Because of this, importing a default detection method from an XML export file is not permitted by the software.

To export a detection method:

1. In the Detection Method Selection List, select the detection method to export.
2. From the File menu, choose **Export>Detection Method**. Browse for Folder appears (Figure 3-12).

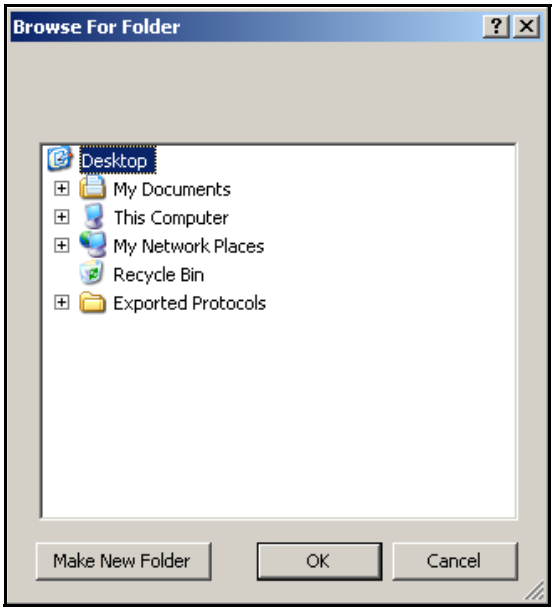


Figure 3-12. Choosing the folder where an exported detection method will be saved

3. In Browse for Folder, browse to the folder where the exported detection method will be saved.

   OR

   Choose **Make New Folder** to create a new folder where the exported detection method will be saved.

4. Choose **OK** to export the detection method. The exported detection method is saved using the default file name format, Method__*MethodName*.xml. In order to import the file at a later date, the filename must not be changed.

   OR

   Choose **Cancel** to stop the operation without exporting the detection method.

To import a detection method from an exported XML file:

1. From the File menu, choose **Import>Detection Method**. Open appears.
2. In Open, browse to and select the desired XML file to import.
3. Choose **Open**. The detection method is imported to the Detection Method Selection List.

4 Creating and Editing Labware

4.1 Overview

Multimode Detection Software supports a wide range of labware, with many common microplate formats already preconfigured and ready for use in protocols. Configured labware types are listed in the Labware Selection List and are available for use in protocols.

New labware types may be created at any time using the Labware Editor. The Labware Editor also provides the ability to edit and delete existing labware types not used in protocols, make copies of labware types, and optimize labware dimensions to compensate for slight dimensional variations that may exist between production lots.

- ***DTX 800*** — supports 96-well and 384-well microplates.
- ***DTX 880*** — supports 6- to 384-well microplates.

All labware functions are accessed from the Labware Selection List (Figure 4-1). Labware functions include:

- *Creating Labware* (Section 4.2).
- *Editing Labware* (Section 4.3).
- *Copying Labware* (Section 4.4).
- *Deleting Labware* (Section 4.5).
- *Optimizing Labware* (Section 4.6).
- *Exporting and Importing Labware* (Section 4.7).

To define and edit labware:

From the navigation pane, choose **Labware**. The Labware Selection List appears (Figure 4-1).

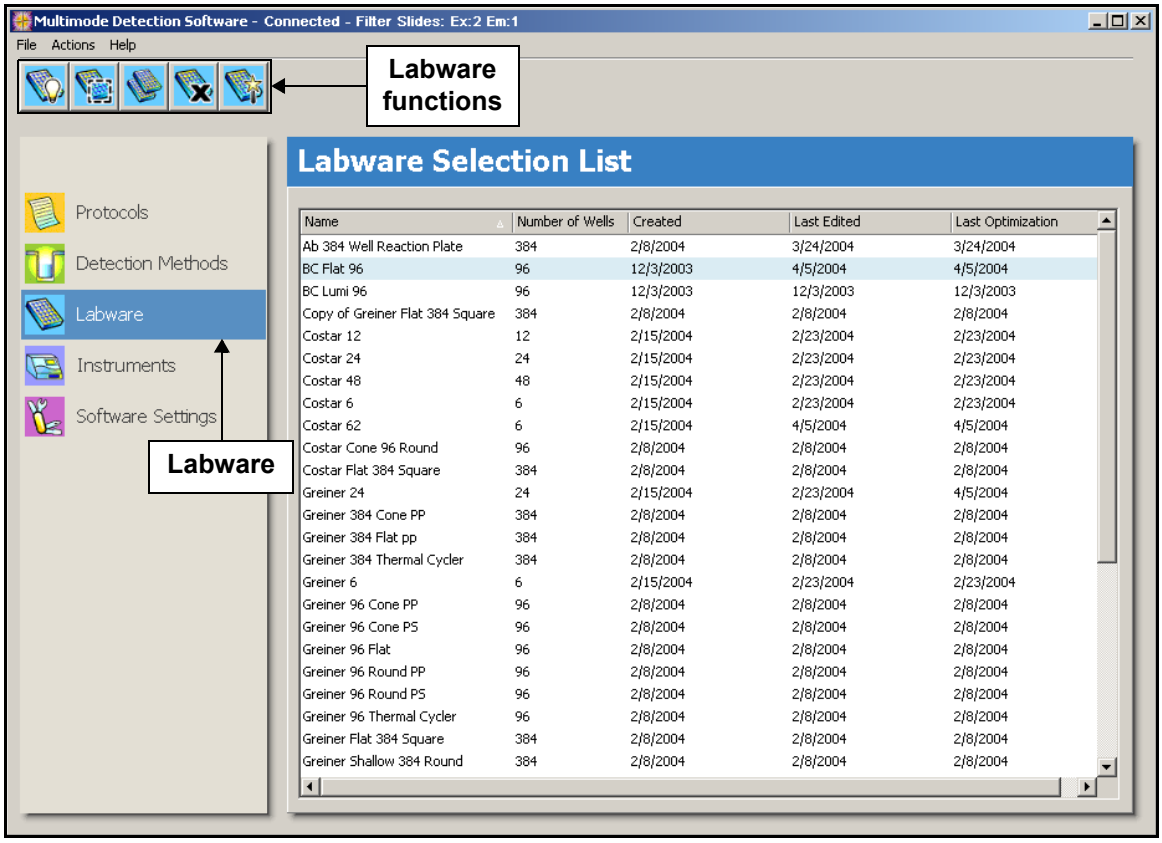


Figure 4-1. Accessing labware functions

4.2 Creating Labware

New types of labware are created in the Labware Editor, which guides the creation process with a wizard-type interface. Creating labware includes:

- *Defining Labware Information* (Section 4.2.1).
- *Configuring Offsets and Well Dimensions for the Default Labware Lot* (Section 4.2.2).

To create and define new labware:

From the toolbar, choose **Create a new piece of labware**. The Labware Editor appears (Figure 4-2).

OR

From the Actions menu, choose **Create a new piece of labware**.

OR

Right-click in the Labware Selection List and choose **Create a new piece of labware** from the menu.

4.2.1 Defining Labware Information

Use Labware Information to define labware names, dimensions, well parameters, and supported measurement techniques (Figure 4-2).

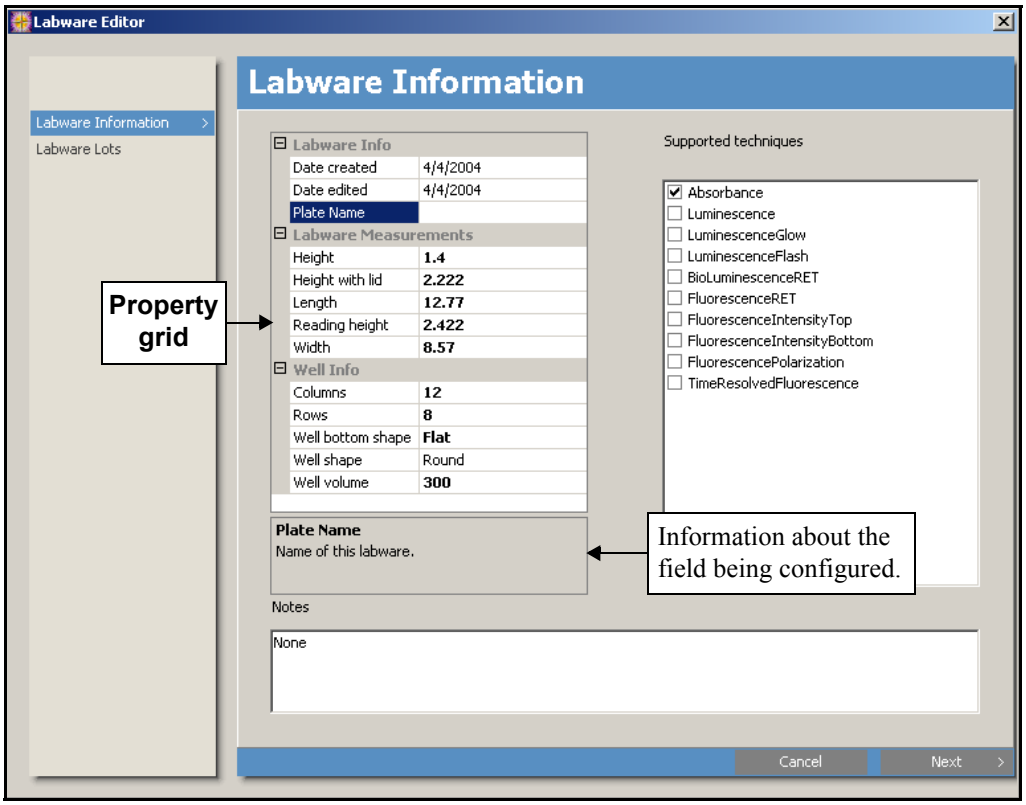


Figure 4-2. Defining plate dimensions and information

To define Labware Information:

1. If necessary, click the **+** next to Labware Info to display the fields in the category.
2. Enter the **Plate Name**. A name must be entered to proceed to the second configuration screen, Labware Lots.
3. If necessary, click the **+** next to Labware Measurements to display the fields in the category.

Note: More information about the field being defined is displayed below the property grid (Figure 4-2).

CAUTION: The plate height configured must not be less than that of the actual plate. Doing so may cause the DTX 880 optics transport to collide with the plate during a Read Height Optimization.

4. Enter the **Height** of the plate. *All* labware dimensions are entered in centimeters.
5. Enter the **Height with lid**.
6. Enter the **Length** of the plate.
7. In Reading height, enter the height from the top of the plate at which the plate is read.
8. Enter the **Width** of the plate.
9. If necessary, click the **+** next to Well Info to display the fields in the category.
10. Enter the number of **Columns** on the plate.
11. Enter the number of **Rows** on the plate.
12. Click in either column of Well bottom shape, then click on the down arrow and choose the shape of well bottoms: **Flat**, **Cone**, or **Round**.
13. Click in either column of Well shape, then click on the down arrow and choose the shape of the wells: **Round**, **Square**, or **Cone**.
14. Enter the maximum **Well volume** in microliters.
15. In Supported Techniques, select all measurement techniques compatible with the plate being defined. Labware may be used only in protocols that use a compatible measurement technique.

Note: Refer to Section 4.2.1.1, *General Labware Selection Guidelines* for more information about choosing the appropriate labware for the desired techniques.

16. In Notes, enter information about the labware or configuration, if desired.
17. Choose **Next** to define the default row and column offsets and well dimensions for the labware type in Labware Lots (refer to Section 4.2.2, *Configuring Offsets and Well Dimensions for the Default Labware Lot*).

    OR

    Choose **Cancel** to close the Labware Editor without creating or saving new labware.

4.2.1.1 General Labware Selection Guidelines

When creating labware, select only measurement techniques compatible with the microplate being defined. Each measurement technique requires labware of a specific color and/or material be used.

Table 4-1 provides general labware color and material guidelines for each measurement technique. Along with these basic guidelines, always select microplates with a surface treatment suitable for the desired application, and follow any additional guidelines provided by the plate manufacturer.

Table 4-1. General Microplate selection guidelines

Measurement Technique	Supported Plate Color	Additional Considerations
Absorbance	clear, white with clear bottom, or black with clear bottom	Clear polystyrene or film plates with transparent bottoms are suitable. Polypropylene or PVC plates do not provide sufficient optical quality.
Luminescence Glow Type	solid black or solid white	Black plates are recommended unless the signal is weak enough to require the higher sensitivity of white plates. However, with strong signals, white plates may produce crosstalk.
Fluorescence Intensity Top	solid black	N/A
Fluorescence Intensity Bottom	black with clear bottom	N/A
Fluorescence Polarization	solid black	N/A
Time-Resolved Fluorescence	solid white	N/A

4.2.2 Configuring Offsets and Well Dimensions for the Default Labware Lot

Use Labware Lots to define row and column offsets and well dimensions (Figure 4-3). The offsets and dimensions entered when new labware is created define the default labware lot (DefaultLot). After the new labware has been saved, additional lots may be created by optimizing the labware to compensate for dimensional variations between different production lots (refer to Section 4.6, *Optimizing Labware*).

In Labware Lots, x and y offsets are defined for all four corners of the labware. An x offset is the distance from the edge of the microplate to the first row; a y offset is the distance from the edge of plate to the first column. Well dimensions defined include well depth, length, and width, as well as distances between rows and columns.

To configure offsets and well dimensions:

1. If necessary, click the **+** next to Labware Lot Measurements to display the fields in the category. More information about the field being defined is displayed below the property grid (Figure 4-3).

   Note: The fields in Labware Lot Info may not be configured when creating new labware.

2. Enter column and row x and y offsets for each of the four corner wells. *All* offsets and well dimensions are entered in centimeters.

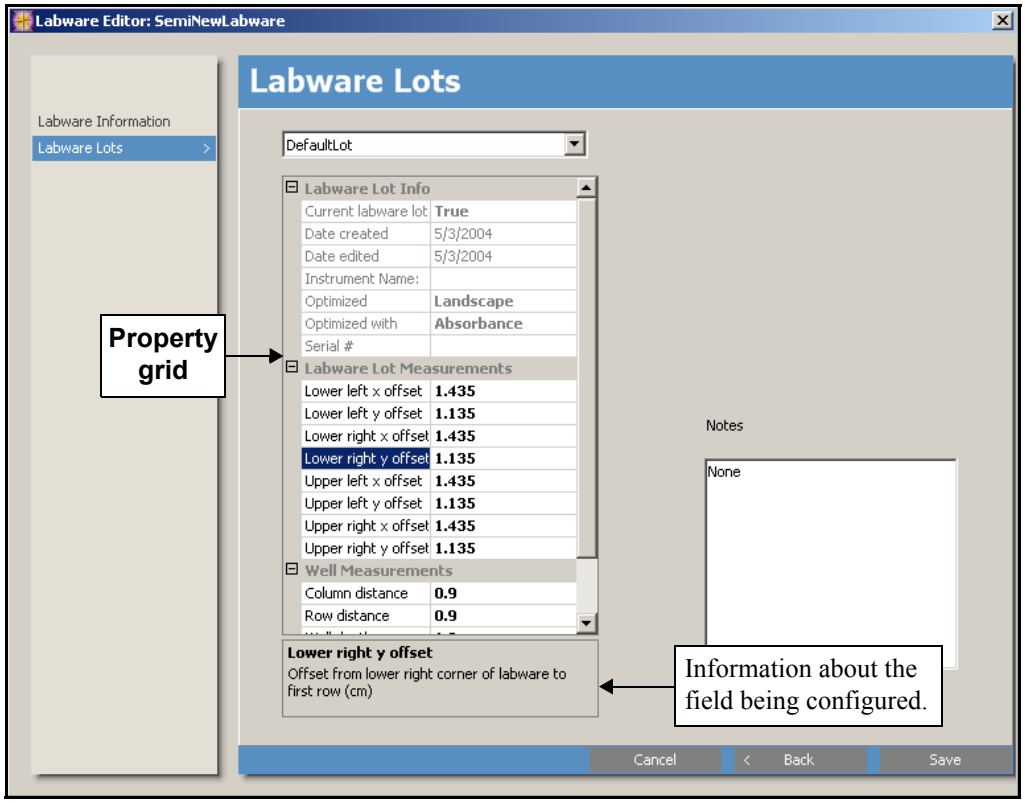


Figure 4-3. Defining offsets and well dimensions

3. If necessary, click the **+** next to Well Measurements to display the fields in the category.
4. In Column Distance, enter the distance between columns.
5. In Row Distance, enter the distance between rows.
6. Enter the **Well depth**.
7. In Well Length, enter the length of the well in the direction of the columns on the plate.
8. In Well Width, enter the length of the well in the direction of the rows on the plate.
9. In Notes, enter information about the labware lot or configuration, if desired.
10. Choose **Save** to save the new labware and close the Labware Editor.

    OR

    Choose **Back** to edit Plate Information (refer to Section 4.2.1, *Defining Labware Information*).

    OR

    Choose **Cancel** to close the Labware Editor without creating new labware.

4.3 Editing Labware

Dimensions and information for user-defined labware not used in measurement protocols may be edited. For default labware included in the software installation or labware used in protocols, dimensions and information may be viewed, but not edited.

When labware lots have been created by optimizing the labware to compensate for dimensional variations between production lots, the active lot may be changed and/or edited. The active lot may be selected and/or edited for *all* labware in the Labware Selection List.

Note: Labware lots are created by optimizing the labware (refer to Section 4.6, *Optimizing Labware*).

Labware is edited in the Labware Editor (Figure 4-4) and includes:

- *Viewing and Editing Labware Information* (Section 4.3.1).
- *Selecting and Editing Labware Lots* (Section 4.3.2).

To view and edit labware dimensions and information:

1. In the Labware Selection List, select the labware to edit.
2. From the toolbar, choose **Edit the selected labware**.

   OR

   From the Actions menu, choose **Edit the selected labware**.

   OR

   Right-click on the selected labware and choose **Edit the selected labware** from the menu that appears.

4.3.1 Viewing and Editing Labware Information

Use Labware Information to view and edit labware dimensions, information, and supported techniques. Plate information for default labware included in the software installation and labware used in protocols may be viewed, but not edited.

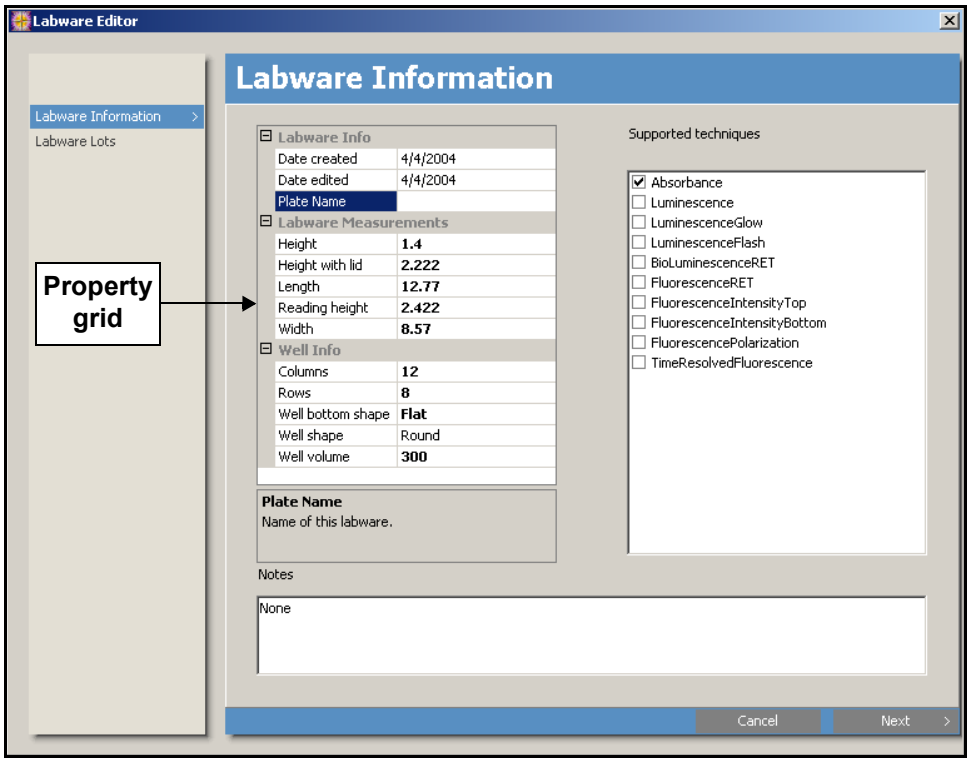


Figure 4-4. Editing labware dimensions and well information

To edit labware information:

1. In the property grid, edit labware dimensions and information as desired. Refer to Section 4.2.1, *Defining Labware Information*, for more information about the fields available in the property grid.
2. In Supported Techniques, change the measurement techniques supported by the labware, if desired.

   Note: Refer to Section 4.2.1.1, *General Labware Selection Guidelines* for more information about labware/technique compatibility.

3. Edit the labware **Notes**, if desired.
4. Choose **Next** to select and edit labware lots and save changes made to the labware (refer to Section 4.3.2, *Selecting and Editing Labware Lots*).

   OR

   Choose **Cancel** to close the Labware Editor without saving changes.

4.3.2 Selecting and Editing Labware Lots

Use Labware Lots to select the active lot and/or edit and save changes made in the Labware Editor (Figure 4-5). Lots can be selected and edited for all labware, including labware used in measurement protocols.

Note: Labware Lots are created by optimizing labware (refer to Section 4.6, *Optimizing Labware*).

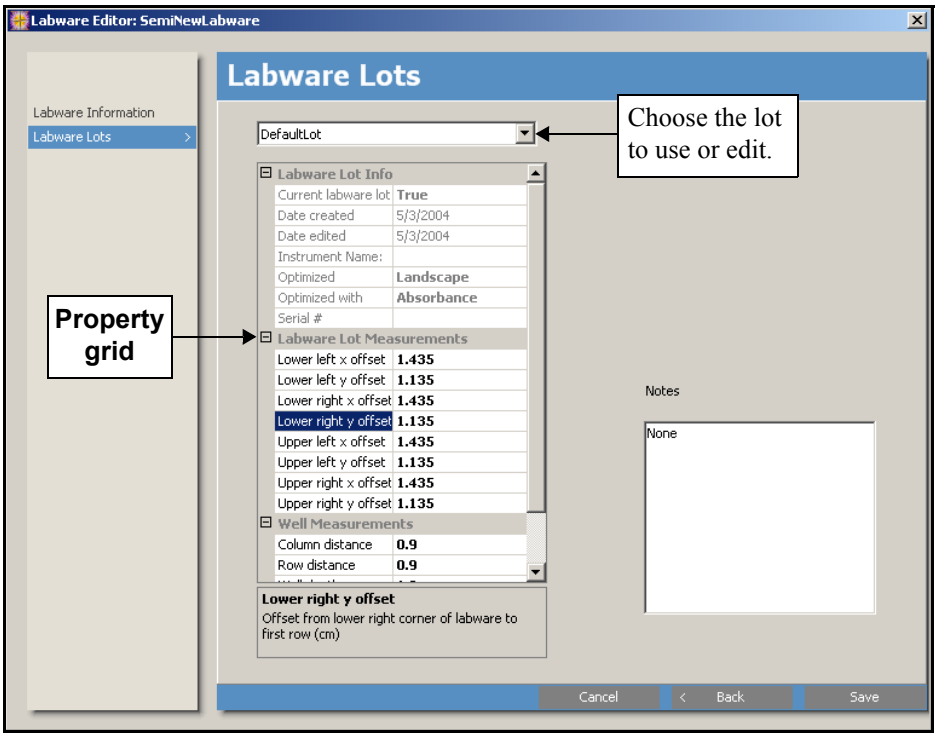


Figure 4-5. Configuring offsets and well dimensions in Labware Lots

To select and edit lots:

1. From the pull-down menu, choose the lot to use or edit. The default lot created when the labware was defined and all lots configured using Optimizing Labware are available (refer to Section 4.6, *Optimizing Labware*).
2. Click in Current labware lot and choose **True**, if necessary. True must be selected in order to save changes made to the labware lot.
3. In the property grid, edit lot dimensions and information as desired. Refer to Section 4.2.2, *Configuring Offsets and Well Dimensions for the Default Labware Lot*, for more information about the fields available in the property grid.
4. Edit the lot **Notes**, if desired.
5. Choose **Save** to save changes made in the Labware Editor.

   OR

   Choose **Back** to view or edit Plate Information (refer to Section 4.3.1, *Viewing and Editing Labware Information*).

   OR

   Choose **Cancel** to close the Labware Editor without saving changes.

4.4 Copying Labware

Labware can be copied and then used as a template for a new labware type by editing the dimensions and parameters in the Labware Editor (refer to Section 4.3, *Editing Labware*).

To make a copy of a labware type:

1. In the Labware Selection List, select the labware type to copy.

2. From the toolbar, choose **Make a copy of the selected labware**.

   OR

   From the Actions menu, choose **Make a copy of the selected labware**.

   OR

   Right-click on the selected labware type and choose **Make a copy of the selected labware** from the menu that appears.

Note: The default name format for copied labware types is Copy of OriginalName. To change the name, edit the labware type (refer to Section 4.3, *Editing Labware*).

4.5 Deleting Labware

User-defined labware not used in any measurement protocols may be deleted from the Labware Selection List. Default labware included with the software installation and labware used in protocols may not be deleted.

To delete labware:

1. In the Labware Selection List, select the labware to delete.

2. From the toolbar, choose **Delete the selected labware**.

   OR

   From the Actions menu, choose **Delete the selected labware**.

   OR

   Right-click on the selected labware and choose **Delete the selected labware** from the menu that appears. Message appears (Figure 4-6).

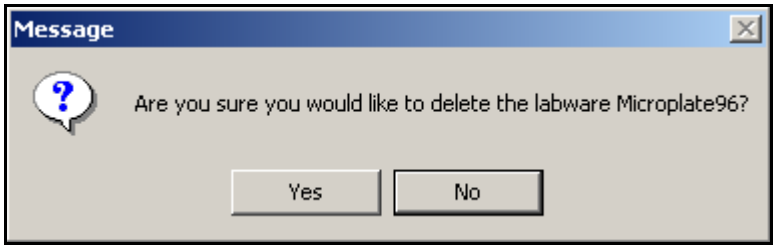


Figure 4-6. Confirming a labware deletion

3. Choose **Yes** to delete the selected labware.

   OR

   Choose **No** to cancel the deletion.

4.6 Optimizing Labware

Microplate dimensions may vary slightly between production lots, which potentially affects measurement accuracy. Multimode Detection Software provides the ability to create different labware lots for each type of labware, which increases accuracy by optimizing labware dimensions and information for each production lot used.

Labware is optimized in Optimizing Labware (Figure 4-7), which guides the process with a wizard type interface. Optimizing labware includes:

- *Selecting the Detection Method* (Section 4.6.1).
- *Preparing and Loading the Labware* (Section 4.6.2).
- *Performing the Optimization Read* (Section 4.6.3).
- *Selecting the Centers of the Four Corner Wells* (Section 4.6.4).
- *Verifying Well Centers* (Section 4.6.5).

To optimize labware:

1. In the Labware Selection List, select the labware to optimize.
2. From the toolbar, choose **Optimize the selected labware**.

   OR

   From the Actions menu, choose **Optimize the selected labware**.

   OR

   Right-click on the selected labware and choose **Optimize the selected labware** from the menu that appears.

4.6.1 Selecting the Detection Method

Labware is optimized by performing absorbance measurements of the four corner wells of the microplate and then defining the well centers using images of the wells generated by the measurements. To ensure the most accurate optimization is performed, use Select Detection Method to select the most appropriate detection method for the optimization (Figure 4-7).

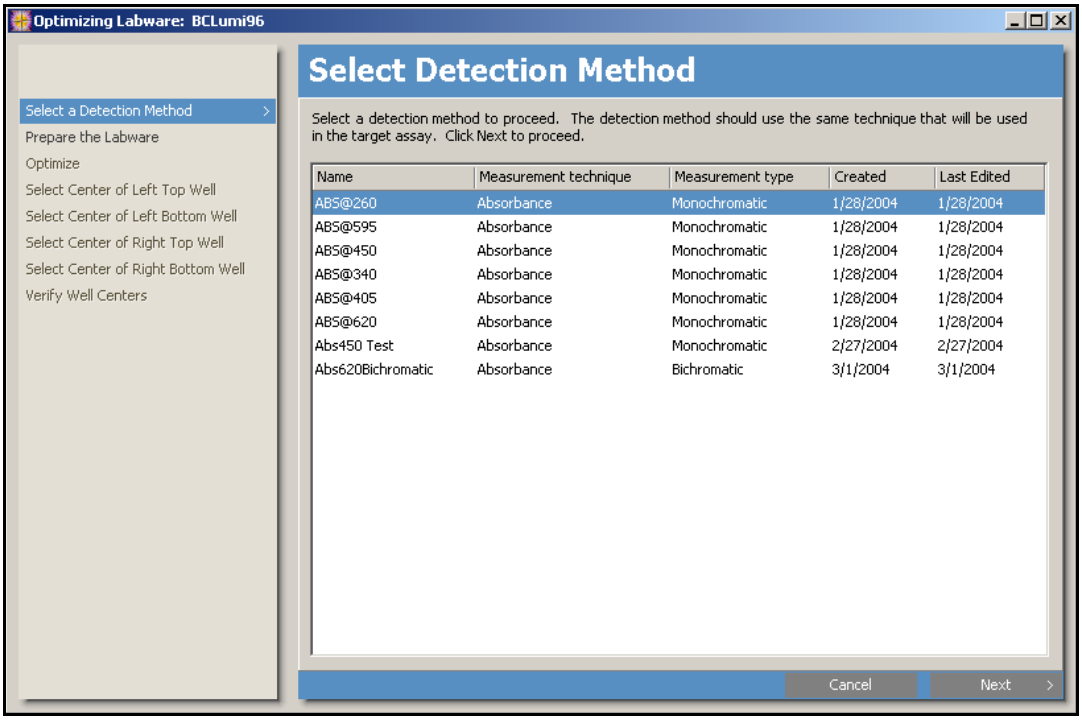


Figure 4-7. Selecting the detection method for labware optimization

To select the detection method:

1. Choose the detection method that most closely matches the method used in the desired protocol. For absorbance and fluorescence intensity top and bottom measurements, this is the exact method used. For luminescence, fluorescence polarization, and time-resolved fluorescence measurements, choose a method performed at the same or similar wavelength.
2. Choose **Next** to Prepare Labware (refer to Section 4.6.2, *Preparing and Loading the Labware*).

   OR

   Choose **Cancel** to close Optimizing Labware without performing the optimization.

4.6.2 Preparing and Loading the Labware

To optimize labware dimensions, the four corner wells of the plate are read while empty. Prepare Labware provides controls to load and eject labware from the instrument and to choose the orientation of the plate on the microplate carrier (Figure 4-8).

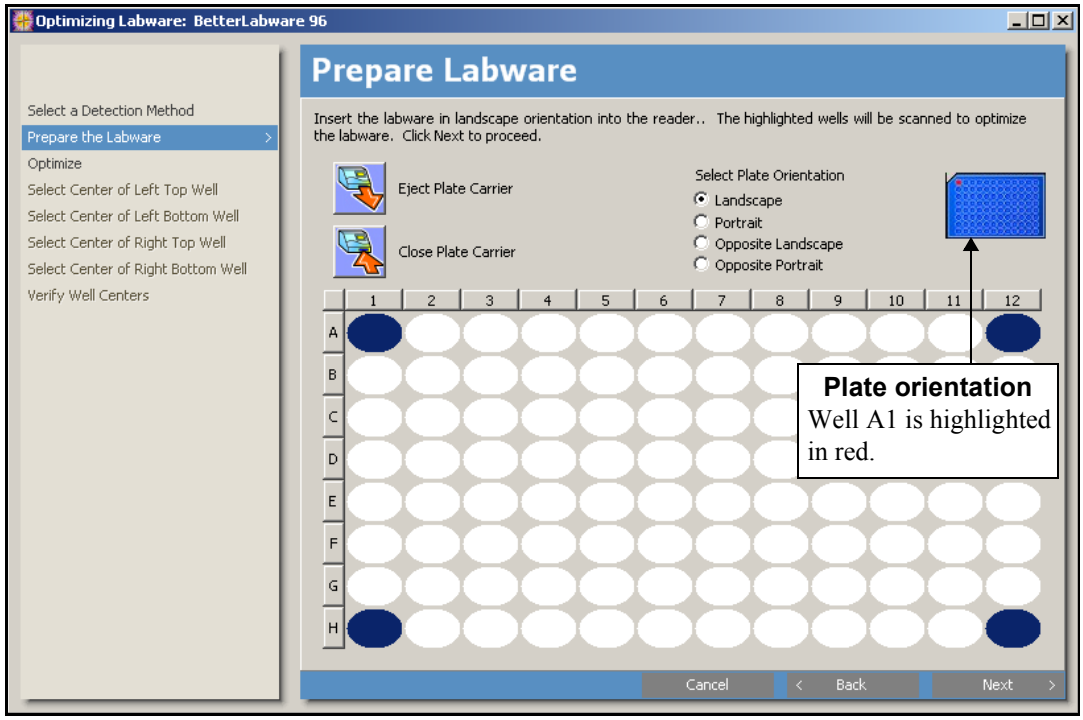


Figure 4-8. Preparing the labware for optimization

To prepare labware for optimization:

1. Choose **Eject Plate Carrier** to move the microplate carrier outside the instrument.
2. Place the microplate to be optimized on the plate carrier.

3. Choose **Close Plate Carrier** to load the microplate into the instrument.
4. In Select Plate Orientation, choose the orientation of the plate on the microplate carrier. The selected orientation is displayed graphically to the right of the screen, with well A1 highlighted in red.
5. Choose **Next** to start the optimization (refer to Section 4.6.3, *Performing the Optimization Read*). The optimization read begins automatically.

   OR

   Choose **Back** to select a different detection method (refer to Section 4.6.1, *Selecting the Detection Method*).

   OR

   Choose **Cancel** to close Optimizing Labware without performing the optimization.

4.6.3 Performing the Optimization Read

Optimization in Progress displays the status of the optimization read and provides the ability to cancel the optimization in progress (Figure 4-9). The optimization read requires several minutes to complete.

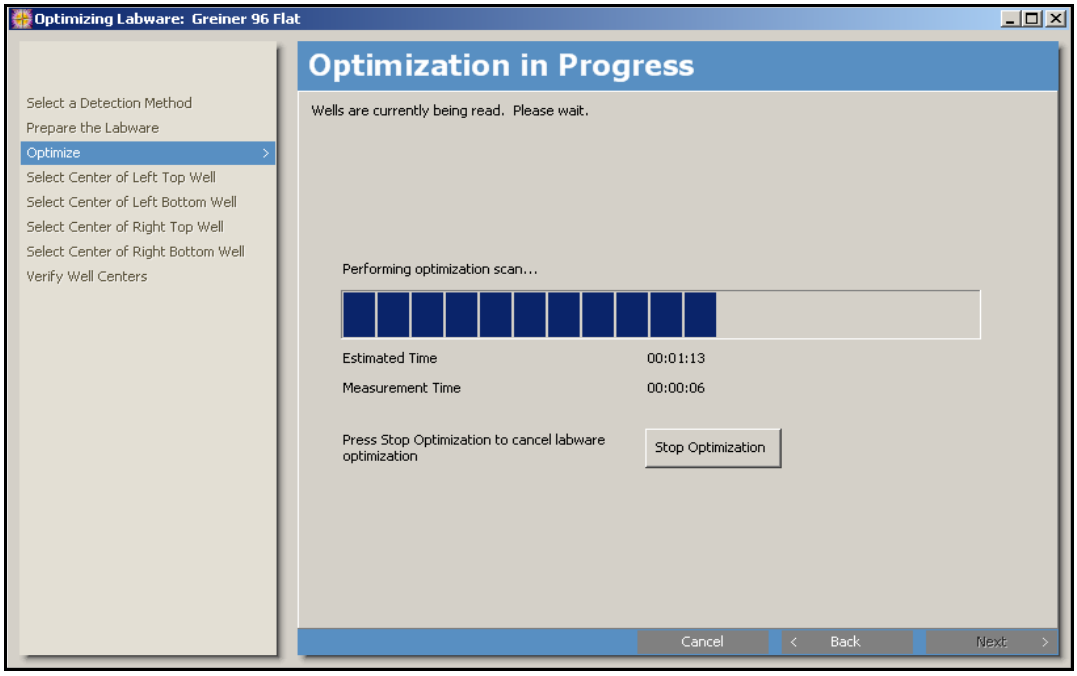


Figure 4-9. Labware optimization in progress

During the optimization read:

Choose **Stop Optimization** to cancel the optimization process and close Optimizing Labware without saving the optimization data.

When the optimization read is complete:

Choose **Next** to select the centers of the four corner wells (refer to Section 4.6.4, *Selecting the Centers of the Four Corner Wells*).

OR

Choose **Cancel** to close Optimizing Labware without performing the optimization.

4.6.4 Selecting the Centers of the Four Corner Wells

Use Select Center to precisely define the centers of the corner wells read in the optimization (Figure 4-10). Select Center displays an image of the well generated by the absorbance measurement. Well centers are defined graphically by dragging cross hairs to the position visually identified as the center. Select Center is performed for each corner well individually.

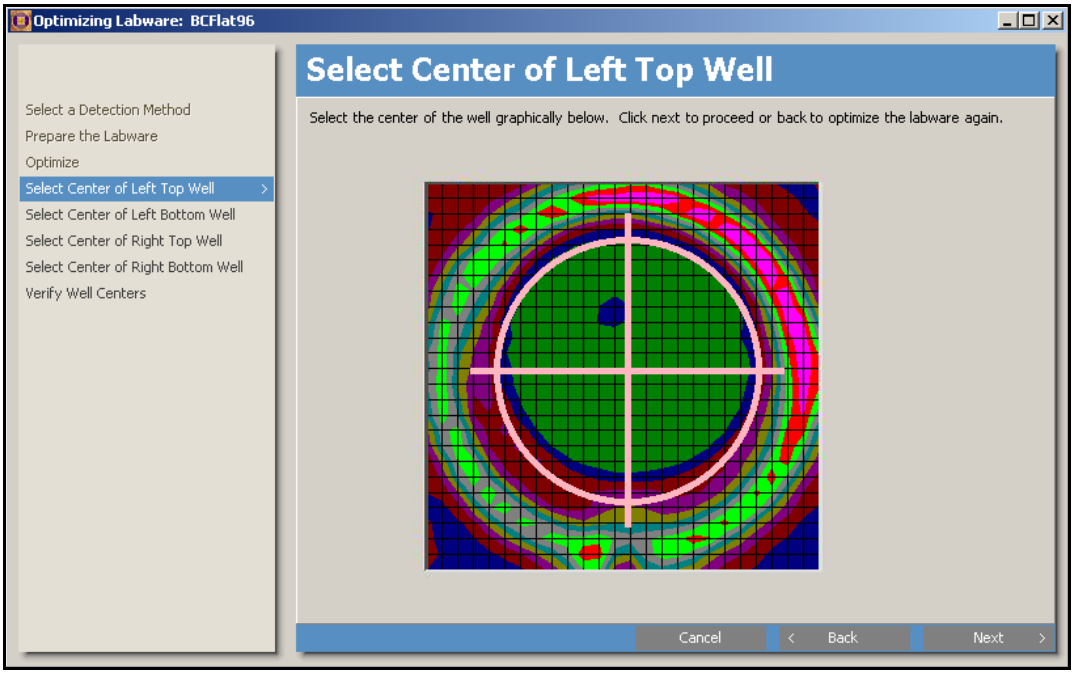


Figure 4-10. Selecting the well center

To define the centers of the wells:

1. Place the cursor in the well image, then click and drag the cross hairs to the desired center of the well.
2. Choose **Next** to define the center of the next well read. When all four well centers are defined, Verify Well Centers appears (refer to Section 4.6.5, *Verifying Well Centers*).

   OR

   Choose **Cancel** to close Optimizing Labware without completing the optimization.

4.6.5 Verifying Well Centers

Use Verify Well Centers to verify that the x and y offsets and distances between rows and columns are correct (Figure 4-11). If desired, the offsets, distances, and lot name may be edited in Verify Well Centers.

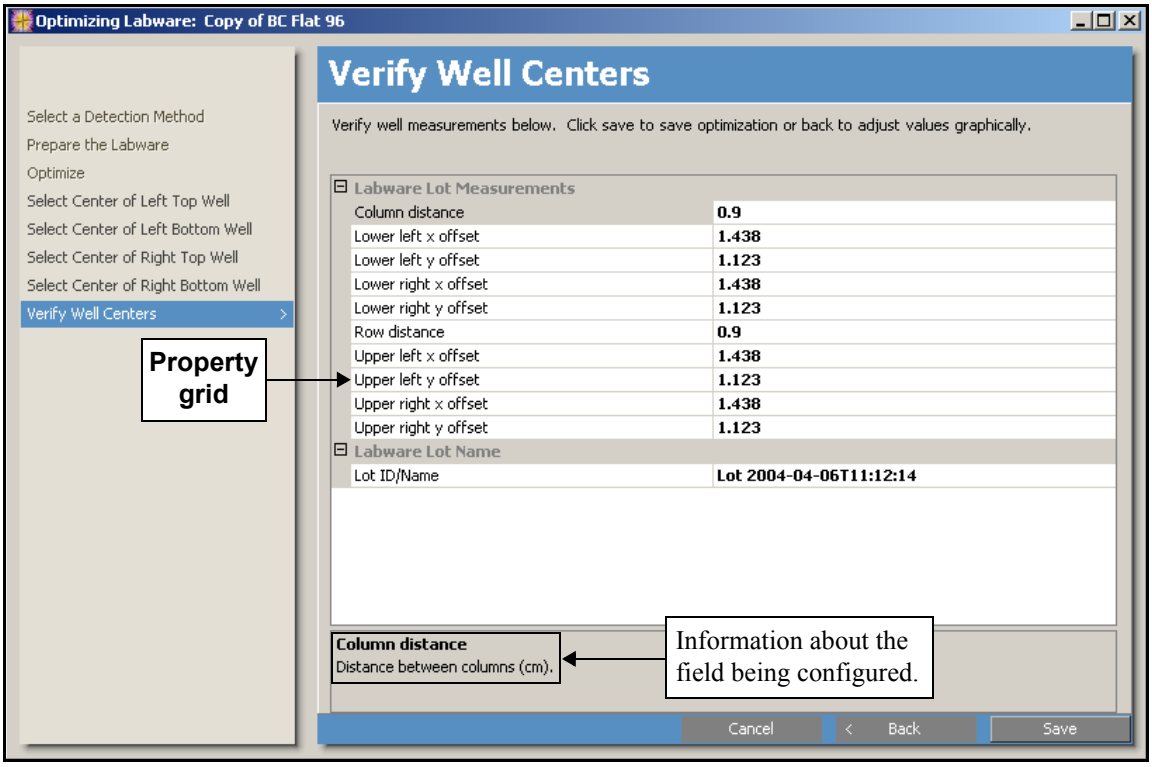


Figure 4-11. Verifying well centers

To verify and edit offsets, distances, and lot name:

1. If necessary, click the **+** next to Labware Lot Dimensions to display the fields in the category.

   Note: More information about the field being defined is displayed below the property grid (Figure 4-11).

2. In Column distance, verify the distance between columns and edit the dimension. *All* offsets and well dimensions are entered in centimeters, if desired.
3. Verify the x and y offsets for the lower two wells and edit the dimensions, if desired.
4. In Row distance, verify the distance between rows and edit the dimension, if desired.
5. Verify the x and y offsets for the upper two wells and edit the dimensions, if desired.
6. If necessary, click the **+** next to Labware Lot Name to display the default name assigned to the new labware lot.
7. Enter a new **Lot ID/Name**, if desired.

8. Choose **Save** to save the optimization data and create the new labware lot.

   Note: To use the optimized lot in a measurement protocol, open the labware for editing and select the new **Labware Lot** (refer to Section 4.3.2, *Selecting and Editing Labware Lots*).

   OR

   Choose **Back** to redefine well centers graphically (refer to Section 4.6.4, *Selecting the Centers of the Four Corner Wells*).

   OR

   Choose **Cancel** to close Optimizing Labware without completing the optimization.

4.7 Exporting and Importing Labware

User-defined labware saved in the Labware Selection List can be exported to an XML file and imported later to restore that configuration or share it with a copy of Multimode Detection Software installed on another computer.

Exporting default labware included with the software installation is not necessary. Since default labware may not be edited, deleted, or overwritten, it is available to users at all times. Because of this, importing default labware from an XML export file is not permitted by the software.

To export labware:

1. In the Labware Selection List, select the labware to export.
2. From the File menu, choose **Export>Labware**. Browse for Folder appears (Figure 4-12).

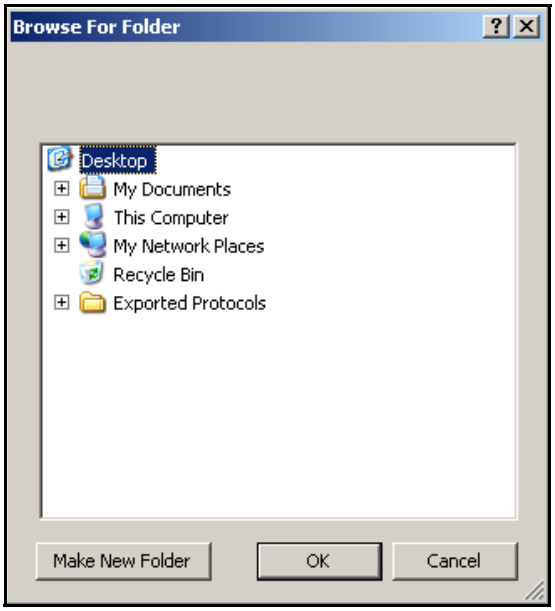


Figure 4-12. Choosing the folder where exported labware will be saved

3. In Browse for Folder, browse to the folder where the exported labware will be saved.

   OR

   Choose **Make New Folder** to create a new folder where the exported labware will be saved.

4. Choose **OK** to export the labware. The exported labware is saved using the default file name format, Labware__*LabwareName*.xml. In order to import the file at a later date, the filename must not be changed.

   OR

   Choose **Cancel** to stop the operation without exporting labware.

To import labware from an exported XML file:

1. From the File menu, choose **Import>Labware**. Open appears.
2. In Open, browse to and select the desired XML file to import.
3. Choose **Open**. The labware is imported to the Labware Selection List.

5 Defining, Editing, and Running Protocols

5.1 Overview

A protocol stores all parameters required to perform a measurement, including technique type, labware type, detection method(s), and preparation methods, such as shaking. Protocols are stored in the Protocol Selection List (Figure 5-1), which provides access to all protocol functions:

- *Creating Protocols* (Section 5.2).
- *Running Protocols* (Section 5.3).
- *Editing Protocols* (Section 5.4).
- *Copying Protocols* (Section 5.5).
- *Deleting Protocols* (Section 5.6).
- *Printing Protocol Configuration Information* (Section 5.7).
- *Exporting and Importing Protocols* (Section 5.8).

To select protocols and access protocol functions:

From the navigation pane, choose **Protocols**. The Protocol Selection List appears (Figure 5-1).

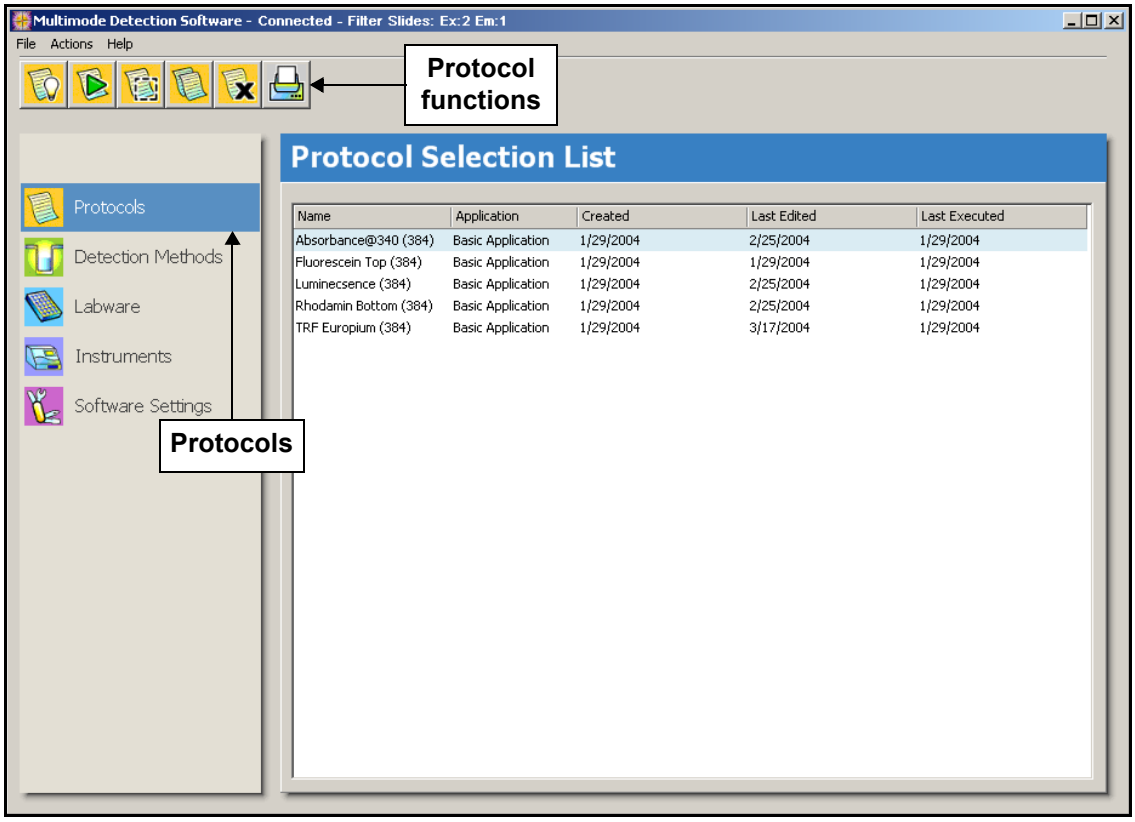


Figure 5-1. Protocol Selection List

5.2 Creating Protocols

New protocols are defined in Create Protocol, which guides the creation process with a wizard-type interface. Creating a new protocol requires:

- *Configuring General Settings* (Section 5.2.1).
- *Selecting the Technique Type* (Section 5.2.2).
- *Selecting the Labware Type Used in the Protocol* (Section 5.2.3).
- *Configuring Labware Layout Settings* (Section 5.2.4).
- *Configuring Detection and Preparation Methods* (Section 5.2.5).
- *Configuring Output Settings* (Section 5.2.6).

To create and configure a new protocol:

From the toolbar, choose **Create a new protocol**.

OR

From the Actions menu, choose **Create a new protocol**.

OR

Right-click in the Protocol Selection List and choose **Create a new protocol** from the menu. Create Protocol appears (Figure 5-2).

5.2.1 Configuring General Settings

Use General Settings to define the protocol name and enter any related notes about the protocol (Figure 5-2).

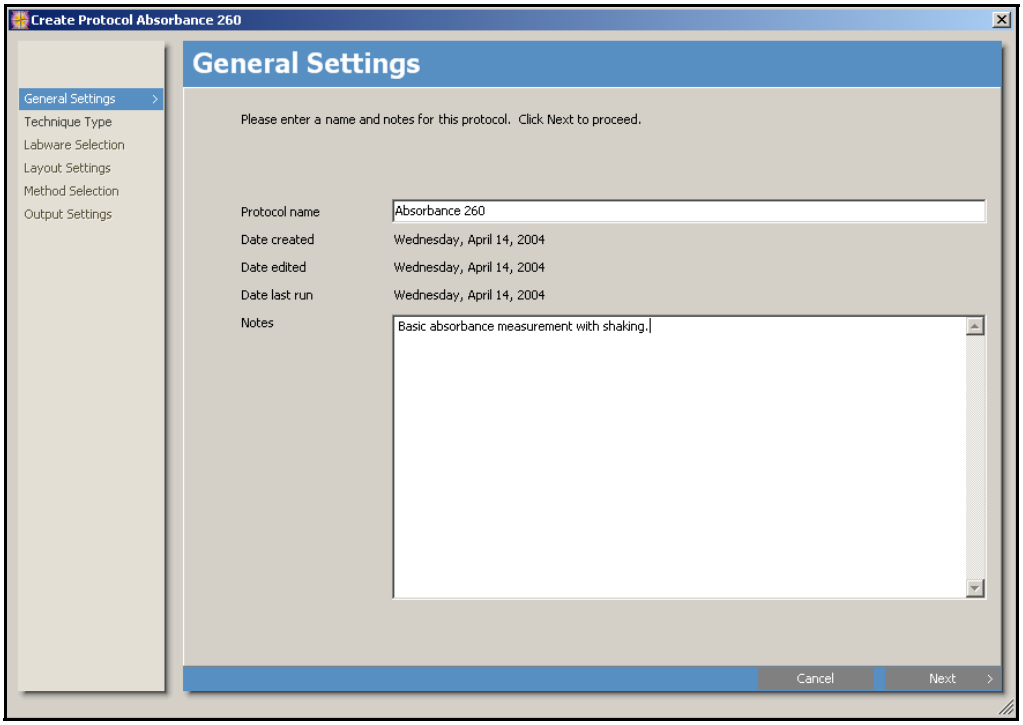


Figure 5-2. Defining a protocol name and entering notes about the protocol

To configure the general settings for a new protocol:

1. In Protocol name, enter a unique name for the protocol. The name must be entered before the next configuration screen in the wizard can be accessed.
2. In Notes, enter a description of the new protocol, if desired.
3. Choose **Next** to select the measurement technique used in the protocol (refer to Section 5.2.2, *Selecting the Technique Type*).

   OR

   Choose **Cancel** to close Create Protocol without saving the new protocol.

5.2.2 Selecting the Technique Type

Use Technique Type to select the measurement technique performed by the protocol (Figure 5-3). Only techniques supported by the instrument and for which detection methods have already been defined are available. For more information about technique types, refer to Chapter 3, *Creating and Editing Detection Methods*.

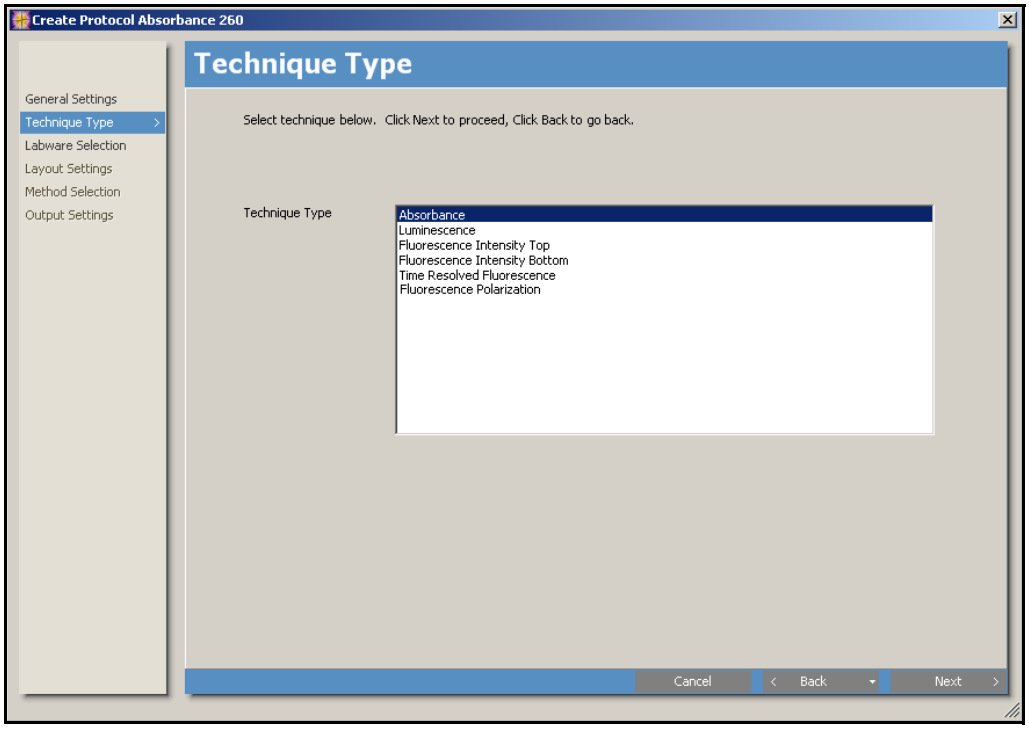


Figure 5-3. Selecting the measurement technique for a DTX 880

To select a technique type:

1. In Technique Type, choose the desired technique.
2. Choose **Next** to select the type of labware used in the protocol (refer to Section 5.2.3, *Selecting the Labware Type Used in the Protocol*).

   OR

   Choose **Back** to configure General Settings for the protocol (refer to Section 5.2.1, *Configuring General Settings*).

   OR

   Choose **Cancel** to close Create Protocol without saving the new protocol.

5.2.3 Selecting the Labware Type Used in the Protocol

Use Labware Selection to select the type of labware used in the protocol (Figure 5-4). Labware must be configured prior to configuring the protocol. Only labware configured for the selected technique type is available. Labware cannot be edited once it is used in a protocol. Refer to Chapter 4, *Creating and Editing Labware*, for detailed information about creating and configuring labware.

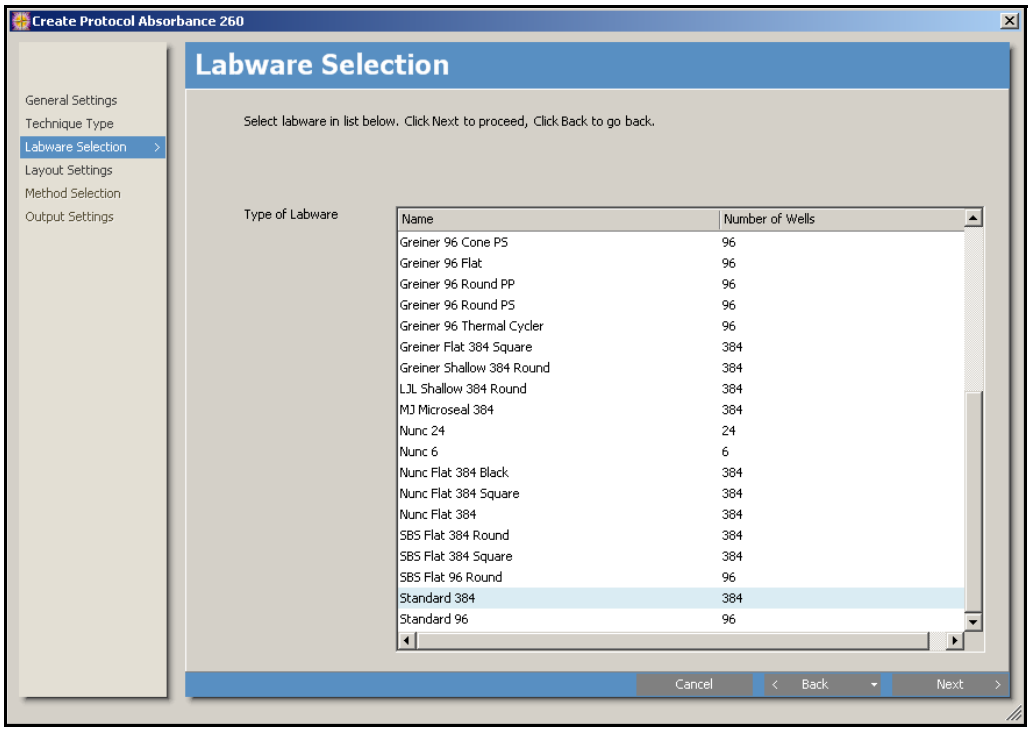


Figure 5-4. Selecting the type of labware used in the protocol

To select labware:

1. Select the desired Type of Labware from the list.
2. Choose **Next** to configure the plate layout (refer to Section 5.2.4, *Configuring Labware Layout Settings*).

   OR

   Choose **Back** to change the measurement technique performed by the protocol (refer to Section 5.2.2, *Selecting the Technique Type*).

   OR

   Choose **Cancel** to close Labware Selection without saving the new protocol.

5.2.4 Configuring Labware Layout Settings

Use Layout Settings to configure how wells on the plate are read (Figure 5-5). Settings are configured in two tabs:

- Layout Selection Settings — configures which wells are read in the measurement (Figure 5-5).
- Reading Direction Settings — configures the order of how wells are read: row-by-row, column-by-column, or well-by-well (Figure 5-6).

To configure labware layout settings:

1. In Layout Settings, choose the **Layout Selection Settings** tab to view the layout of the selected labware (Figure 5-5).

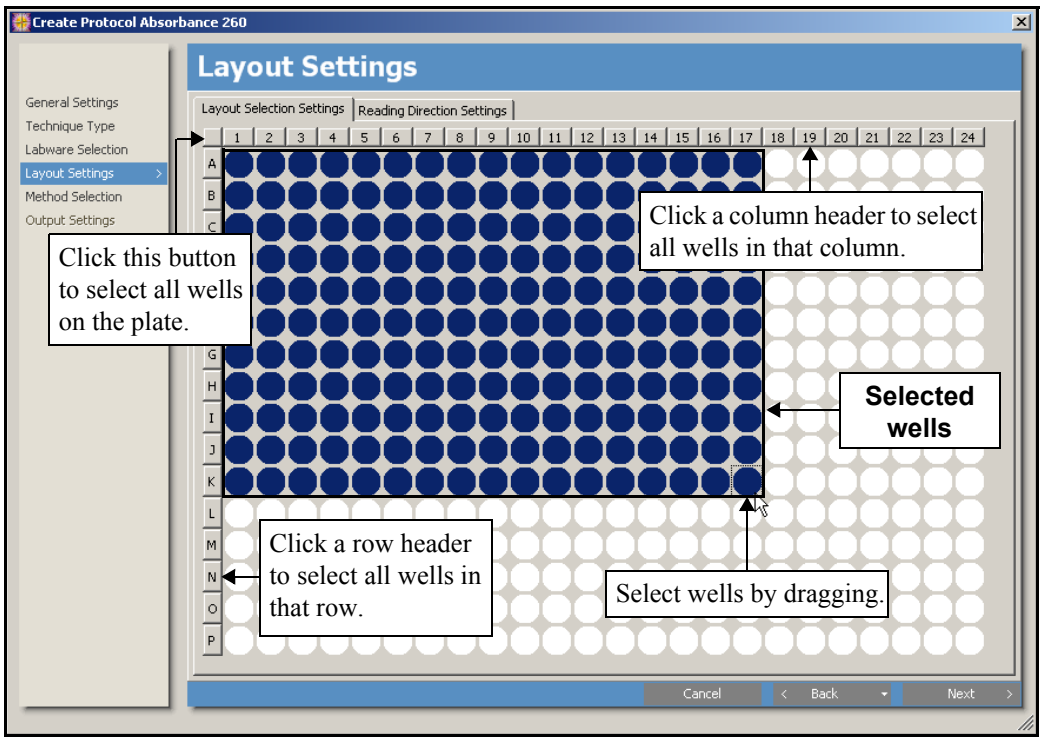


Figure 5-5. Selecting which wells are measured on a 384-well plate

2. Select the desired wells to read:
    - **All wells on the plate** — click the small button in the upper left corner of the plate layout display.
    - **All wells in a column** — click the desired column header. Multiple columns may be selected.
    - **All wells in a row** — click the desired row header. Multiple rows may be selected.
    - **Individual wells** — click each desired well.
    - **Groups of wells** — click and drag over the desired group of wells. Multiple groups may be selected.

Note: Wells are deselected using the methods listed above; for example, clicking the header of a selected column deselects all wells in the column.

3. Choose the **Reading Direction Settings** tab to configure the order in which wells are read in the protocol (Figure 5-6).

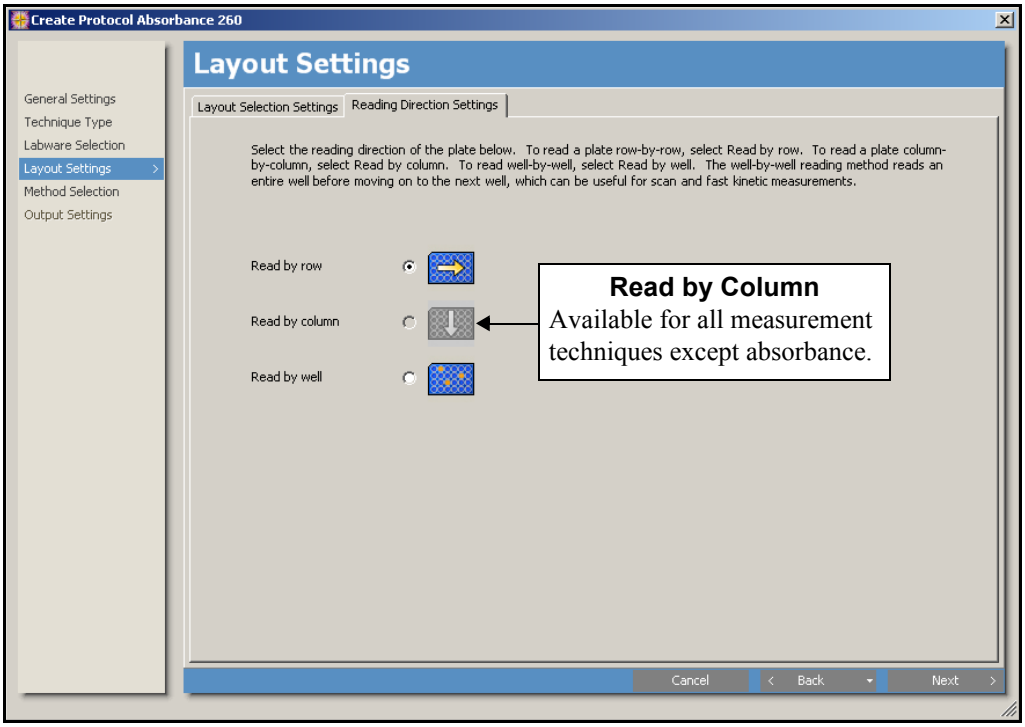


Figure 5-6. Selecting how wells on the plate are read

4. Select the desired reading direction:
    - **Read by row** — Reads plates row-by-row.
    - **Read by column** — Reads plates column-by-column. Available for all measurement techniques except absorbance.
    - **Read by well** — Reads each well individually before reading the next well, which is useful for kinetic and scan measurements.

5. Choose **Next** to configure detection and preparation methods (refer to Section 5.2.5, *Configuring Detection and Preparation Methods*).

   OR

   Choose **Back** to select a different labware type (refer to Section 5.2.3, *Selecting the Labware Type Used in the Protocol*).

   OR

   Choose **Cancel** to close Create Protocol without saving the new protocol.

5.2.5 Configuring Detection and Preparation Methods

Use Method Selection to select and configure detection and preparation methods (Figure 5-7). Two configuration modes are available:

- **Basic** — a single detection method may be selected and optionally configured as a kinetic or scan measurement, if desired. Microplate shaking may also be configured (refer to Section 5.2.5.1, *Configuring a Method in Basic Mode*).
- **Advanced** — provides the configuration options available in basic mode and adds the ability to define multiple detection methods and to configure preparation methods, such as shaking and temperature control (880 only). Detection and preparation methods may be arranged in any order within the execution sequence (refer to Section 5.2.5.2, *Configuring a Method in Advanced Mode*).

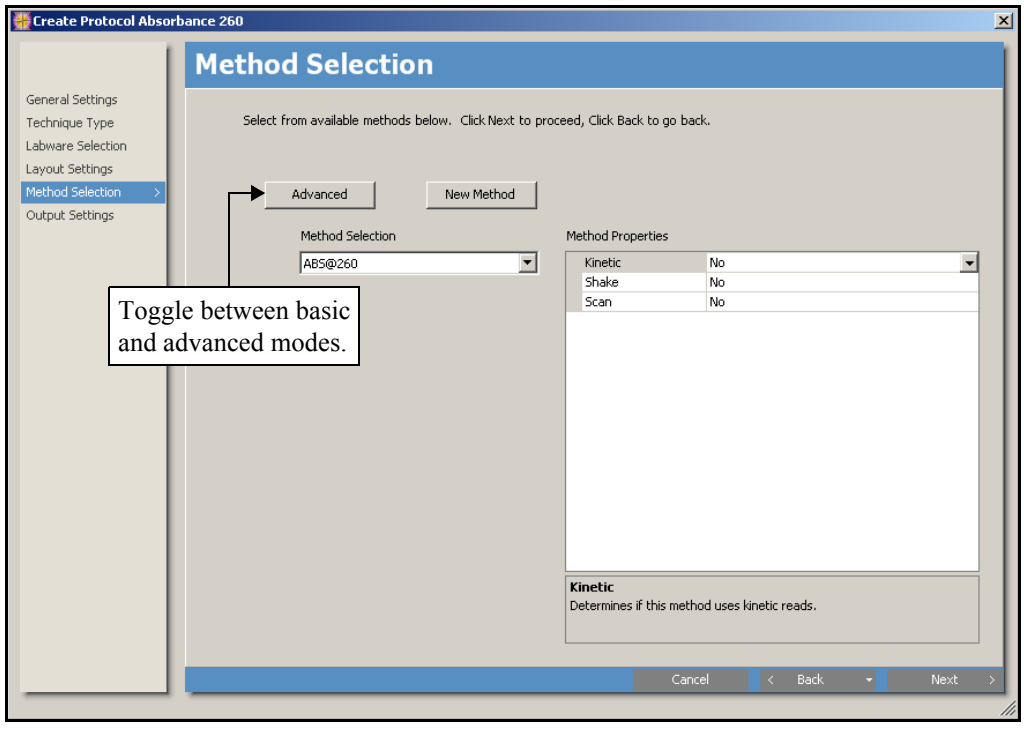


Figure 5-7. Configuring parameters in Method Selection

5.2.5.1 Configuring a Method in Basic Mode

Configuring a method in basic mode includes selecting a detection method and configuring properties for a kinetic or scan measurement and shaking, if desired.

To select and configure a method in basic mode:

Basic

1. In Method Selection, choose **Basic** to toggle from advanced mode, if necessary. Basic mode configuration parameters appear (Figure 5-8). Refer to Section 5.2.5.2, *Configuring a Method in Advanced Mode*, to configure a method with multiple detection and/or preparation methods.

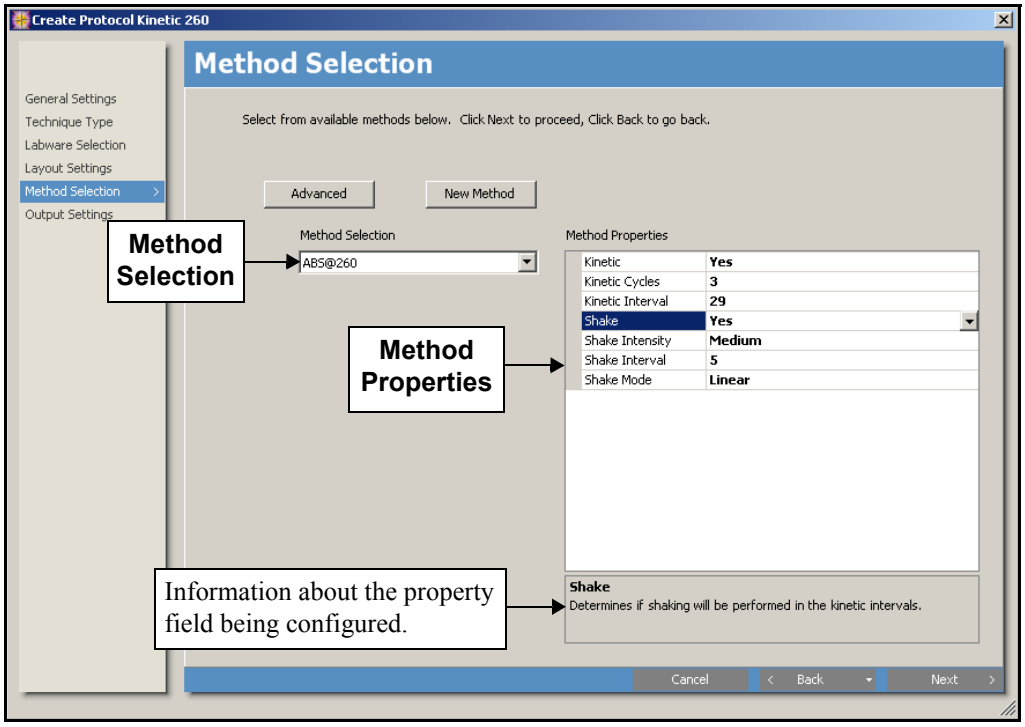


Figure 5-8. Configuring a method in basic mode

2. In Method Selection, select the detection method for use with the protocol. Only detection methods of the technique selected for the protocol are available (refer to Section 5.2.2, *Selecting the Technique Type*).

   OR

   Choose **New Method** to create a new detection method using the currently-selected technique type. Refer to Chapter 3, *Creating and Editing Detection Methods*, for more information about creating detection methods.

3. To configure a kinetic measurement, in Method Properties, click in Kinetic and choose **Yes**. Kinetic measurements read wells several times at specified intervals.

   Note: If a kinetic measurement not desired, proceed to step 6.

4. In Kinetic Cycles, enter the number of measurement cycles to be performed. Kinetic measurements may be set to perform 1 to 100 cycles.

5. In Kinetic Interval, enter the length of time, in seconds, between each measurement of the same well.

   Note: The minimum kinetic interval is populated automatically in Kinetic Interval, and is determined by the labware type and layout settings configured in the protocol. The maximum interval between each measurement cycle is 65,535 seconds.

6. To shake the microplate before performing the measurement, click in Shake and choose **Yes**. In kinetic measurements, the plate is shaken for the specified intensity and interval before each measurement cycle.

   Note: If shaking is not desired, proceed to step 10.

CAUTION: Shake low density plates, such as 6- or 48-well plates, at low speed only. Shaking low density plates at higher speeds may cause liquid in wells to spill.

7. In Shake Intensity, select the desired intensity of shaking: **Low**, **Medium**, or **High**.
8. In Shake Interval, enter the length of time in seconds (0–60) to shake the microplate.
9. In Shake Mode, select the desired shaking pattern.
   - **Linear** — shakes from side to side.
   - **Orbital** — shakes labware in a circular pattern.
   - **Squared** — shakes labware in a square pattern, moving at right angles.
10. To configure an area or linear scan, click in Scan and select **Yes**. Area scans read a number of measurement points arranged in a grid pattern across each well. Linear scans read a number of points in a linear axis crossing the center of each well. Scan measurements are not available when a kinetic measurement is configured.

    Note: Scans may only be performed on samples on plates with 96 or fewer wells.

    Note: If a scan measurement is not desired, proceed to step 15.

11. To configure the type of scan measurement and the number of points measured, click in Scan Points and choose the configuration button. Scan Selection Editor appears (Figure 5-9).

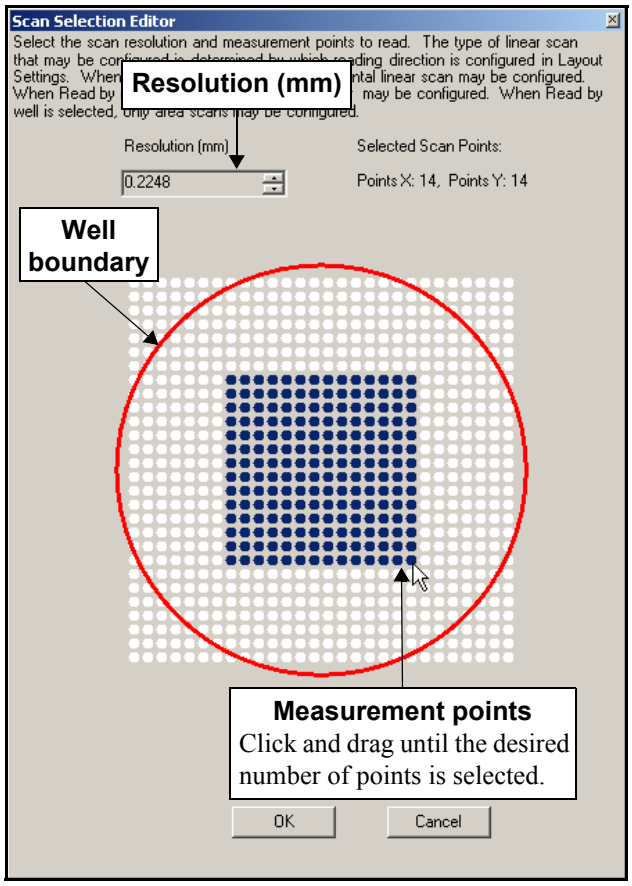


Figure 5-9. Configuring an area scan

12. In Resolution (mm), use the up and down arrows to choose the proximity of measurement points. Choosing a smaller value increases the number of measurement points available; choosing a larger value decreases the number of points available. Available resolutions are determined by the type of labware selected and configured in the protocol. Refer to Sections 5.2.3, *Selecting the Labware Type Used in the Protocol* and 5.2.4, *Configuring Labware Layout Settings*, for more information.

13. To configure an area scan, click anywhere inside the well boundary, except the center row, and drag until the desired number of measurement points is selected (Figure 5-9).

    OR

    To configure a linear scan, click anywhere on the center row or column inside the well boundary and drag towards the boundary until the desired number of measurement points is selected (Figure 5-10).

Note: The type of scan that may be configured is determined by the Reading Direction Settings in Layout Settings (refer to Section 5.2.4, *Configuring Labware Layout Settings*). For example, horizontal linear scans along the center row may be configured only when Read by row is selected, while vertical linear scans along the center column may be configured only when Read by column is selected. When Read by well is selected, only area scans may be configured.

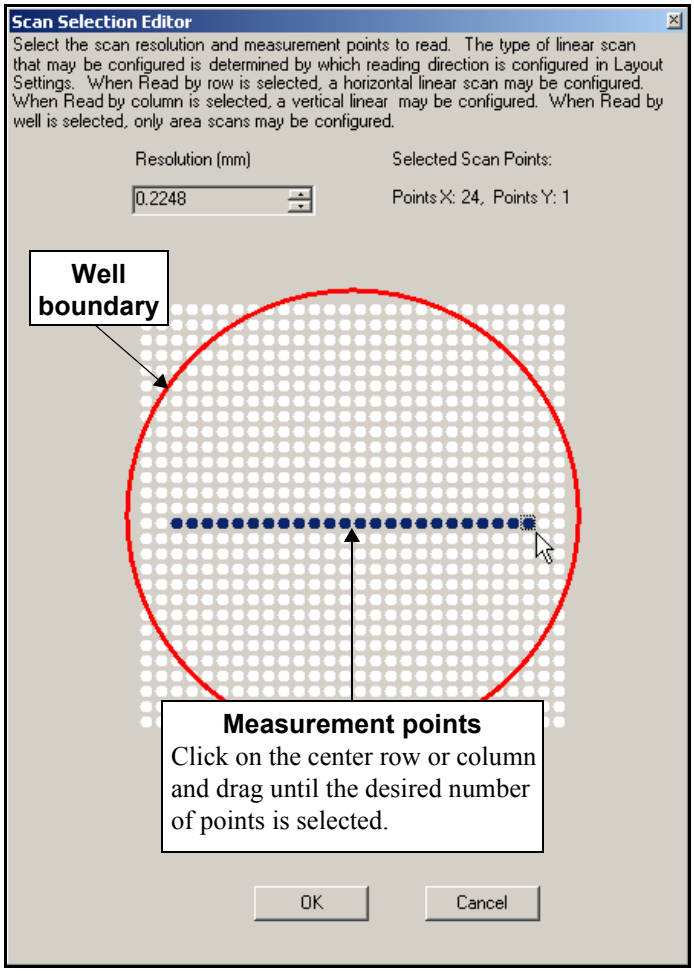


Figure 5-10. Configuring a linear scan

14. Choose **OK** to save the scan configuration and close the Scan Selection Editor.

    OR

    Choose **Cancel** to close the Scan Selection Editor without updating changes.

15. Choose **Next** to configure how measurement results are saved and/or printed (refer to Section 5.2.6, *Configuring Output Settings*).

    OR

    Choose **Back** to configure the plate layout (refer to Section 5.2.4, *Configuring Labware Layout Settings*).

    OR

    Choose **Cancel** to close Create Protocol without saving the new protocol.

5.2.5.2 Configuring a Method in Advanced Mode

Configuring methods in advanced mode includes selecting detection and preparation methods, arranging the execution sequence of methods, and optionally configuring properties for a kinetic or scan measurement.

Protocols that require multiple measurements at different wavelengths, such as multiwavelength absorbance or fluorescence resonance energy transfer (FRET) measurements, must be configured in advanced mode.

To select and configure a method in advanced mode:

Advanced

1. In Method Selection, choose **Advanced** to toggle from basic mode, if necessary. Advanced mode configuration parameters appear (Figure 5-11). Refer to Section 5.2.5.1, *Configuring a Method in Basic Mode*, to configure a protocol that requires only one detection method and no preparation methods.

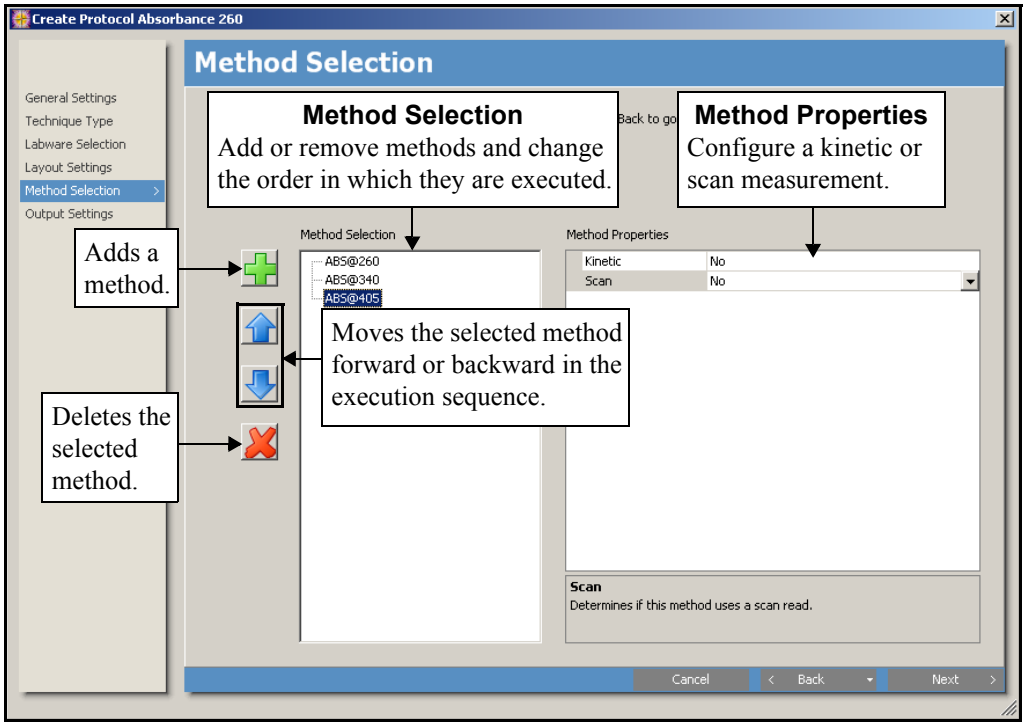


Figure 5-11. Configuring a method in advanced mode

2. Add a detection method by choosing **Add a method to this protocol**, then selecting the desired Detection Method from the menu that appears.

   OR

   Right-click in Method Selection and choose **Add Method>Detection Methods>** and the desired detection method from the menu that appears.

   OR

   Choose **New Method** to create a new detection method. Refer to Chapter 3, *Creating and Editing Detection Methods*, for detailed information about creating detection methods.

3. Configure Method Properties as desired. Refer to Section 5.2.5.1, *Configuring a Method in Basic Mode*, for more information about configuring method property parameters.

4. Add and configure additional detection methods by repeating steps 2 and 3 for each method desired.

5. Add a preparation method by choosing **Add a method to this protocol**, then selecting **Preparation Methods>Shake** or **Set Temperature** from the menu that appears.

   OR

   Right-click in Method Selection and choose **Add Method>Preparation Methods>Shake** or **Set Temperature** from the menu that appears.

   - ***DTX 880*** — Set Temperature is available only when controlling this instrument.

6. Configure the preparation method as desired. Refer to Section 5.2.5.2.1, *Configuring a Shake Preparation Method*, for more information about shaking. Refer to Section 5.2.5.2.2, *Configuring a Set Temperature Preparation Method (880 only)*, for more information about configuring temperature control in the protocol.

7. Add and configure additional preparation methods by repeating steps 5 and 6 for each method desired.

8. To reorder the detection and preparation methods in the method execution sequence, select the method to be moved and choose **Move the selected method forward** or **Move the selected method backward**.

   OR

   In Method Selection, right-click on the desired detection or preparation method and choose **Move Up** or **Move Down**.

9. To delete a detection or preparation method from the protocol, in Method Selection, select the method to be deleted and choose **Delete the selected method from this protocol**.

   OR

   In Method Selection, right-click on the desired detection or preparation method and choose **Remove Method**.

10. Choose **Next** to configure Output Settings (refer to Section 5.2.6, *Configuring Output Settings*).

    OR

    Choose **Back** to change Labware Layout Settings (refer to Section 5.2.4, *Configuring Labware Layout Settings*).

    OR

    Choose **Cancel** to close Create Protocol without saving the new protocol.

5.2.5.2.1 Configuring a Shake Preparation Method

A Shake preparation method shakes the microplate at the point in the execution sequence where it is positioned. For example, Figure 5-12 shows a protocol where the microplate is shaken before absorbance measurements are performed at three wavelengths.

Note: A Shake preparation method only shakes the plate once for the configured duration and intensity. To perform interval shaking between each cycle in a kinetic measurement, shaking must be configured in Method Properties for the desired detection method (refer to Section 5.2.5.1, *Configuring a Method in Basic Mode*).

To configure a Shake preparation method:

1. In Method Selection, choose the desired **Shake** preparation method. Method Properties for the selected method appear (Figure 5-12).

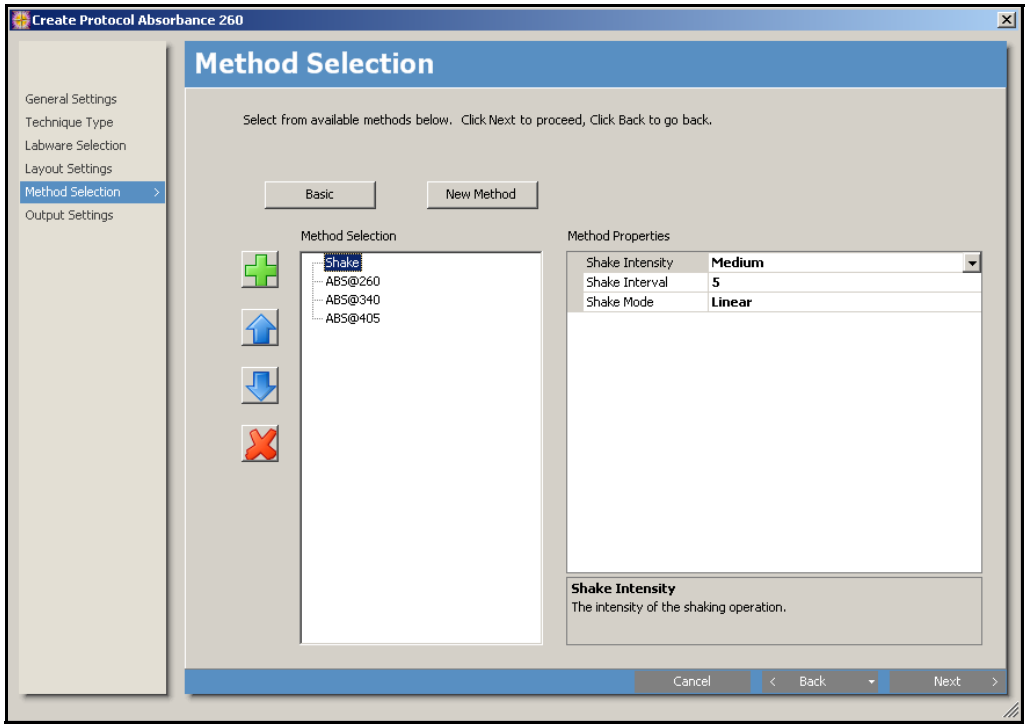


Figure 5-12. Configuring a Shake preparation method

CAUTION: Shake low density plates, such as 6- or 48-well plates, at low speed only. Shaking low density plates at higher speeds may cause liquid in wells to spill.

2. In Shake Intensity, select the desired intensity of shaking: **Low**, **Medium**, or **High**.

3. In Shake Interval, enter the length of time in seconds (**0**–**60**) to shake the microplate.
4. In Shake Mode, select the desired shaking pattern.
    - **Linear** — shakes from side to side.
    - **Orbital** — shakes labware in a circular pattern.
    - **Squared** — shakes labware in a square pattern, moving at right angles.

5. To move the Shake preparation method to a different location in the execution sequence, in Method Selection, select the method and choose **Move the selected method forward** or **Move the selected method backward**.

    OR

    In Method Selection, right-click on the Shake preparation method and choose **Move Up** or **Move Down**.

5.2.5.2.2 Configuring a Set Temperature Preparation Method (880 only)

A Set Temperature preparation method sets the temperature inside the microplate chamber. The temperature is set by heating the microplate chamber; cooling the chamber is not supported by the instrument. Depending on the measurement wavelength and light source used in the protocol, the set temperature may range from 3°C (5.4°F) or 4°C (7.2°F) above ambient to 45°C (113°F).

It takes a minimum of 30 minutes for the instrument to reach the desired temperature. The actual time required depends on the relative change in temperature. When a protocol with a Set Temperature preparation method is run, the protocol does not pause to wait for the desired temperature to be reached. To set an exact temperature before starting a protocol run, use Manual Control to set the temperature and check it at regular intervals until the desired temperature is reached (refer to Section 2.3.3.1, *Temperature Control (880 only)*).

Note: Set Temperature preparation methods are useful for kinetic measurements intended to measure the effects of temperature changes on samples.

To configure a Set Temperature preparation method:

1. In Method Selection, choose the desired **Set Temperature** preparation method. Method Properties for the selected method appear (Figure 5-13).

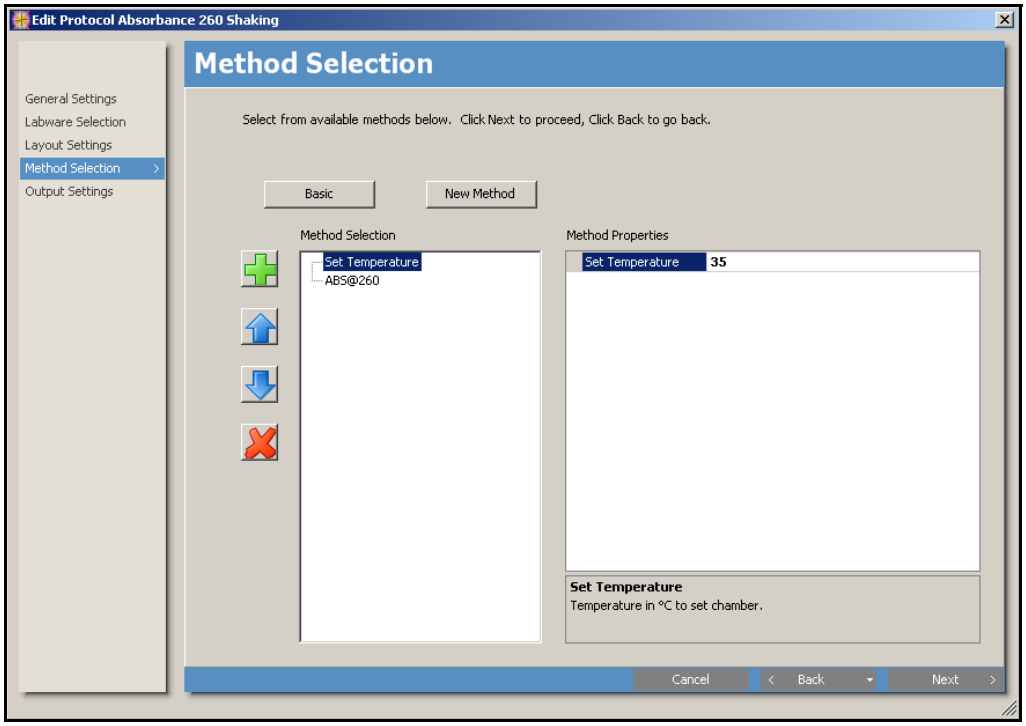


Figure 5-13. Configuring a Set Temperature preparation method

2. In **Set Temperature**, enter the desired microplate chamber temperature in degrees Celsius. In protocols that perform measurements only at visible wavelengths (>369 nm), the minimum temperature that may be set is 3°C (5.4°F) above ambient. When the protocol performs measurements in the UV band using the deuterium lamp, the minimum temperature is 4°C (7.2°F) above ambient. The maximum temperature that may be set is 45°C (113°F).

Note: The temperature remains at the current setting until overridden by another Set Temperature preparation method, by changing the temperature in Manual Control, or by turning the instrument off and on.

3. To move the Set Temperature preparation method to a different location in the execution sequence, in Method Selection, select the method and choose **Move the selected method forward** or **Move the selected method backward**.

   OR

   In Method Selection, right-click on the Set Temperature preparation method and choose **Move Up** or **Move Down**.

5.2.6 Configuring Output Settings

Use Output Settings to configure how measurement results are saved and/or printed after a protocol run (Figure 5-14). Options available include choosing to export results and automatically open them in Microsoft® Excel following a protocol run, choosing the file formats used to save measurement results, and configuring which protocol information and measurement results are included in printouts. An external software application, such as a database, may also be configured to execute after a protocol run.

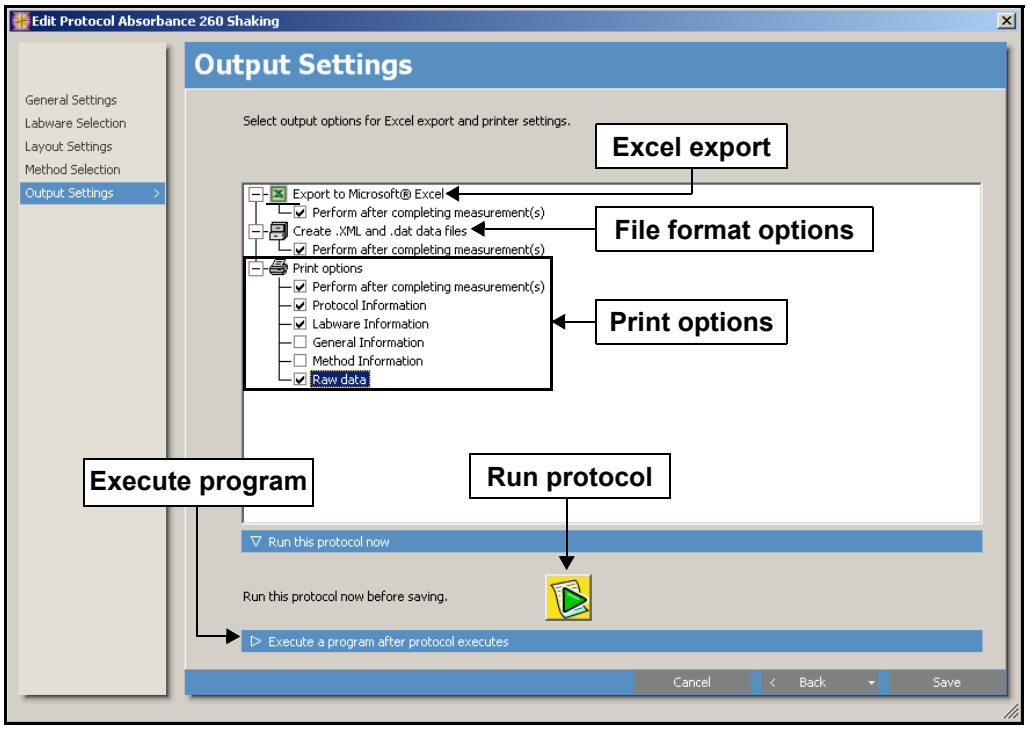


Figure 5-14. Configuring output settings

To configure Output Settings:

1. Choose Excel, data file, and print output options as desired. Table 5-1 describes the options available.

2. Choose **Run this protocol now before saving** to run the protocol immediately. Refer to Section 5.3, *Running Protocols*, for more information about running protocols.
3. Choose **Execute a program after protocol executes** to display options for configuring an external software application to run immediately after the protocol run is completed. Refer to Section 5.2.6.1, *Configuring a Program to Run after Protocol Executes*, for more information.
4. Choose **Save** to save the protocol and close Create Protocol.

   OR

   Choose **Cancel** to close Create Protocol without saving the new protocol.

   OR

   Choose **Back** to configure methods used in the protocol (refer to Section 5.2.5, *Configuring Detection and Preparation Methods*).

Table 5-1. Output Options

Output Option	Description
Export to Microsoft® Excel	
Perform after completing measurement(s)	Saves results in a format compatible with Microsoft® Excel, and automatically opens Excel when the protocol run is completed. **Note:** Versions of Excel prior to Office 2000 are not supported by the Export to Microsoft Excel function, but can open measurement results stored in tab-delimited data (.dat) files.
Create .XML and .dat data files	
Perform after completing measurement(s)	Automatically exports measurement results to a tab-delimited data (*.dat) file and an XML file. These files may be opened by software applications compatible with tab-delimited data or XML files. **Note:** The directory where the data files are saved is configured in System Settings. Refer to Section 1.4.2.4, *Choosing the Directory Where Measurement Results are Saved*.
Print options	
Perform after completed measurement(s)	Automatically prints the results after completing a protocol run. This option *must* be selected to print measurement results.
Raw Data	Prints raw data measured for each sample. For kinetic measurements, prints raw data measured in each cycle. For scan measurements, prints raw data for all measurement points. **Note:** Printing kinetic and scan measurement results may require numerous pages, depending on the number of measurement cycles performed or points measured.
Protocol Information	Prints the name of the protocol underneath the printout header.
Labware Information	Prints information about the labware used, including plate type, number of wells measured, lot name, and the date that the lot was optimized.
General Information	Prints general information about the protocol including when the protocol was created or last edited and performed.
Plate Graph	Prints graphs of kinetic and scan measurement results. Only available for configuration when a kinetic, area scan, or linear scan measurement is configured in the protocol.
Method Information	Prints information about the detection method(s) used in the protocol.

5.2.6.1 Configuring a Program to Run after Protocol Executes

Output Settings can be configured to open an external software application, such as a custom data logger or database, after completing a protocol run. If the selected application supports entering commands using a command line interface, a specific command for the application may also be configured.

To configure a program to run after a protocol is executed:

1. In Output Settings, choose **Execute a program after protocol executes**. Execute Program expands to show the fields that may be configured (Figure 5-15).

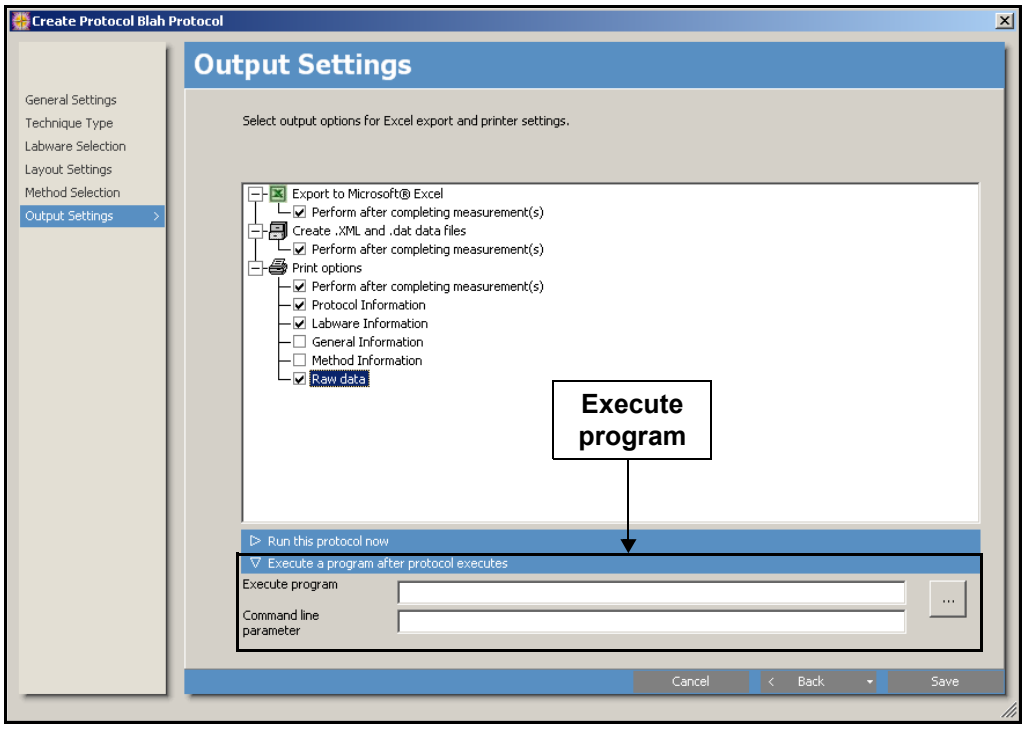


Figure 5-15. Configuring an external software application in Output Settings

2. In Execute Program, enter the complete path of the program to run; for example, **C:\Program Files\Data Logger\MyCustomDataLogger.exe**.

   OR

   Choose **Browse to find an executable program**. Open appears.

3. In Open, browse to the location of the desired application and select it.
4. Choose **Open** to return to Output Settings. The selected path appears in Execute Program.

   OR

   Choose **Cancel** to close Open without selecting an application.

5. If the selected application supports command line parameters, in Command line parameter, enter the desired parameter.

5.3 Running Protocols

Saved protocols may be accessed and run at any time, either on an instrument or in simulation mode. The run options available are different for each mode. For more information, refer to:

- *Running a Protocol on an Instrument* (Section 5.3.1).
- *Running a Protocol When Simulation Mode is Enabled* (Section 5.3.2).

5.3.1 Running a Protocol on an Instrument

Running a protocol on an instrument performs measurements on samples and outputs results data following the parameters configured in the protocol.

To run a protocol on an instrument:

1. In the Protocol Selection List, select the protocol to run.
2. From the toolbar, choose **Run the selected protocol**.

   OR

   From the Actions menu, choose **Run the selected protocol**.

   OR

   Right-click on the selected protocol and choose **Run the selected protocol** from the menu. Prepare to Run Protocol appears (Figure 5-16).

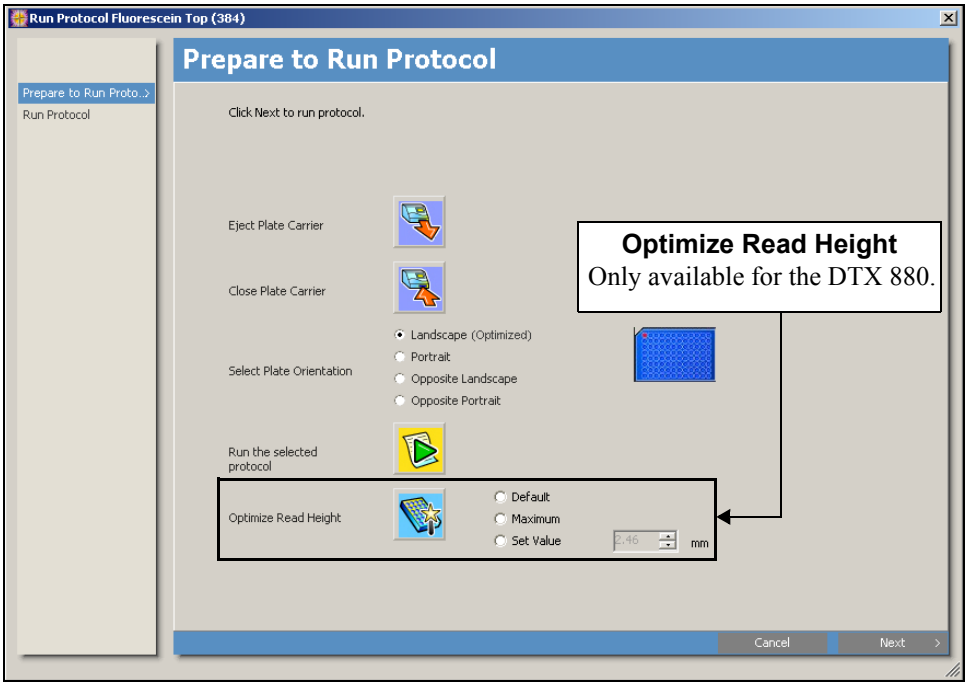


Figure 5-16. Preparing to run a protocol on an instrument

3. Use **Eject Plate Carrier** and **Close Plate Carrier** to load the microplate into the instrument, if necessary.

4. Select the orientation that matches how the plate is positioned on the plate carrier:

   - **Landscape** — the long edges of the plate run parallel to the front of the instrument, with well A1 located in the upper left corner.
   - **Portrait** — the short edges of the plate run parallel to the front of the instrument, with well A1 located in the upper right corner.
   - **Opposite Landscape** — the edges of the plate run parallel to the front of the instrument, with well A1 located in the lower right corner.
   - **Opposite Portrait** — the short edges of the plate run parallel to the front of the instrument, with well A1 located in the lower left corner.

Note: Well A1 is identified in the plate graphic by a red highlight.

5. When preparing to run a fluorescence top, fluorescence polarization, or time-resolved (TRF) protocol, choose **Optimize Read Height** to automatically determine and set the optimal read height used in the protocol run (refer to Section 5.3.1.1, *Optimizing Read Height (880 only)*).

   ➢ ***DTX 880*** — Optimize Read Height is available for this instrument only.

   OR

   Select the read height manually:

   - **Default** — uses the read height defined in Labware Information for the labware type used in the protocol (refer to Section 4.2.1, *Defining Labware Information*).
   - **Maximum** — sets the read height at 10.6 mm, the maximum height supported by the instrument.
   - **Set Value** — manually sets the reading height using the adjacent configuration field. Setting the reading height manually is useful when the height determined by a prior optimization for the same labware type is known.

6. Choose **Run** or **Next** to run the protocol.

   OR

   Choose **Cancel** to close Run Protocol without running the protocol.

5.3.1.1 Optimizing Read Height (880 only)

The DTX 880 features an objective lens that may be moved up and down to optimize the read height used in fluorescence intensity top, fluorescence polarization, and time-resolved fluorescence protocols. Read height is the distance between the top surface of the microplate being read and the lower surface of the objective lens. Optimizing read height matches the focus of the optics with the sample volume. This maximizes the raw signal, which yields the highest precision and maximum sensitivity.

Read height is optimized using the Read Height Optimization Wizard (Figure 5-17). A single sample with a known maximum signal is placed on the same type of microplate used in the protocol. The sample is measured using the same, or very similar, detection method used in the protocol.

The optimized read height is saved in the protocol and is used for all subsequent runs of the protocol until reset by performing a new optimization or manually selecting a read height option.

To optimize read height:

1. In Run Protocol, select **Optimize Read Height**. The Read Height Optimization Wizard appears (Figure 5-17).

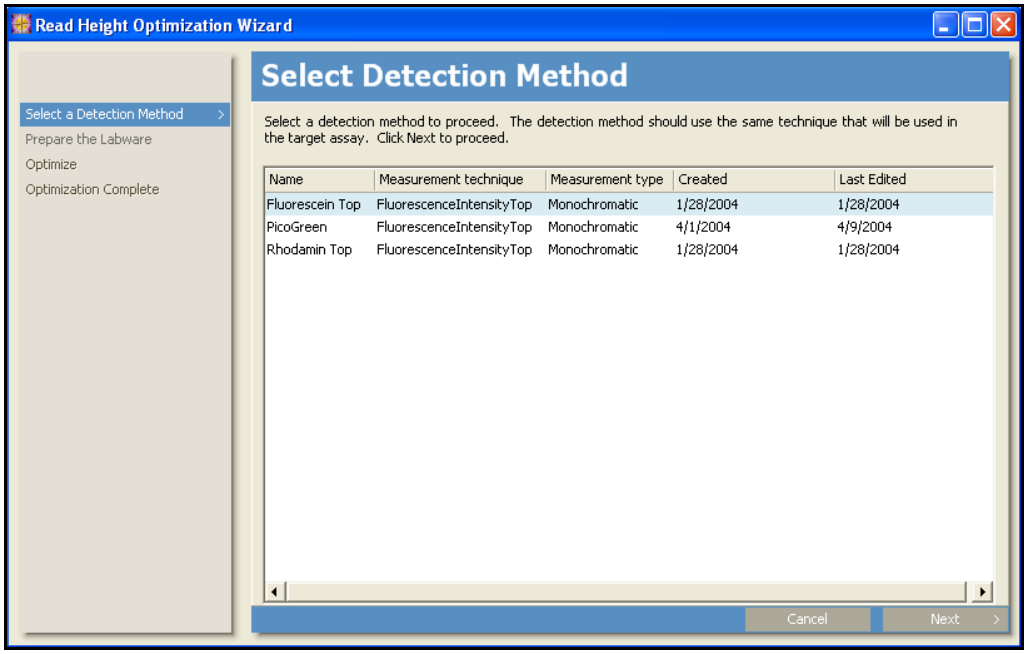


Figure 5-17. Selecting the detection method used in the read height optimization

2. In Select Detection Method, select the detection method used in the optimization. For fluorescence top protocols, choose the same detection method used in the protocol. For fluorescence polarization and time-resolved fluorescence protocols, choose the detection method that is the closest match to that used in the protocol.
3. Choose **Next** to select the well measured to perform the optimization. Prepare Labware appears (Figure 5-18).

   OR

   Choose **Cancel** to close the wizard without performing the optimization.

4. Pipette liquid with a known maximum signal to a single well on the microplate used in the optimization. The liquid volume of the optimization sample should be the same as that of samples measured in the protocol.
5. Load the plate into the instrument.

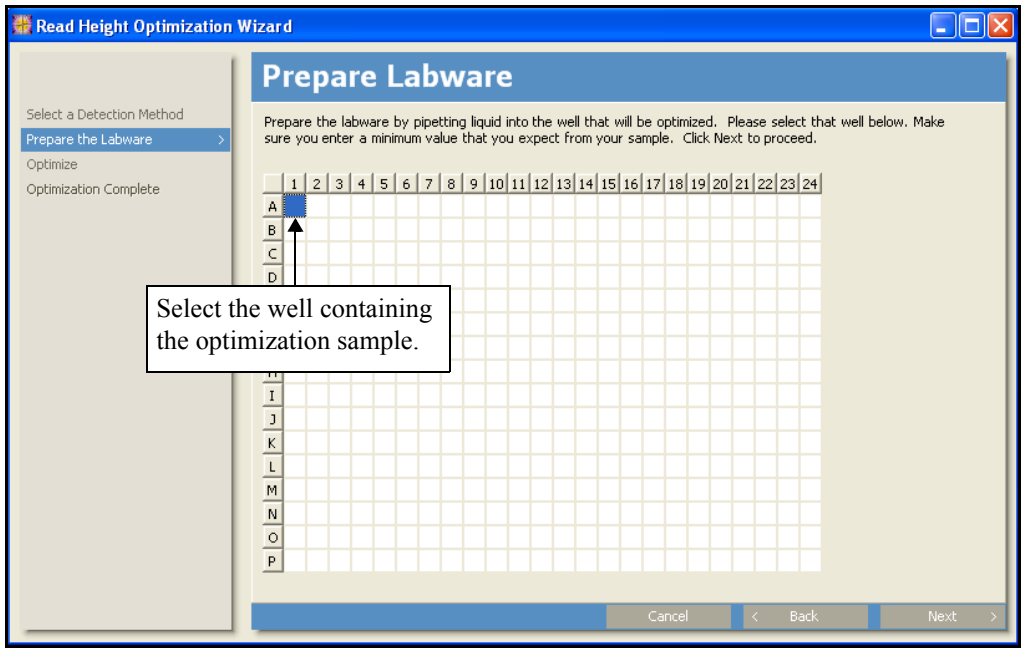


Figure 5-18. Selecting the well read in the read height optimization

6. In Prepare Labware, select the well containing the optimization sample (Figure 5-18).
7. Choose **Next** to start the optimization. Optimization in Progress appears (Figure 5-19). The optimization may take several minutes.

   OR

   Choose **Back** to change the detection method used in the calibration.

   OR

   Choose **Cancel** to close the wizard without performing the optimization.

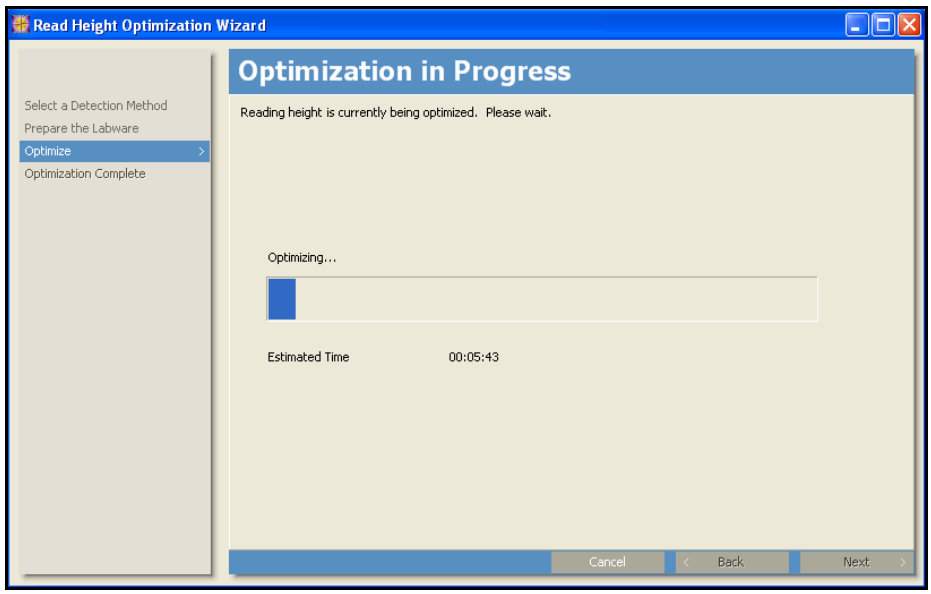


Figure 5-19. Read height optimization in progress

8. While the optimization is in progress, choose **Cancel** to stop the optimization read and close the wizard, if desired. When the read is finished, Optimization Complete appears, displaying the Optimized Read Height (Figure 5-20).

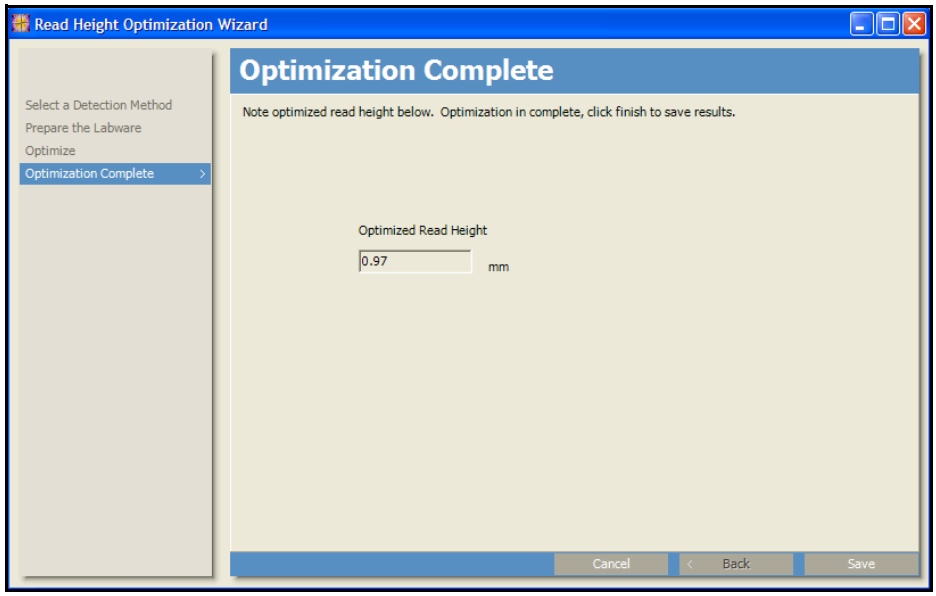


Figure 5-20. Read height optimization completed

9. In Optimization Complete, choose **Save** to save the optimized read height in the protocol. The optimized read height is used for all subsequent runs of the protocol until reset by performing a new optimization or manually selecting a read height option.

   OR

   Choose **Cancel** to close the wizard without saving and using the optimized read height in the protocol run.

5.3.2 Running a Protocol When Simulation Mode is Enabled

Running a protocol in simulation mode allows the protocol configuration to be tested using simulated data before performing the protocol on actual samples. Simulated data is either generated randomly or read from a file.

Simulation mode is automatically enabled when the host computer is not connected to an instrument. When an instrument is connected, simulation mode may be enabled manually in Instruments (refer to Section 2.2.5, *Enabling Simulation Mode*).

To run a protocol in simulation mode:

1. In the Protocol Selection List, select the protocol to run.

2. From the toolbar, choose **Run the selected protocol**.

   OR

   From the Actions menu, choose **Run the selected protocol**.

   OR

   Right-click on the selected protocol and choose **Run the selected protocol** from the menu. Prepare to Run Protocol appears (Figure 5-21).

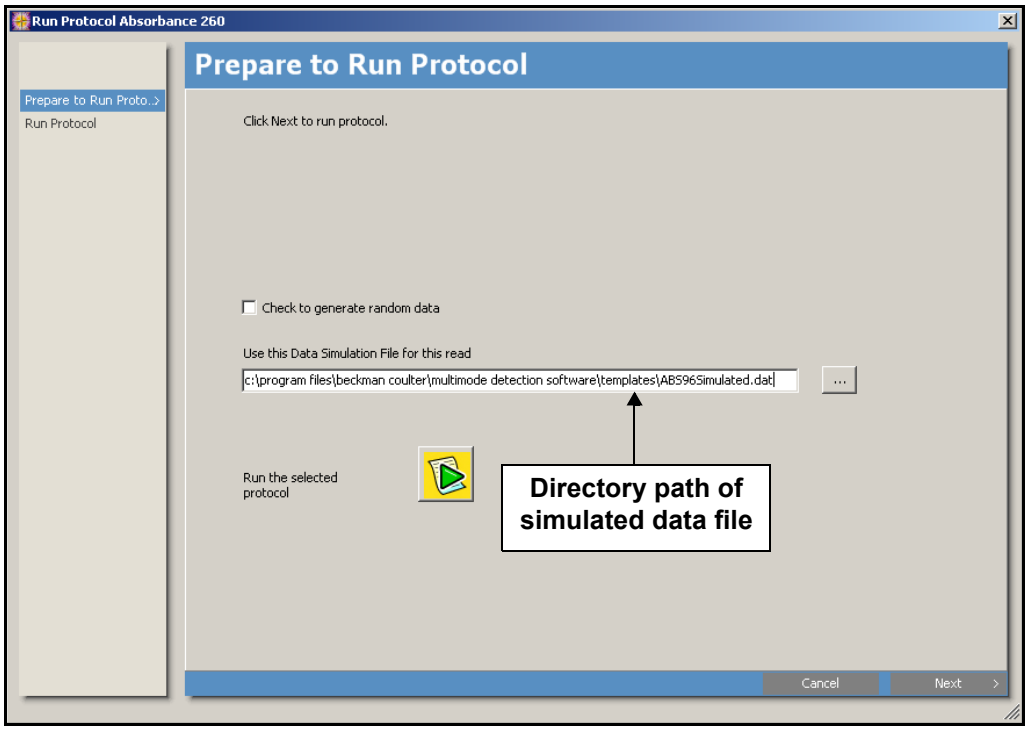


Figure 5-21. Preparing to run a protocol in simulation mode

3. Select **Check to generate random data** to run the protocol using random data generated by the software, if desired.

4. In Use this Data Simulation file for this read:

   - Leave the directory path of the simulated data path as configured to use the data file selected in Software Settings (refer to Section 1.4.2.3, *Choosing Simulated Data Files*).
   - Enter the directory path of the desired data file. The complete directory path must be entered; for example, **c:\program files\Beckman Coulter multimode reader\simulations\simulated absorbance data.dat**.
   - Browse to the location where the desired data file is saved and select it.

Note: Results from prior measurements saved in .dat format may be used as simulated data files (refer to Section 5.2.6, *Configuring Output Settings*). Simulated data files are used when the number of measurement points in the simulated protocol run is the same as those present in the data file. When the number of measurement points is different, the software generates random data.

When a different simulated data file is selected in Prepare to Run Protocol, the file is used for the current simulated run only. After the simulated run has finished, the data file defaults to the file selected in Software Settings (refer to Section 1.4.2.3, *Choosing Simulated Data Files*).

5. Choose **Run** or **Next** to run the selected protocol.

   OR

   Choose **Cancel** to close Run Protocol without running the protocol.

5.4 Editing Protocols

The parameters configured for user-defined protocols may be edited. Default protocols included with the software installation may not be edited.

To edit a protocol:

1. In the Protocol Selection List, select the protocol to edit.

2. From the toolbar, choose **Edit the selected protocol**.

   OR

   From the Actions menu, choose **Edit the selected protocol**.

   OR

   Right-click on the selected protocol and choose **Edit the selected protocol** from the menu.

   OR

   Double-click on the selected protocol. Edit Protocol appears (Figure 5-22).

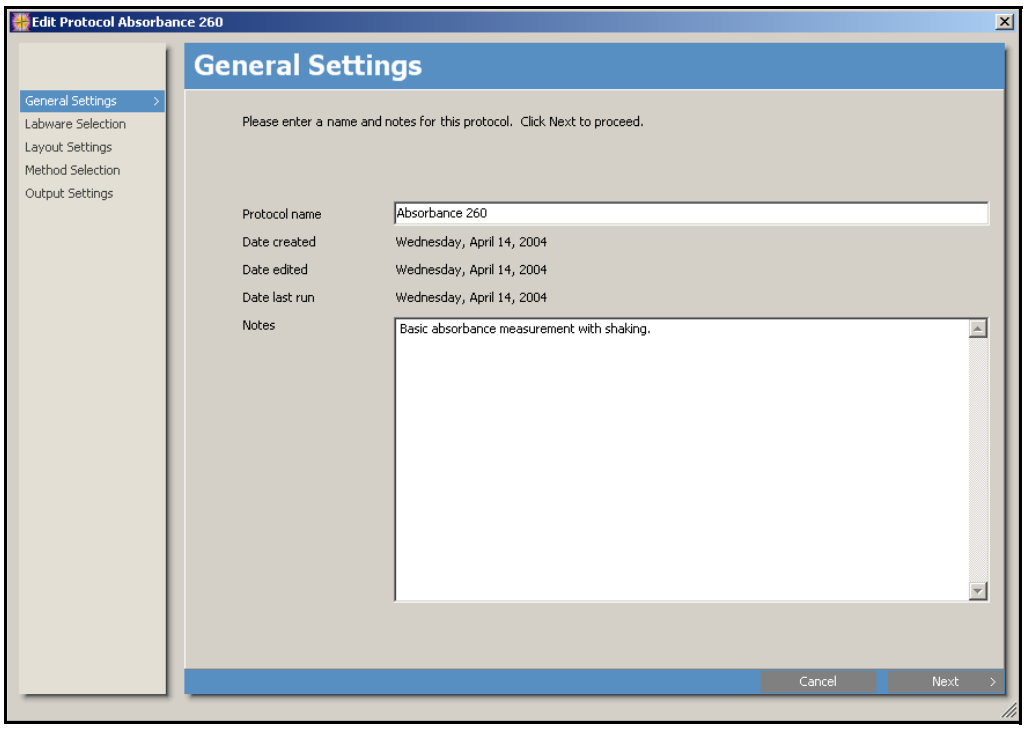


Figure 5-22. Editing an absorbance protocol

3. Edit the parameters in each Edit Protocol screen as desired.

   Note: Refer to Section 5.2, *Creating Protocols*, for detailed information about configuring protocol parameters.

4. Choose **Save** to close Edit Protocol and save the changes.

   OR

   Choose **Cancel** to close Edit Protocol without saving changes.

5.5 Copying Protocols

Copies may be made of existing protocols. After a copy has been created, it may be used as a template for a new protocol using the same technique.

To make a copy of a protocol:

1. In the Protocol Selection List, select the protocol to copy.
2. From the toolbar, choose **Make a copy of the selected protocol**.

OR

From the Actions menu, choose **Make a copy of the selected protocol**.

OR

Right-click on the selected protocol and choose **Make a copy of the selected protocol** from the menu. A copy of the selected protocol appears in the Protocol Selection List.

Note: The default name format for copied protocols is Copy of OriginalName. To change the name, open the protocol for editing and enter the new protocol name (refer to Section 5.4, *Editing Protocols*).

5.6 Deleting Protocols

User-defined protocols that are no longer used to perform measurements may be deleted from the Protocol Selection List. Default protocols included with the software installation may not be deleted.

To delete a user-defined protocol:

1. In the Protocol Selection List, select the protocol to delete.

2. From the toolbar, choose **Delete the selected protocol**.

   OR

   From the Actions menu, choose **Delete the selected protocol**.

   OR

   Right-click on the selected protocol and choose **Delete the selected protocol** from the menu. Message appears (Figure 5-23).

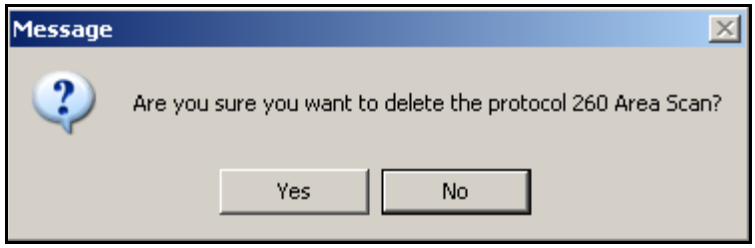


Figure 5-23. Confirming the deletion of a protocol

3. Choose **Yes** to delete the selected protocol.

   OR

   Choose **No** to cancel the deletion.

5.7 Printing Protocol Configuration Information

Information about the protocol configuration and sample layout may be printed separately from measurement results.

To print protocol configuration information:

1. In the Protocol Selection List, select the protocol to print.
2. From the toolbar, choose **Print the selected protocol**.

OR

From the Actions menu, choose **Print the selected protocol**.

OR

Right-click on the selected protocol and choose **Print the selected protocol** from the menu.

Note: Depending on how Print Settings are configured, Print and/or Print Preview may appear before the protocol configuration prints. Refer to Section 1.4.2.2, *Configuring Print Settings*, for more information about enabling and disabling Print and Print Preview.

3. If Print and Print Preview are not enabled in Print Settings, no additional configuration is required. The protocol configuration and layout of samples on the plate prints.

   OR

   If Print appears, configure printing options as desired and choose **OK**.

   OR

   If Print Preview appears, use the toolbar controls to change the magnification, layout view, or page(s) displayed, if desired (Figure 5-24).

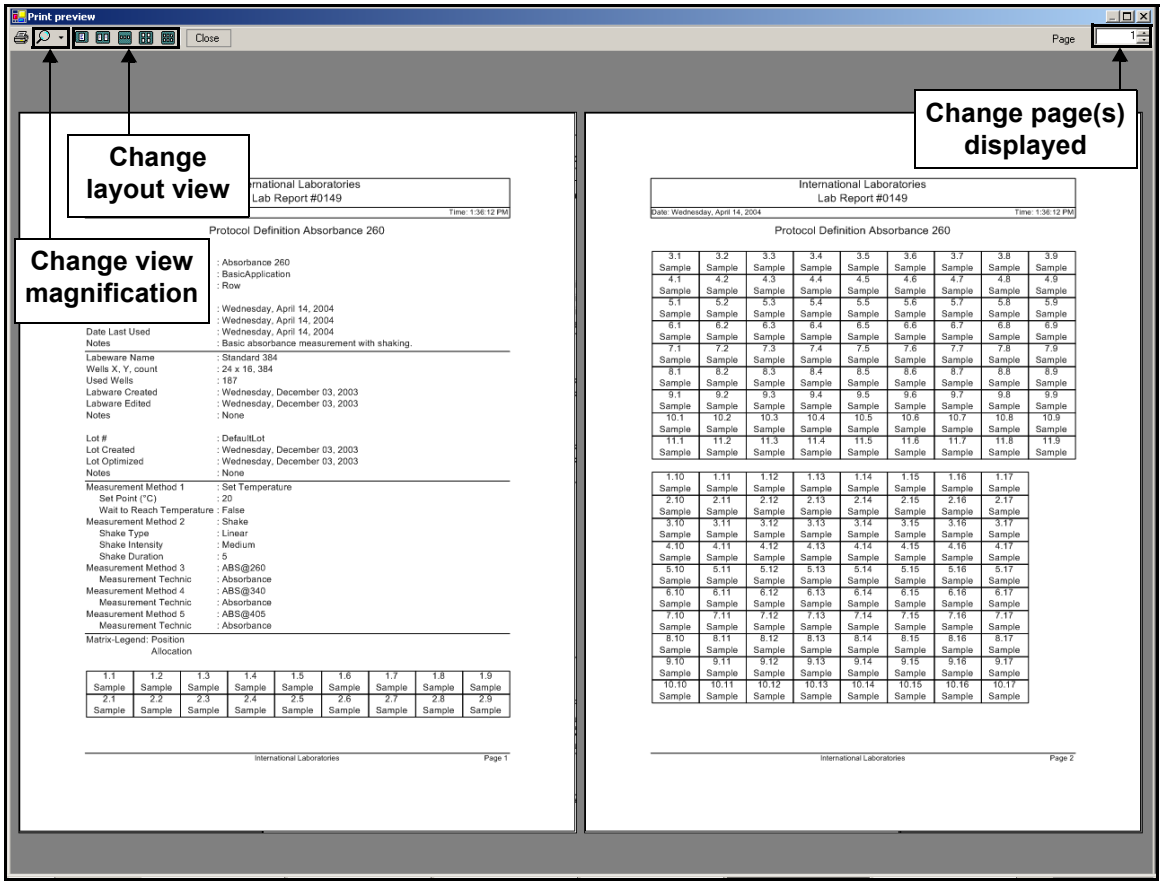


Figure 5-24. Previewing a protocol configuration and sample layout printout

4. In Print Preview, choose **Print** to print out the measurement results.

5. In Print Preview, choose **Close** to close the window.

Note: Choosing Close without first choosing Print cancels the printout.

5.8 Exporting and Importing Protocols

User-defined protocols saved in the Protocol Selection List can be exported to an XML file and imported later to restore that configuration or share it with a copy of Multimode Detection Software installed on another computer. An exported XML file also saves detection method(s) and labware configurations used in the protocol.

Exporting default protocols included with the software installation is not necessary. Since default protocols may not be edited, deleted, or overwritten, they are always available to users at all times. Because of this, importing a default protocol from an XML export file is not permitted by the software.

To export a protocol:

1. In the Protocol Selection List, select the protocol to export.
2. From the File menu, choose **Export>Protocol**. Browse for Folder appears (Figure 5-25).

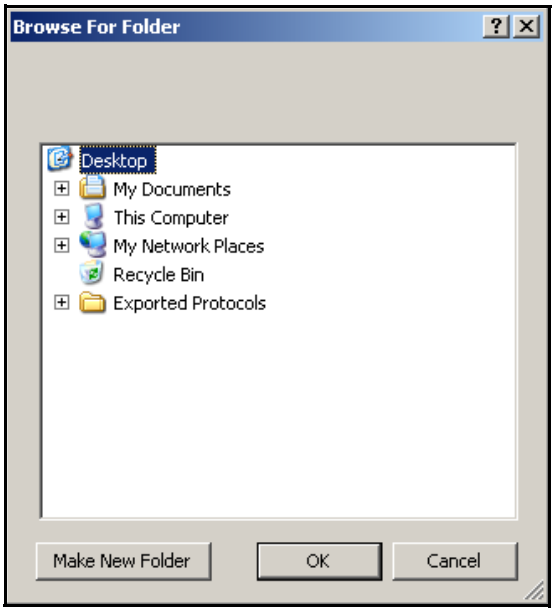


Figure 5-25. Choosing the folder where an exported protocol will be saved

3. In Browse for Folder, browse to the folder where the exported protocol will be saved.

   OR

   Choose **Make New Folder** to create a new folder where the exported protocol will be saved.

4. Choose **OK** to export the protocol. The exported protocol is saved using the default file name format, Protocol_*ProtocolName*.xml. In order to import the file at a later date, the filename must not be changed.

   OR

   Choose **Cancel** to stop the operation without exporting the protocol.

To import a protocol from an exported XML file:

1. From the File menu, choose **Import>Protocol**. Open appears.
2. In Open, browse to and select the desired XML file to import.
3. Choose **Open**. The protocol, as well as detection method(s) and labware used in the protocol, are imported. Any default detection method(s) or labware used in the protocol are not imported because default methods and labware may not be edited, deleted, or overwritten. Instead, the imported protocol uses the same default methods and labware stored in the software.

6 Viewing Measurement Results

6.1 Overview

Multimode Detection Software displays measurement results in a series of tabs within Run Protocol (Figure 6-1). Raw data is displayed immediately after each column or row is read. All measurements display sample status and raw data; kinetic and scan measurements also display graphs for each sample. Detection methods and measurement cycles performed in the protocol are displayed in the left pane of the tab; results are displayed in the right pane. The tabs may be viewed until Finish is chosen.

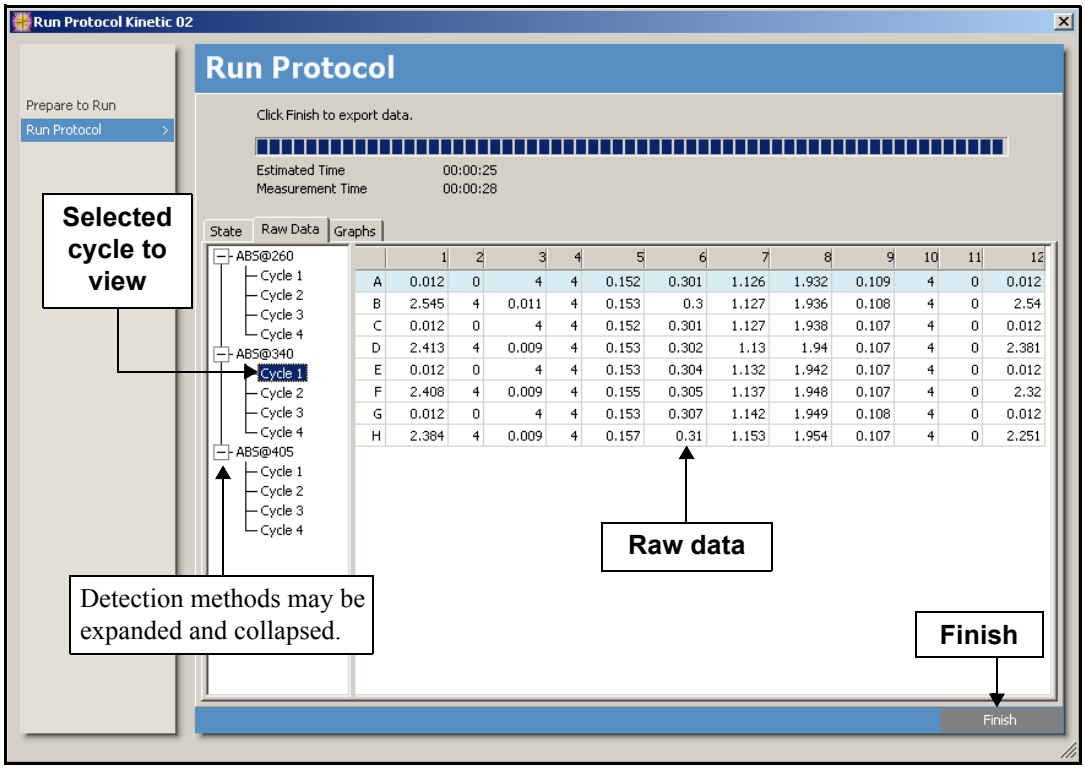


Figure 6-1. Measurement results displayed after a protocol run

When Finish is chosen, Run Protocol closes and results are exported and printed according to the file and printing options defined in the protocol (refer to Section 5.2.6, *Configuring Output Settings*). When no file or printing options are defined, measurement results are deleted and may not be recovered.

Measurement results are exported to the data directory specified in System Settings (refer to Section 1.4.2, *Configuring Software Settings*). Exported results may be viewed, edited, evaluated, and printed using Microsoft® Excel or other applications compatible with the file format of the exported data. Multimode Detection Software does not provide the ability to open, view, or evaluate exported results.

This chapter covers:

- *Viewing Measurement Results in Run Protocol* (Section 6.2).
- *Viewing Exported Measurement Results* (Section 6.3).
- *Printing Measurement Results* (Section 6.4).

6.2 Viewing Measurement Results in Run Protocol

Measurement results are displayed in a series of tabs within Run Protocol:

- State — displays which samples were measured successfully and which were not because of errors during the measurement (refer to Section 6.2.1, *Viewing State*).
- Raw Data — displays actual values measured. For kinetic measurements, results are displayed for each measurement cycle. For linear and area scan measurements, results are displayed for each measurement point (refer to Section 6.2.2, *Viewing Raw Data*).
- Graphs — displays graphs of measurement data for kinetic and scan measurements only (refer to Section 6.2.3, *Viewing Kinetic and Scan Graphs*).

6.2.1 Viewing State

State displays which samples were measured successfully and which were not because of errors during the measurement (Figure 6-2):

- OK — sample measured successfully.
- Error — sample not measured because an instrument error occurred.
- Overflow — no results available because the value measured exceeds the upper measurement limit.
- Incorrect — sample was measured, but is incorrect because an instrument error occurred.
- Unused — sample not selected for measurement in the protocol.

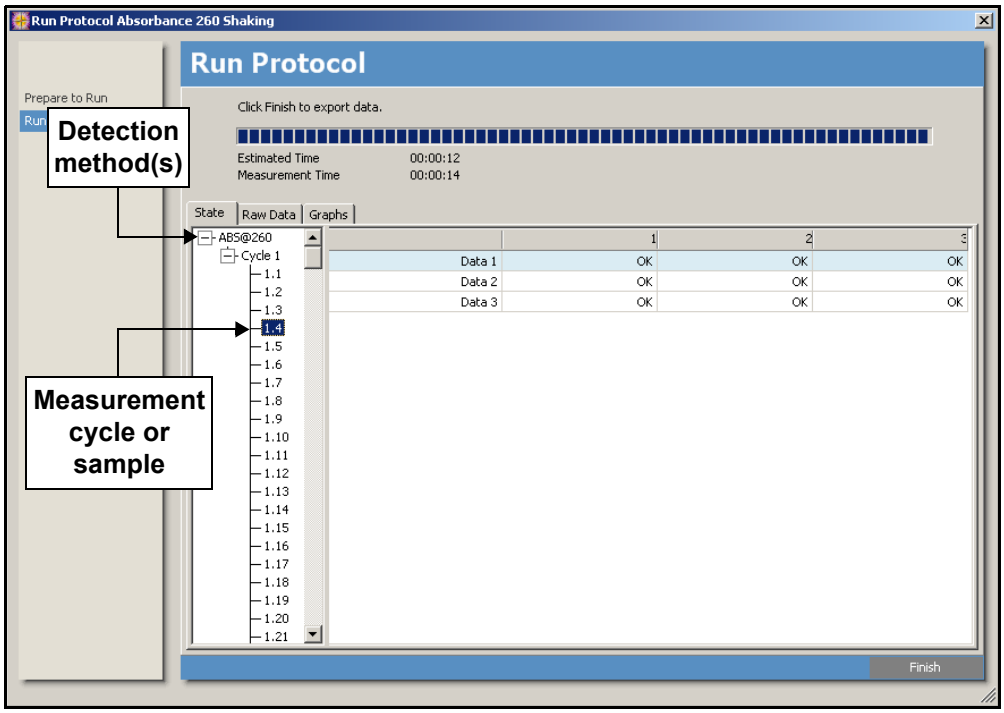


Figure 6-2. State displayed for an area scan measurement

To view State of samples or measurement points:

1. If necessary, in the left panel of the tab, click the **+** next to the desired detection method to expand the view and access measurement cycles (Figure 6-2).
2. For absorbance, luminescence, and fluorescence measurements, select the desired measurement cycle. The status of each sample measured in the cycle appears in the right pane.

   OR

   For linear and area scan measurements, choose the desired sample. The status of each measurement point for the chosen sample appears in the right pane.
3. Choose a different cycle or sample to view.

   OR

   Choose a different results tab to view.

   OR

   Choose **Finish** to close Run Protocol. Measurement results are saved and/or prepared for printing when file and print options are defined in the protocol (refer to Section 5.2.6, *Configuring Output Settings*).

6.2.2 Viewing Raw Data

Raw Data displays the results for each sample or point measured in every cycle (Figure 6-3).

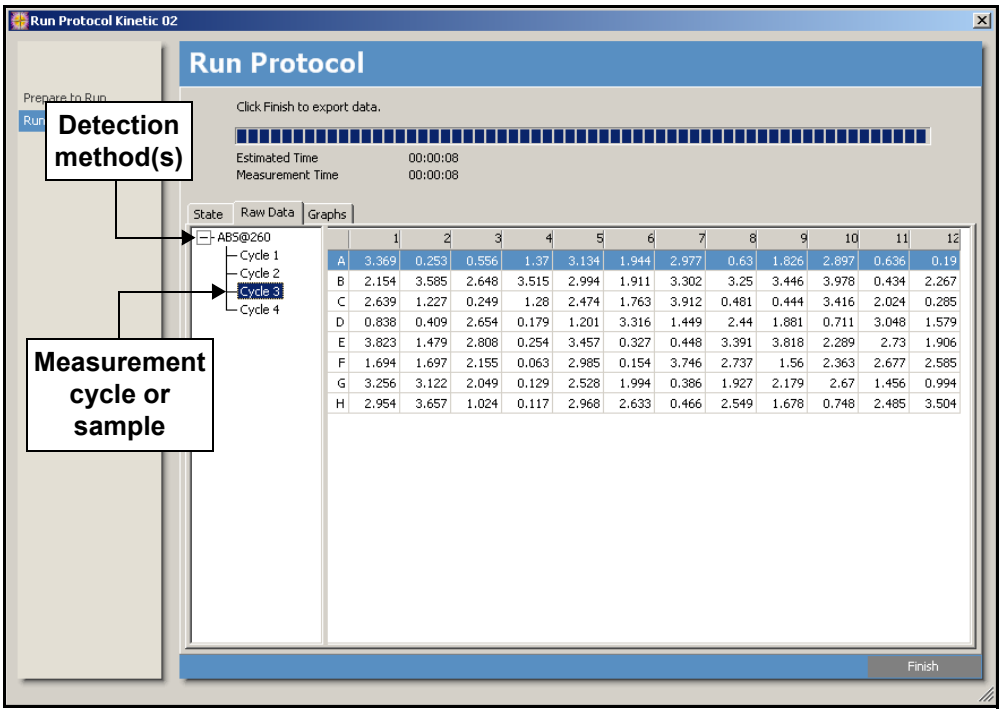


Figure 6-3. Raw Data displayed for a kinetic absorbance measurement

To view Raw Data of samples or measurement points:

1. If necessary, in the left panel of the tab, click the **+** next to the desired detection method to expand the view and access measurement cycles (Figure 6-3).
2. For absorbance, luminescence, and fluorescence measurements, select the desired measurement cycle. Raw data measured for each sample in the cycle appears in the right pane.

   OR

   For linear and area scan measurements, choose the desired sample. Raw data measured at each point within the chosen sample appears in the right pane.
3. If desired, right-click on the raw data results and choose **Copy** to copy the raw data values to the clipboard. The values may then be pasted into another document or file.
4. Choose a different cycle or sample to view.

   OR

   Choose a different results tab to view.

   OR

   Choose **Finish** to close Run Protocol. Measurement results are saved and/or prepared for printing when file and print options are defined in the protocol (refer to Section 5.2.6, *Configuring Output Settings*).

6.2.3 Viewing Kinetic and Scan Graphs

Graphs displays graphs for each sample measured in kinetic, area scan, and linear scan protocols (Figure 6-4). The tab is not displayed unless a kinetic or scan measurement is performed.

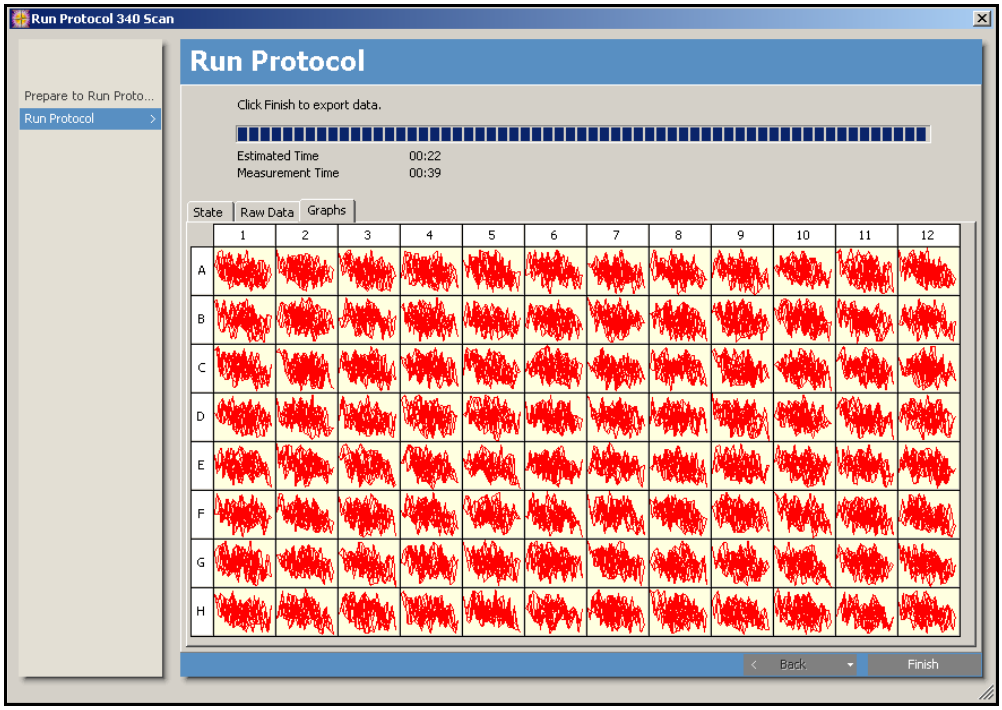


Figure 6-4. Viewing area scan measurement graphs

Graphs for individual samples can be viewed in detail. To view a detailed graph of an individual sample:

Click on the desired sample to view. Graphic View appears (Figure 6-5). Refer to Section 6.2.3.1, *Viewing Detailed Graphs of Individual Samples*, for more information about viewing and printing detailed graphs.

To exit Graphs:

Choose a different results tab to view.

OR

Choose **Finish** to close Run Protocol. Measurement results are saved and/or prepared for printing when file and print options are defined in the protocol (refer to Section 5.2.6, *Configuring Output Settings*).

6.2.3.1 Viewing Detailed Graphs of Individual Samples

Graphic View displays a detailed graph of the measurement results for an individual sample. Two types of graphs are available:

- Area scan measurements display a three-dimensional graph that may be rotated horizontally and vertically (refer to Section 6.2.3.1.1, *Viewing and Printing an Area Scan Graph*).
- Kinetic and linear scan measurements display a two-dimensional graph that may be zoomed (refer to Section 6.2.3.1.2, *Viewing and Printing a Kinetic or Linear Scan Graph*).

6.2.3.1.1 Viewing and Printing an Area Scan Graph

Area scan measurements display a three-dimensional graph with the optical density (OD) of all points measured color coded (Figure 6-5). The graph may be rotated horizontally and vertically and printed.

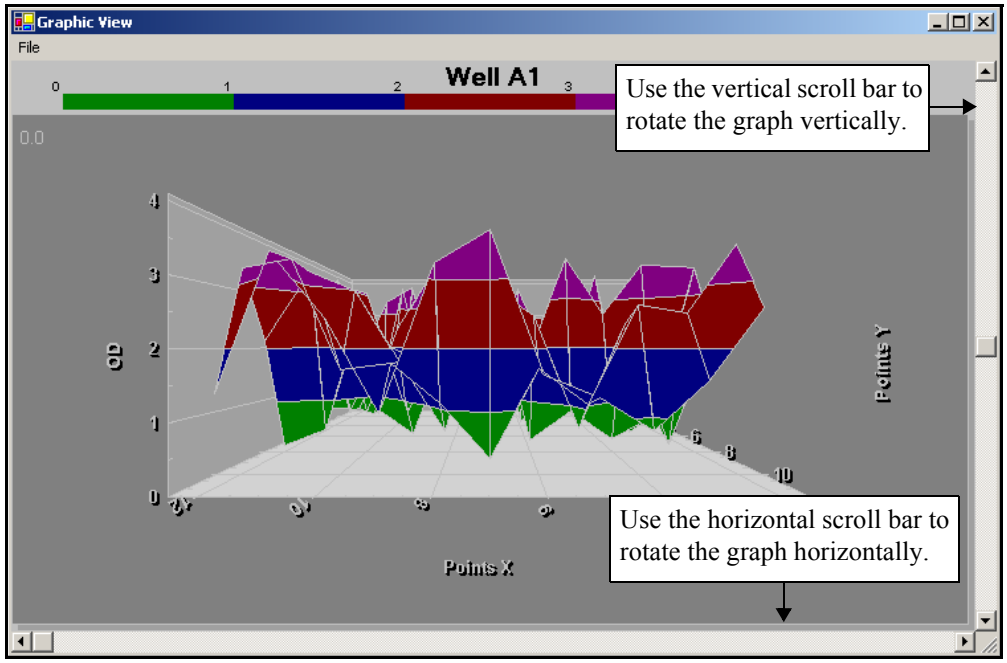


Figure 6-5. Three-dimensional area scan graph

To rotate the graph:

- Use the vertical scroll bar to rotate the graph vertically.
- Use the horizontal scroll bar to rotate the graph horizontally.
- Double-click on the graph to start and stop a continuous animated horizontal rotation of the graph.

To print the graph:

1. From the File menu, choose **Print**. Print appears.
2. Configure printing options as desired, and choose OK to print the graph.

To close Graphic View:

From the File menu, choose **Close**.

6.2.3.1.2 Viewing and Printing a Kinetic or Linear Scan Graph

Kinetic and linear scan measurements display a two-dimensional graph tracking the optical density (OD) measured in each cycle or at each point (Figure 6-6). Two-dimensional graphs support zooming in on selected regions and printing.

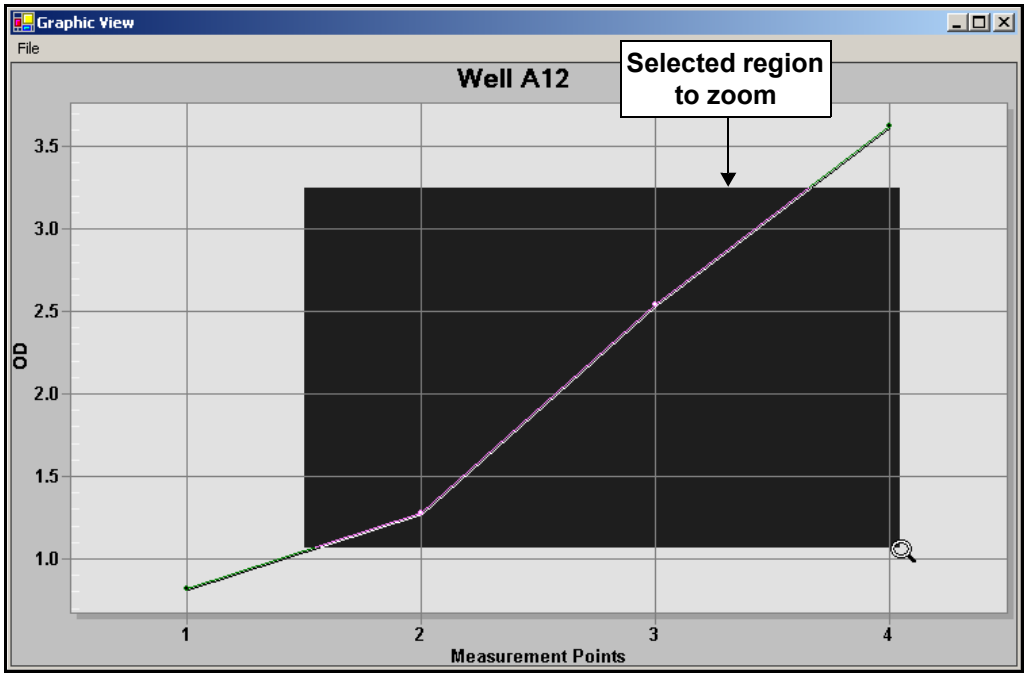


Figure 6-6. Two-dimensional kinetic graph with zoom region selected

To zoom in on a region:

1. Position the cursor at the desired starting point for the region, then click and hold the mouse button down.
2. Drag the mouse until the desired region is selected (Figure 6-6). The selected region is highlighted in black.
3. Release the mouse button. Graphic View displays the zoomed view of the selected region.

To print the graph:

1. From the File menu, choose **Print**. Print appears.
2. Configure printing options as desired, and choose **OK** to print the graph.

To close Graphic View:

From the File menu, choose **Close**.

6.3 Viewing Exported Measurement Results

When a protocol has export and/or data file options configured in Output Settings, measurement results are exported after completing a protocol run. Exported results are always opened, viewed, and evaluated in a compatible software application, such as Microsoft® Excel. Multimode Detection Software only provides the ability to view results before they are exported.

Two file export and save options are available:

- Export to Microsoft Excel — saves results in a format compatible with Microsoft Excel, and automatically opens Excel when the protocol run is completed. Refer to Section 6.3.1, *Viewing Measurement Results Using Microsoft® Excel*, for more information about viewing exported results in Excel.

  Note: Multimode Detection Software automatically determines the appropriate export method based on the version of Microsoft Office installed on the host computer. XML (.xml) files are exported when Office XP is installed. When Office 2000 installed, the measurement results are copied into a new spreadsheet which must be saved in Excel. Versions of Excel prior to Office 2000 are not supported by the Export to Microsoft Excel function, but can open measurement results stored in tab-delimited data (.dat) files.

- Create .XML and .dat data files — saves results in .xml and tab-delimited data (.dat) files, which may be opened by compatible software applications.

Note: Refer to Section 5.2.6, *Configuring Output Settings*, for more information about configuring export and file options.

To view saved measurement results:

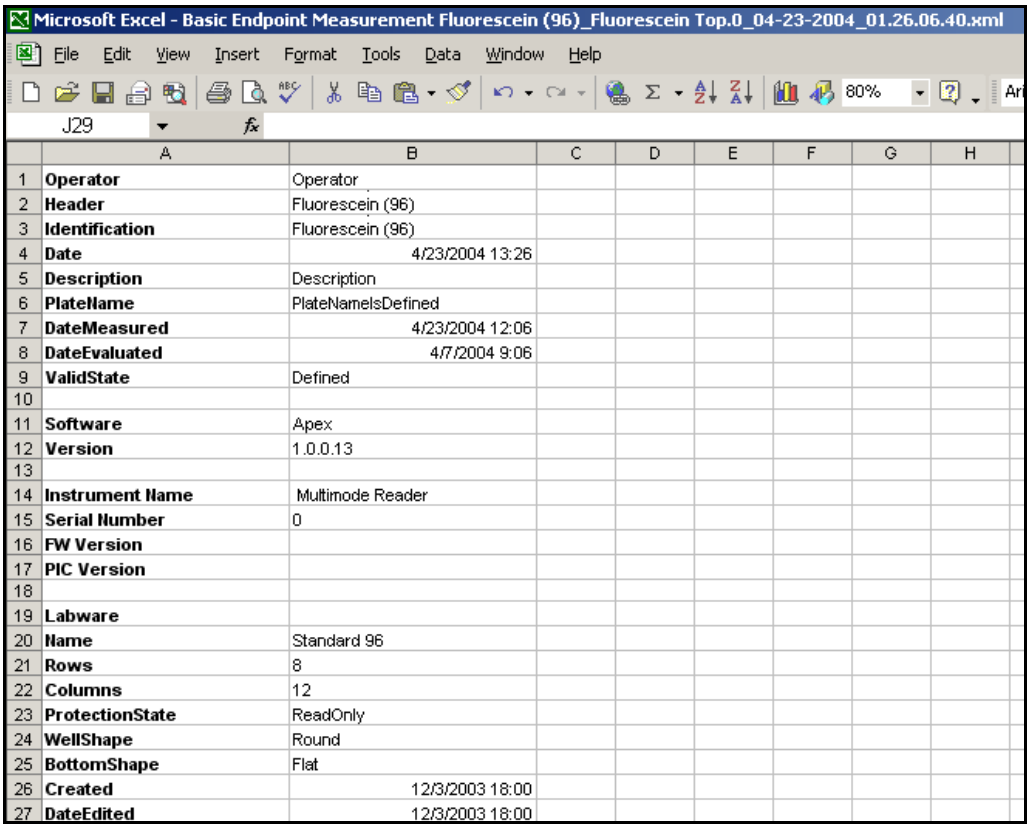
1. Open the desired software application.
2. Browse to the directory where exported measurement results are stored, and open the desired file. Saved measurement results are stored in the data directory selected in Software Settings (refer to Section 1.4.2.4, *Choosing the Directory Where Measurement Results are Saved*).

6.3.1 Viewing Measurement Results Using Microsoft® Excel

When Export to Microsoft Excel is defined in the protocol, protocol information and measurement results are exported to a file and automatically opened in Excel after completing a protocol run. Each exported Excel spreadsheet contains data for a single protocol run. Data is presented in a series of worksheets. The first worksheet, General, contains information about the protocol (refer to Section 6.3.1.1, *Viewing Information About the Protocol*). Raw data is contained in separate worksheets for each measurement cycle (refer to Section 6.3.1.2, *Viewing Measurement Results*).

6.3.1.1 Viewing Information About the Protocol

General, the first worksheet in all Excel files exported by Multimode Detection Software, contains information about parameters configured in the protocol and the protocol run itself (Figure 6-7).



	A	B	C	D	E	F	G	H
1	**Operator**	Operator						
2	**Header**	Fluorescein (96)						
3	**Identification**	Fluorescein (96)						
4	**Date**	4/23/2004 13:26						
5	**Description**	Description						
6	**PlateName**	PlateNamesDefined						
7	**DateMeasured**	4/23/2004 12:06						
8	**DateEvaluated**	4/7/2004 9:06						
9	**ValidState**	Defined						
10								
11	**Software**	Apex						
12	**Version**	1.0.0.13						
13								
14	**Instrument Name**	Multimode Reader						
15	**Serial Number**	0						
16	**FW Version**							
17	**PIC Version**							
18								
19	**Labware**							
20	**Name**	Standard 96						
21	**Rows**	8						
22	**Columns**	12						
23	**ProtectionState**	ReadOnly						
24	**WellShape**	Round						
25	**BottomShape**	Flat						
26	**Created**	12/3/2003 18:00						
27	**DateEdited**	12/3/2003 18:00						

Figure 6-7. Viewing protocol information in the General worksheet (excerpt)

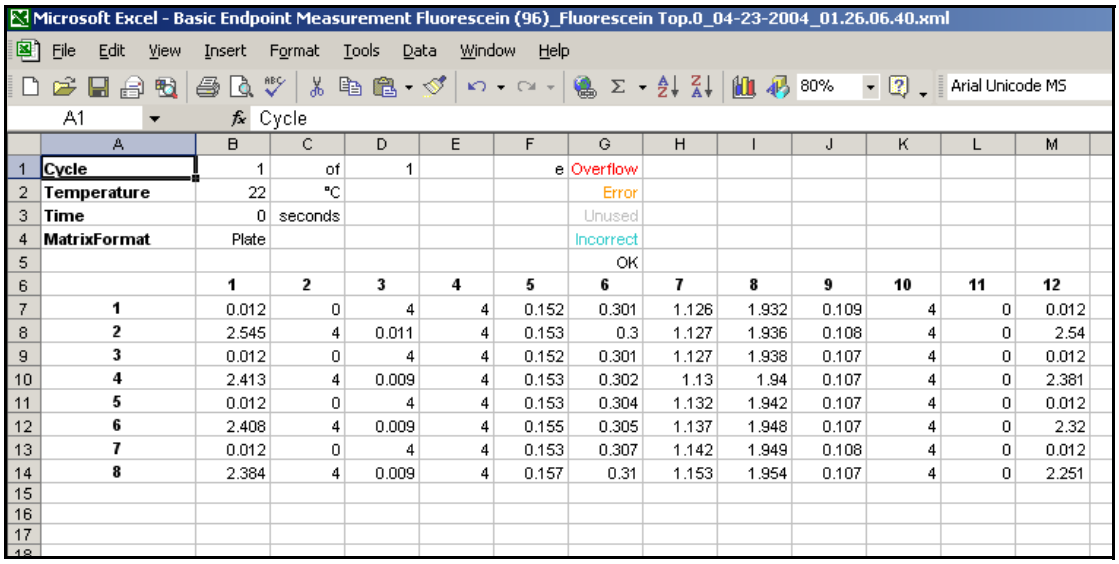
To view the General worksheet:

In Excel, choose the **General** tab in the lower left corner of the window.

6.3.1.2 Viewing Measurement Results

Measurement results exported to Excel display raw data in matrices that represent the layout of measured wells on the plate (Figure 6-8). When multiple measurement cycles were performed in a protocol run, data from each cycle is presented in a separate worksheet.

- ***DTX 880*** — Fluorescence polarization measurement results contain raw data and calculated mP (refer to Section 6.3.1.2.1, *Viewing Fluorescence Polarization Results (880 only)*).



Microsoft Excel - Basic Endpoint Measurement Fluorescein (96)_Fluorescein Top.0_04-23-2004_01.26.06.40.xml

File Edit View Insert Format Tools Data Window Help

80% Arial Unicode MS

A1 fx Cycle

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	**Cycle**	1	of	1		e	Overflow						
2	**Temperature**	22	°C				Error						
3	**Time**	0	seconds				Unused						
4	**MatrixFormat**	Plate					Incorrect						
5							OK						
6		**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**
7	**1**	0.012	0	4	4	0.152	0.301	1.126	1.932	0.109	4	0	0.012
8	**2**	2.545	4	0.011	4	0.153	0.3	1.127	1.936	0.108	4	0	2.54
9	**3**	0.012	0	4	4	0.152	0.301	1.127	1.938	0.107	4	0	0.012
10	**4**	2.413	4	0.009	4	0.153	0.302	1.13	1.94	0.107	4	0	2.381
11	**5**	0.012	0	4	4	0.153	0.304	1.132	1.942	0.107	4	0	0.012
12	**6**	2.408	4	0.009	4	0.155	0.305	1.137	1.948	0.107	4	0	2.32
13	**7**	0.012	0	4	4	0.153	0.307	1.142	1.949	0.108	4	0	0.012
14	**8**	2.384	4	0.009	4	0.157	0.31	1.153	1.954	0.107	4	0	2.251
15													
16													
17													

Figure 6-8. Viewing absorbance data in a Cycle worksheet (excerpt)

To view results for a measurement cycle:

In Excel, choose the **Cycle** tab for the desired measurement cycle.

6.3.1.2.1 Viewing Fluorescence Polarization Results (880 only)

Fluorescence polarization measurement results exported to Excel contain raw data measured at both the parallel and perpendicular polarization planes and calculate an approximate mP for each sample (Figure 6-9). Data is laid out in matrices that represent the wells on the plate. When multiple measurement cycles were performed in a protocol run, data from each cycle is presented in a separate worksheet.

The mP calculated for each sample is considered approximate because:

- Blank samples are not defined at run time, but are required to increase the accuracy of the results.
- The G-factor used in the calculation, which factors out differences in detection efficiency between the polarization planes, is derived from fluorescein measurements performed on a number of DTX 880 instruments.

Note: Minor deviations between the actual G-Factor present in the measurement and the average G-factor used in the calculation result in a common shift for all mP values measured. Differences in mP between samples and controls are not affected to a first order approximation. If a more accurate G-factor has been determined for the instrument, it may be entered in the G-factor cell in the spreadsheet.

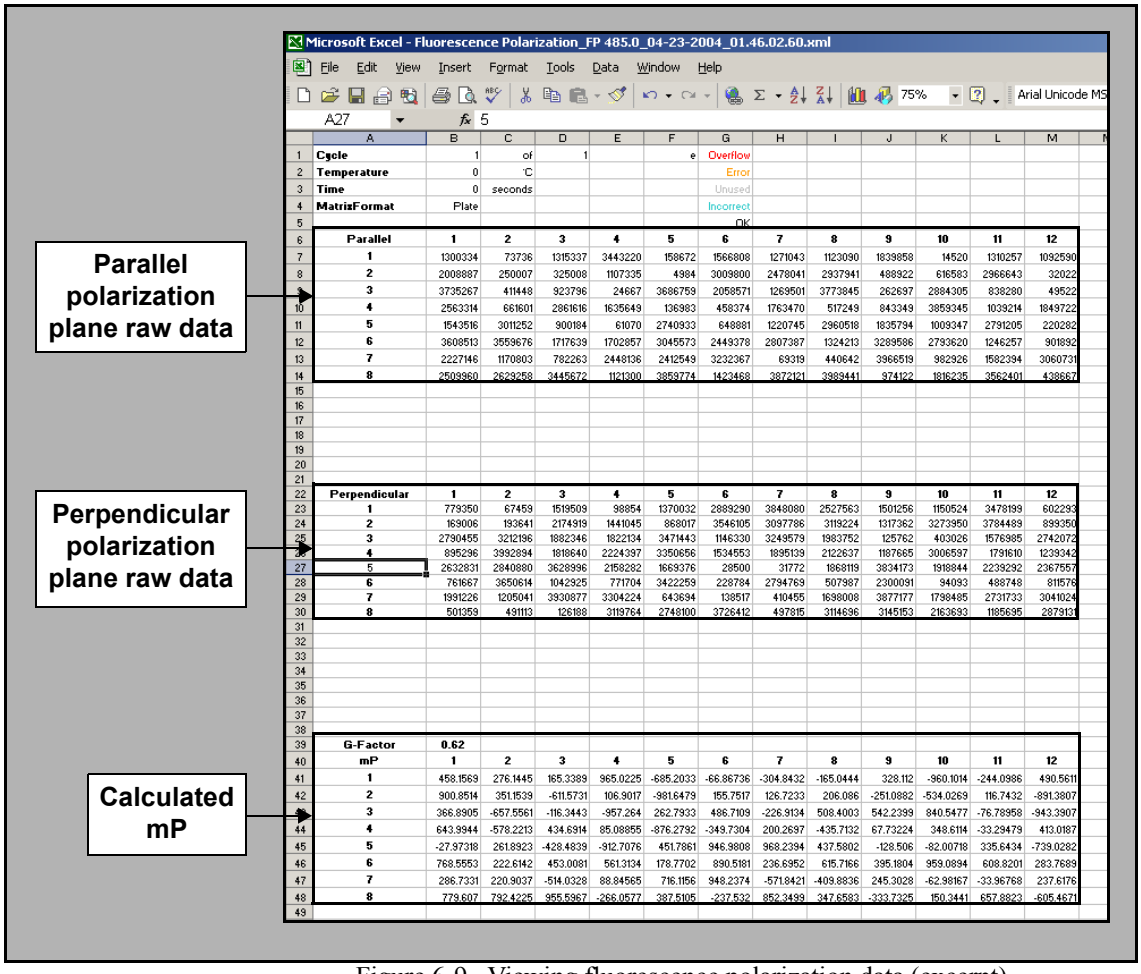


Figure 6-9. Viewing fluorescence polarization data (excerpt)

6.4 Printing Measurement Results

When a protocol has Print options configured in Output Settings, the measurement results are available to print after completing a protocol run. Refer to Section 5.2.6, *Configuring Output Settings*, for information about the printing options available.

Note: Depending on how Print Settings are configured, Print and/or Print Preview may appear before the measurement results print. Refer to Section 1.4.2.2, *Configuring Print Settings* for information about enabling and disabling Print and Print Preview.

To print measurement results:

1. When the measurement is complete, press **Finish**. If Print and Print Preview are not enabled in Print Settings, no additional configuration is required. The measurement results print out.
2. If Print appears, configure printing options as desired and choose **OK**.

   OR

   If Print Preview appears, use the toolbar controls to change the magnification, layout view, or page(s) displayed, if desired (Figure 6-10)

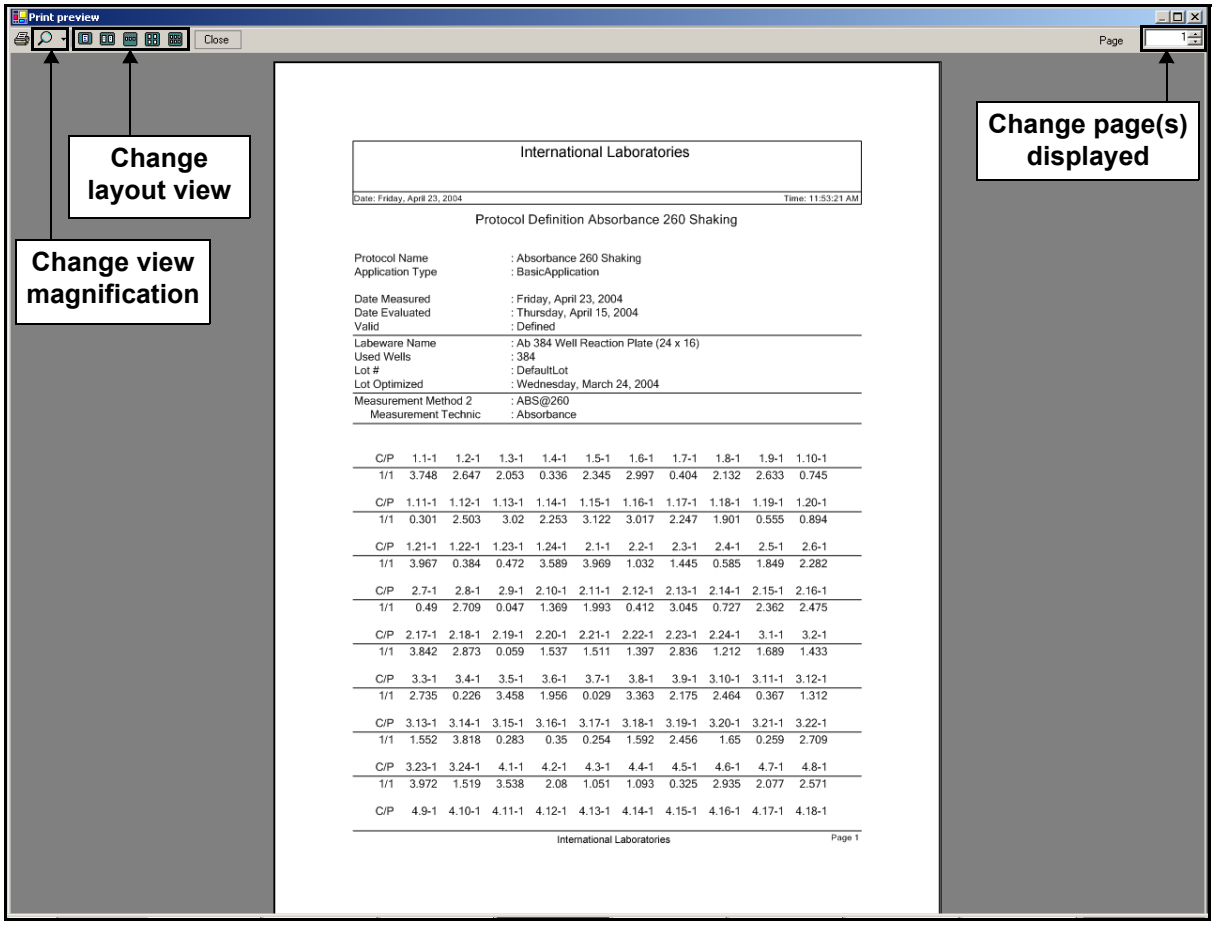


Figure 6-10. Previewing a measurement results printout

3. In Print Preview, choose **Print** to print out the measurement results.

4. In Print Preview, choose **Close** to close the window.

Note: Choosing Close without first choosing Print cancels the printout.

A Appendix A: Software Error Codes

A.1 Troubleshooting Software Error Codes

Table A-1 lists software error codes, error messages, and recommended solutions for resolving errors.

Table A-1. Software Error Codes and Solutions

Error Code	Error Message	Recommended Solution
1	Commanded transport motion exceeds actual physical travel limits.	Check plate parameters configured in Labware Information (refer to Section 4.2.1, *Defining Labware Information*).
3	Auto calibration ramp parameter exceeds valid range.	Make sure the microplate carrier movement is not obstructed and run the measurement again.
4	Transport is not operational or the transport sensor is defective.	Make sure the transport lock has been removed from the microplate carrier.
5	Transport lost more than 2 full steps.	Make sure the microplate carrier movement is not obstructed.
6	Calibration positioning failed.	N/A
7	Offset calculated in auto calibration is out of expected range. Possible causes: ADC values are too low, lamp is not adjusted, incorrect EEPROM parameters.	Check filter parameters configured in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).
8	Offset calculated in auto calibration is out of expected range. Possible causes: ADC values are too low, lamp is not adjusted, incorrect EEPROM parameters.	Contact a Beckman Coulter Service Engineer.
9	Z-transport adjust error. Possible causes: wrong well for adjustment, low fluorescein concentration.	In the Read Height Optimization Wizard, confirm that the correct well is selected for the optimization read (refer to Section 5.3.1.1, *Optimizing Read Height (880 only)*).
10	Measurement attempted with no labware loaded.	Load a plate into the instrument.

Table A-1. Software Error Codes and Solutions

Error Code	Error Message	Recommended Solution
11	The current labware definition is invalid.	Check plate parameters configured in Labware Information (refer to Section 4.2.1, *Defining Labware Information*).
12	Detection method parameters are invalid.	Check the configuration of detection method parameters.
13	The current labware format is not supported.	Make sure the plate type has been defined (refer to Section 4.2, *Creating Labware*).
14	Currently loaded labware is not in the correct orientation.	Load the plate into the instrument with the orientation configured in the protocol.
16	Linear scan range exceeds well diameter.	Reduce the number of measurement points in the scan configuration (refer to Section 5.2.5.1, *Configuring a Method in Basic Mode*).
21	Bright measurement below limit. Check Platedefinition.	Check plate parameters configured in Labware Information (refer to Section 4.2.1, *Defining Labware Information*).
41	Filter wavelength or filter type in current detection method is not installed.	Check filter parameters configured in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).
42	Filter wavelength in command is not defined in any Slide.	Check filter parameters configured in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).
44	Currently inserted slide code does not match set slide code. Reinsert slide.	Make sure the correct filter slides are installed in the instrument.
46	No filter slide detected.	Make sure filter slides are installed.
47	Currently inserted filter slide code is invalid.	Make sure the correct filter slides are installed in the instrument.
48	Currently inserted filter slide contains no valid defined filters.	Make sure the correct filter slides are installed in the instrument and check filter parameters configured in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).
51	Temperature setpoint exceeds 45ºC/113ºF.	Check the temperature set in Manual Control and/or the protocol. Set a desired temperature less than 45ºC, if necessary (refer to Section 2.3.3.1, *Temperature Control (880 only)*).
60	Currently inserted LED is not defined.	Contact a Beckman Coulter Service Engineer.
61	LED causing ADC overflow.	Make sure the correct filter slides are installed in the instrument and check filter parameters configured in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).

Table A-1. Software Error Codes and Solutions

Error Code	Error Message	Recommended Solution
62	LED energy too low.	Make sure the correct filter slides are installed in the instrument and check filter parameters configured in Instrument Settings (refer to Section 2.3.2, *Defining and Editing Filter Slides*).
63	No LED for selected filter installed.	Contact a Beckman Coulter Service Engineer.
64	ADC gain cannot be maintained	Contact a Beckman Coulter Service Engineer.
65	LED gain setting failed.	Contact a Beckman Coulter Service Engineer.
66	ADC gain fluctuated during measurement, correction of PMT counts is not possible. Try measurement again.	Run the measurement again.
67	Reference ADC too high for valid luminescence measurement.	Make sure the filter compartment and microplate chamber doors are closed.
68	PMT dark counts exceed valid range for measurement. Check threshold or ensure that flaps are closed.	Make sure the filter compartment and microplate chamber doors are closed.
69	Reference ADC LED too low, retry measurement.	Run the measurement again.
140	Command canceled. Aborted by user.	N/A
141	Measurement time exceeds limit for current measurement method.	Use a shorter measurement time in the detection method configuration.
142	Current measurement incomplete, wrong command received.	Check the parameters configured in the protocol.
144	Instrument is busy.	Wait until the amber LED on the instrument is off. If instrument is still busy when the LED is off, turn the instrument off and on.
400	Unknown PIC error.	Contact a Beckman Coulter Service Engineer.
401	Communication timeout with PIC controller.	Contact a Beckman Coulter Service Engineer.
403	PIC timeout during a measurement or initialize.	Contact a Beckman Coulter Service Engineer.
410	Number of steps are zero.	Check the parameters configured in the protocol.
411	LED transport commanded to move to a position less than 0.	Contact a Beckman Coulter Service Engineer.
412	Shutter transport commanded to move to a position less than 0.	Contact a Beckman Coulter Service Engineer.
413	Z transport commanded to move to a position less than 0.	Contact a Beckman Coulter Service Engineer.
415	LED transport commanded to move to a position greater than maximum travel.	Contact a Beckman Coulter Service Engineer.
416	Shutter transport commanded to move to a position greater than maximum travel.	Contact a Beckman Coulter Service Engineer.

Table A-1. Software Error Codes and Solutions

Error Code	Error Message	Recommended Solution
417	Z transport commanded to move to a position greater than maximum travel.	Contact a Beckman Coulter Service Engineer.
419	LED transport initialization failed.	Contact a Beckman Coulter Service Engineer.
420	Shutter transport initialization failed	Contact a Beckman Coulter Service Engineer.
421	Z transport initialization failed	Contact a Beckman Coulter Service Engineer.
423	LED transport lost steps.	Contact a Beckman Coulter Service Engineer.
424	Shutter transport lost steps.	Contact a Beckman Coulter Service Engineer.
425	Z transport lost steps.	Contact a Beckman Coulter Service Engineer.
427	LED transport is not energized	Initialize the instrument (refer to Section 2.2.3, *Initializing the Instrument*).
428	Shutter transport is not energized	Initialize the instrument (refer to Section 2.2.3, *Initializing the Instrument*).
429	Z transport is not energized	Initialize the instrument (refer to Section 2.2.3, *Initializing the Instrument*).
431	LED transport error while positioning.	Contact a Beckman Coulter Service Engineer.
432	Shutter transport error while positioning.	Contact a Beckman Coulter Service Engineer.
433	Z transport error while positioning.	Contact a Beckman Coulter Service Engineer.
440	LED number is out of valid range (0-15).	Contact a Beckman Coulter Service Engineer.
441	LED not initialized	Contact a Beckman Coulter Service Engineer.
442	The LED is not powered.	Contact a Beckman Coulter Service Engineer.
443	The LED has exceeded maximum allowed power consumption.	Contact a Beckman Coulter Service Engineer.
444	LED power is low.	Contact a Beckman Coulter Service Engineer.
445	LED power overflow.	Contact a Beckman Coulter Service Engineer.
446	No LED installed.	Contact a Beckman Coulter Service Engineer.
450	No temperature installed.	Check instrument configuration.
451	Temperature setpoint exceeds maximum valid setpoint.	Decrease the temperature setting (refer to Section 2.3.3.1, *Temperature Control (880 only)*).
452	Temperature sensor is not responding.	Check the current temperature setting (refer to Section 2.3.3.1, *Temperature Control (880 only)*).
453	Temperature sensor not found	Decrease the temperature setting (refer to Section 2.3.3.1, *Temperature Control (880 only)*).

Index