User Guide

FTB-5240S/5240BP

Optical Spectrum Analyzer for FTB-500

www.EXFO.com
Telecom Test and Measurement

EXFO
EXPERTISE REACHING OUT

Information provided by EXFO is believed to be accurate and reliable. However, no responsibility is assumed by EXFO for its use nor for any infringements of patents or other rights of third parties that may result from its use. No license is granted by implication or otherwise under any patent rights of EXFO.

EXFO's Commerce And Government Entities (CAGE) code under the North Atlantic Treaty Organization (NATO) is 0L8C3.

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

Trademarks

EXFO's trademarks have been identified as such. However, the presence or absence of such identification does not affect the legal status of any trademark.

Units of Measurement

Units of measurement in this publication conform to SI standards and practices.

Patents

The Optical Spectrum Analyzer, including its proprietary measurement software, is protected by US patent 6,636,306 (and equivalents in several other countries), and pending PCT international application PCT/CA2008/000647; EXFO's Universal Interface is protected by US patent 6,612,750.

Version number 5.0.0

Contents

Certification Information

F.C.C. Information

Electronic test equipment is exempt from Part 15 compliance (FCC) in the United States. However, compliance verification tests are systematically performed on most EXFO equipment.

CE Information

Electronic test equipment is subject to the EMC Directive in the European Union. The EN61326 standard prescribes both emission and immunity requirements for laboratory, measurement, and control equipment.
This unit has undergone extensive testing according to the European Union Directive and Standards.

EXFO CE **DECLARATION OF CONFORMITY**

Application of Council Directive(s):	2006/95/EC - The Low Voltage Directive 2004/108/EC - The EMC Directive 93/68/EEC - CE Marking And their amendments
Manufacturer's Name:	EXFO Electro-Optical Engineering Inc.
Manufacturer's Address:	400 Godin Avenue Quebec, Quebec Canada, G1M 2K2 (418) 683-0211
Equipment Type/Environment:	Test & Measurement / Industrial
Trade Name/Model No.:	FTB-5230/5240/5240B/5240BP/5240S Optical Spectrum Analyzer

Standard(s) to which Conformity is Declared:

EN 61010-1:2001 Edition 2.0	**Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements.**
EN 61326-1:2006	**Electrical Equipment for Measurement, Control and Laboratory Use - EMC Requirements**
EN 60825-1:2007 Edition 2.0	**Safety of laser products – Part 1: Equipment classification and requirements**
EN 55022: 2006 + A1: 2007	**Information technology equipment — Radio disturbance characteristics — Limits and methods of measurement**

I, the undersigned, hereby declare that the equipment specified above conforms to the above Directive and Standards.

Manufacturer
Signature:

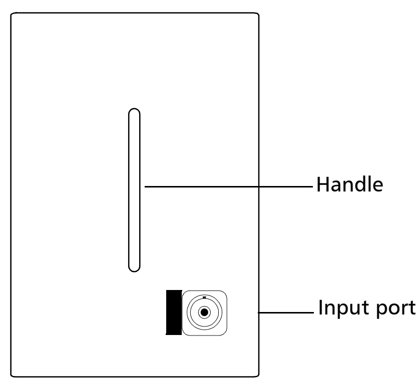
Full Name: Stephen Bull, E. Eng
Position: Vice-President Research and Development
Address: 400 Godin Avenue, Quebec (Quebec), Canada, G1M 2K2
Date: January 09, 2009

1 Introducing the FTB-5240S/5240BP Optical Spectrum Analyzer

The FTB-5240S/5240BP Optical Spectrum Analyzer (OSA) is designed to measure optical power as a function of wavelength or frequency and Optical Signal to Noise Ratio (OSNR).

Your Optical Spectrum Analyzer offers truly portable spectral characterization for DWDM network commissioning as well as In-Band Optical Signal to Noise Ratio (OSNR) measurement for ROADM and 40 Gbit/s signals and networks.

The FTB-5240S is an expert DWDM OSA that is designed for efficient commissioning, maintenance and trouble-shooting of DWDM components and links in the field, from 50 GHz to CWDM network.



The FTB-5240BP is a high-resolution OSA designed for accurate and precise spectral measurements, even for channels with 12.5 GHz spacing.

Models

The Optical Spectrum Analyzer comes in different models:

- FTB-5240S: The FTB-5240S is a small form factor expert OSA (two-slot unit). It offers new modulation schemes, such as non-return-to-zero (NRZ), duo binary, differential phase-shift keying (DPSK), quadrature phase shift keying (QPSK), which present large line widths and often display multiple peaks. In-depth analysis ensures the correct identification and signal measurement of each carrier.
- FTB-5240S-P: It is the FTB-5240S model with a polarization controller. It is a hardware-ready version of an expert OSA (two-slot unit), without the software to compute the In-Band OSNR. You can upgrade this model on field using the software key, and it will become fully capable of In-Band OSNR measurement.
- FTB-5240S-P-InB: It is the FTB-5240 S-P model (two-slot unit) with the addition of an automated and integrated polarization controller. It is possible to change the state of polarization (SOP) at the input of the OSA and take advantage of a polarization diverse design to determine the OSNR in the band of a DWDM channel. This is required when noise is not flat under adjacent peaks or when crosstalk is dominant.
- FTB-5240BP: It is a three-slot model with a polarization controller for automated In-band testing and better optical performance.
- High Power Model (HPW): This model allows you to connect the FTB-5240S or FTB-5240S-P OSA to a network that carries very high optical power. This situation becomes more common with the deployment of latest CATV networks. The sensitivity of this OSA model is shifted accordingly and the module is protected to work under these extended power levels.

Typical Applications

You can use your Optical Spectrum Analyzer for the following tasks:

- Characterizing channels in the O- to U-band spectra
- Testing laser sources for spectral purity and power distribution
- Testing the transmission characteristics of optical devices
- Troubleshooting and monitoring key parameters on CWDM or DWDM signals to check system stability
- Characterizing all channel spacings, from 50 GHz DWDM to CWDM (from 12.5 GHz for FTB-5240BP)
- Testing high-speed networks (40 Gbit/s and higher)
- Measuring OSNR, but specifically within the channel (In-Band OSNR) for FTB-5240S-P-InB and FTB-5240BP models

Conventions

Before using the product described in this manual, you should understand the following conventions:

WARNING

Indicates a potentially hazardous situation which, if not avoided, could result in *death or serious injury*. Do not proceed unless you understand and meet the required conditions.

CAUTION

Indicates a potentially hazardous situation which, if not avoided, may result in *minor or moderate injury*. Do not proceed unless you understand and meet the required conditions.

CAUTION

Indicates a potentially hazardous situation which, if not avoided, may result in *component damage*. Do not proceed unless you understand and meet the required conditions.

IMPORTANT

Refers to information about this product you should not overlook.

2 Safety Information

WARNING

Do not install or terminate fibers while a light source is active. Never look directly into a live fiber and ensure that your eyes are protected at all times.

WARNING

Use of controls, adjustments and procedures for operation and maintenance other than those specified herein may result in hazardous radiation exposure or impair the protection provided by this unit.

Your instrument is a Class 1 laser product in compliance with standards IEC 60825-1:2007 and 21 CFR 1040.10. Laser radiation may be encountered at the output port.

The following label indicates that a product contains a Class 1 source:

CLASS 1
LASER PRODUCT

Note: *Label shown for information purposes only. It is not affixed to your product.*

3 Getting Started with Your Optical Spectrum Analyzer

Inserting and Removing Test Modules

CAUTION

Never insert or remove a module while the FTB-500 is turned on. This will result in immediate and irreparable damage to both the module and unit.

WARNING

When the laser safety LED ( ) is flashing on the FTB-500, at least one of your modules is emitting an optical signal. Please check all modules, as it might not be the one you are currently using.

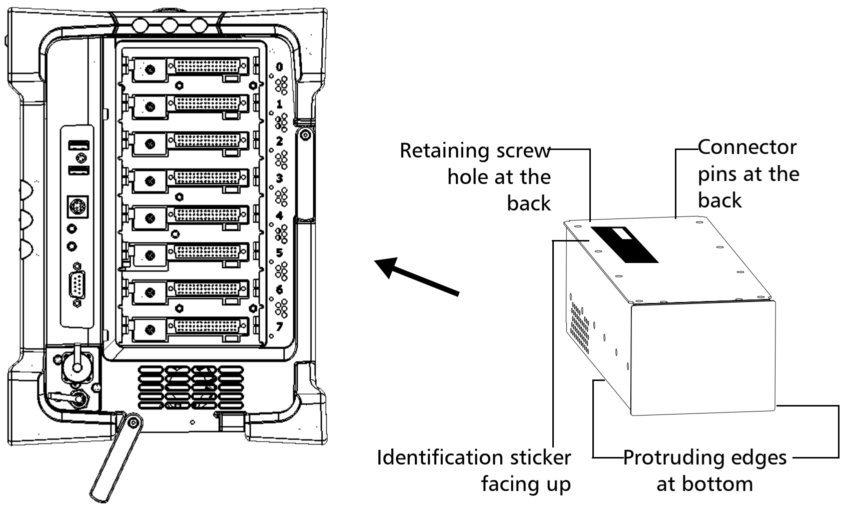
To insert a module into the FTB-500:

1. Exit ToolBox and turn off your unit.
2. Position the FTB-500 so that its right panel is facing you.

3. Take the module and place it so that the connector pins are at the back, as explained and shown below.

Identification sticker must be facing up and connector pins at the right of the retaining screw hole.

FTB-500 right panel



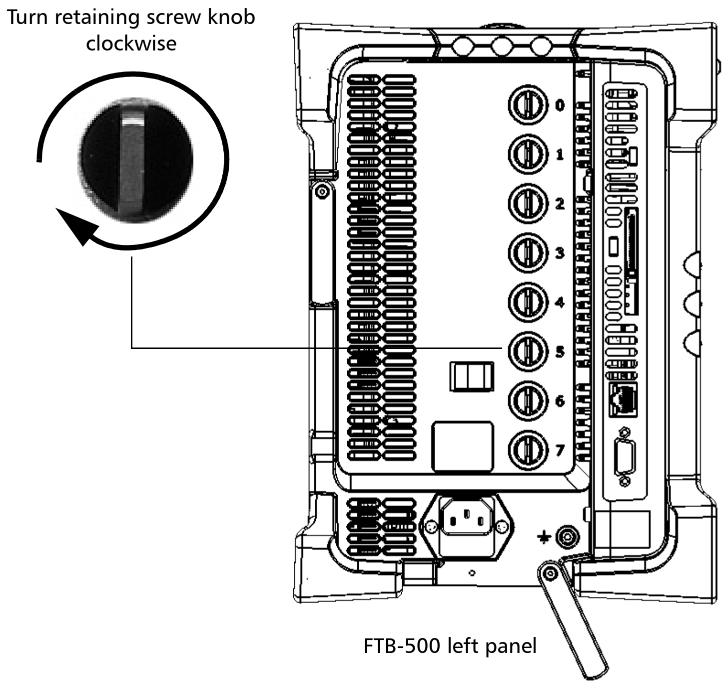
4. Insert the protruding edges of the module into the grooves of the receptacle's module slot.

5. Push the module all the way to the back of the slot, until the retaining screw makes contact with the receptacle casing.

6. Place the FTB-500 so that its left panel is facing you.

7. While applying slight pressure to the module, turn the retaining screw clockwise until it is tightened.

   This will secure the module into its “seated” position.

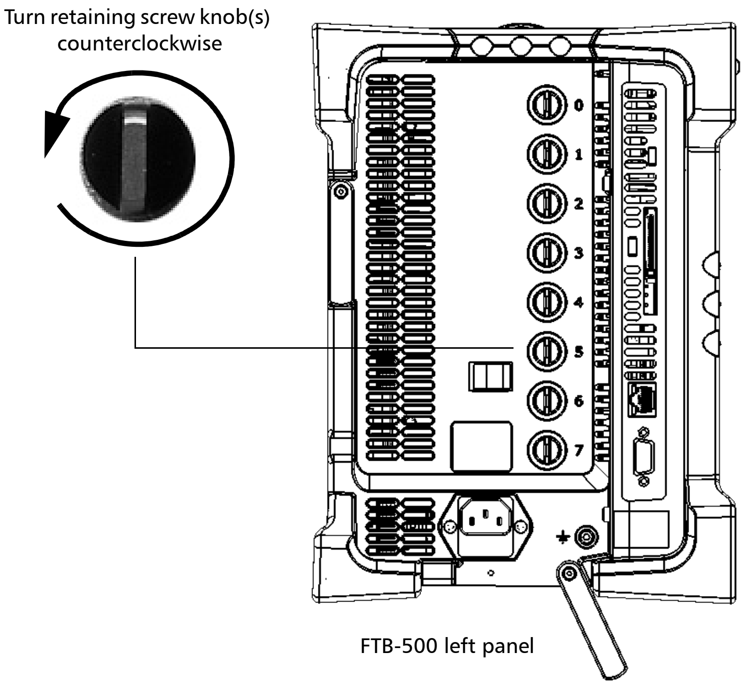


When you turn on the unit, the startup sequence will automatically detect the module.

To remove a module from the FTB-500:

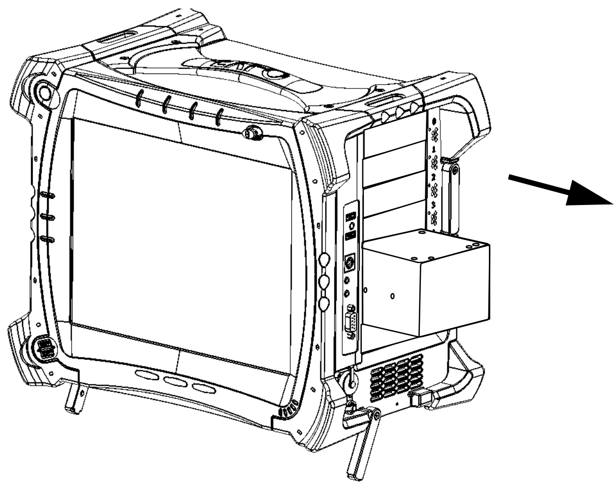
1. Exit ToolBox and turn off your unit.
2. Position the FTB-500 so that the left panel is facing you.
3. Turn the retaining screw counterclockwise until it stops.

   The module will be slowly released from the slot.



4. Place the FTB-500 so that the right panel is facing you.

5. Hold the module by its sides or by the handle (*NOT by the connector*) and pull it out.



Starting the Optical Spectrum Analyzer Application

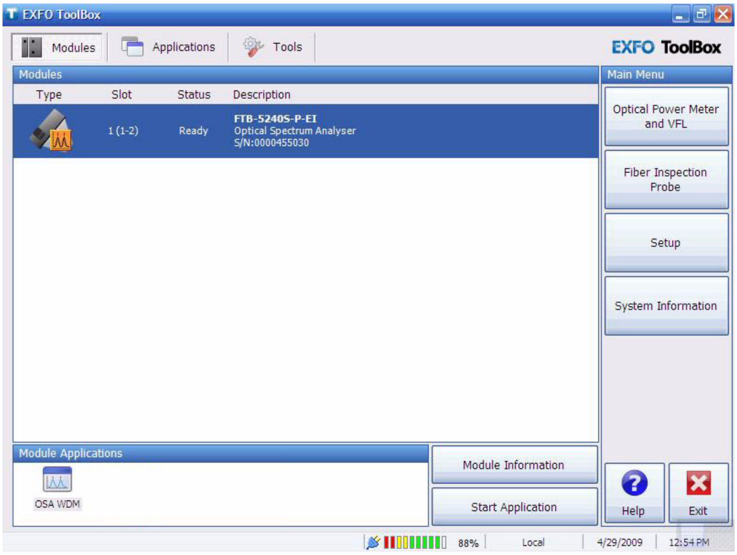
Your FTB-5240S/5240BP Optical Spectrum Analyzer module can be configured and controlled from its dedicated ToolBox application.

Note: *For details about ToolBox, refer to the* FTB-500 *user guide.*

To start the application:

1. From the main window, select the module to use.

   It will turn blue to indicate that it is highlighted.



2. Click the corresponding button in the **Module Applications** box.

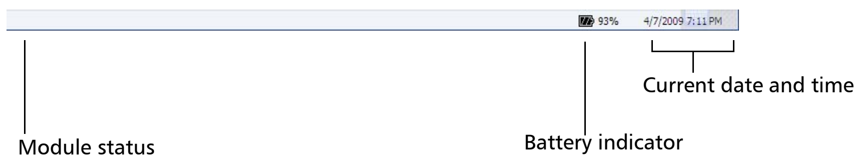
The main window (shown below) contains all the commands required to control the Optical Spectrum Analyzer:

IMPORTANT

For optimal test results, you must allow a minimum warm up period of two hours for your Optical Spectrum Analyzer before starting your tests.

Status Bar

The status bar, located at the bottom of the main window, identifies the current operational status of the FTB-5240S/5240BP Optical Spectrum Analyzer.



For more information about automating or remotely controlling the FTB-5240S/5240BP Optical Spectrum Analyzer, refer to the *FTB-500* user guide.

Exiting the Application

Closing any application that is not currently being used helps freeing system memory.

To close the application from the main window:

Click  in the top right corner of the main window.

OR

Click the **Exit** button located at the bottom of the function bar.

4 Preparing Your Optical Spectrum Analyzer for a Test

IMPORTANT

For optimal test results, you must allow a minimum warm up period of two hours for your Optical Spectrum Analyzer before starting your tests.

Cleaning and Connecting Optical Fibers

IMPORTANT

To ensure maximum power and to avoid erroneous readings:

- **Always inspect fiber ends and make sure that they are clean as explained below before inserting them into the port. EXFO is not responsible for damage or errors caused by bad fiber cleaning or handling.**
- **Ensure that your patchcord has appropriate connectors. Joining mismatched connectors will damage the ferrules.**

To connect the fiber-optic cable to the port:

1. Inspect the fiber using a fiber inspection microscope. If the fiber is clean, proceed to connecting it to the port. If the fiber is dirty, clean it as explained below.
2. Clean the fiber ends as follows:

   2a. Gently wipe the fiber end with a lint-free swab dipped in isopropyl alcohol.

   2b. Use compressed air to dry completely.

   2c. Visually inspect the fiber end to ensure its cleanliness.

3. Carefully align the connector and port to prevent the fiber end from touching the outside of the port or rubbing against other surfaces.

   If your connector features a key, ensure that it is fully fitted into the port's corresponding notch.

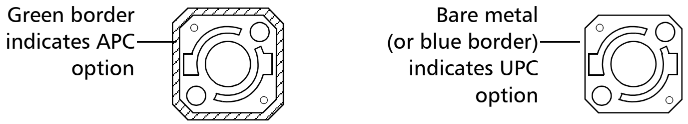
4. Push the connector in so that the fiber-optic cable is firmly in place, thus ensuring adequate contact.

   If your connector features a screwsleeve, tighten the connector enough to firmly maintain the fiber in place. Do not overtighten, as this will damage the fiber and the port.

Note: *If your fiber-optic cable is not properly aligned and/or connected, you will notice heavy loss and reflection.*

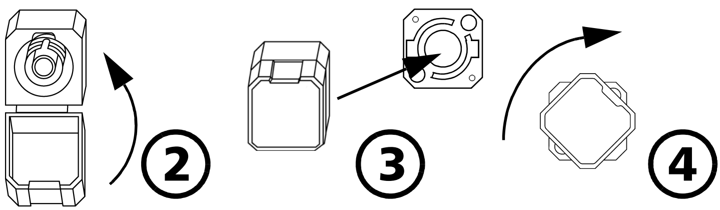
Installing the EXFO Universal Interface (EUI)

The EUI fixed baseplate is available for connectors with angled (APC) or non-angled (UPC) polishing. A green border around the baseplate indicates that it is for APC-type connectors.



To install an EUI connector adapter onto the EUI baseplate:

1. Hold the EUI connector adapter so the dust cap opens downwards.



2. Close the dust cap in order to hold the connector adapter more firmly.

3. Insert the connector adapter into the baseplate.

4. While pushing firmly, turn the connector adapter clockwise on the baseplate to lock it in place.

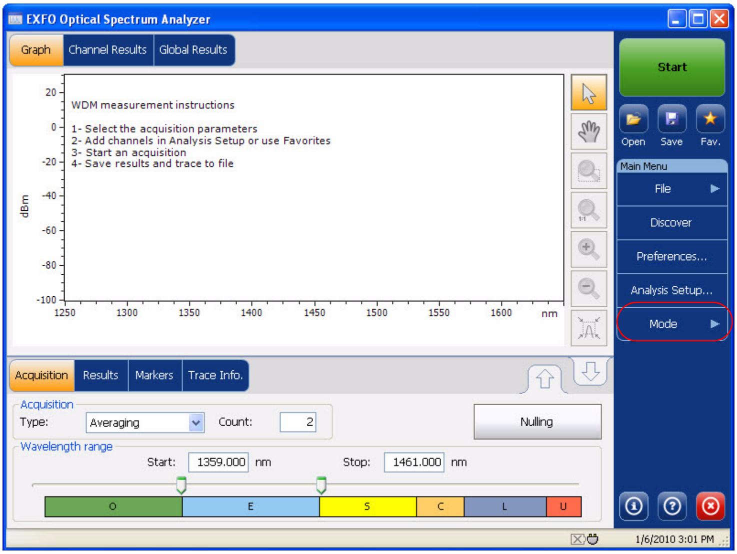
Selecting a Test Mode

Your Optical Spectrum Analyzer gives you different ways to test all your DWDM systems. Your OSA provides the following test modes:

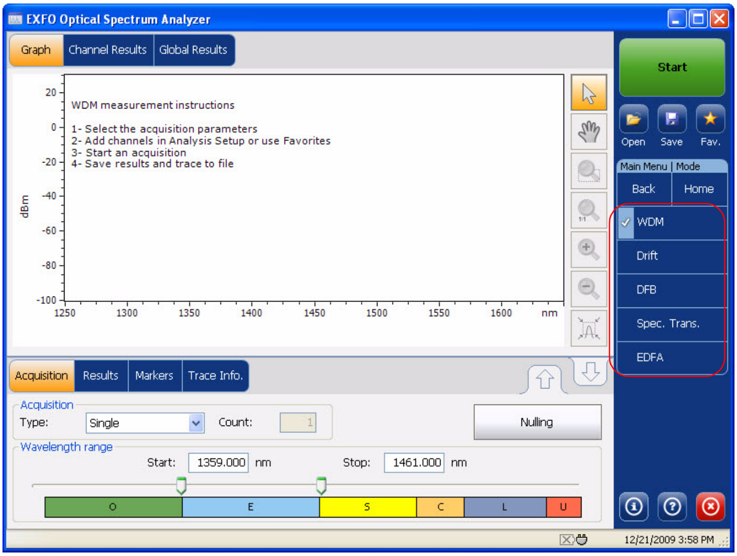
- **WDM**: Allows you to analyze an optical link using an OSA module. By default, WDM test mode is selected.
- **Drift**: Allows you to monitor an optical link for a fixed duration using an OSA module.
- **DFB**: Allows you to characterize a DFB laser source using the OSA module.
- **Spectral Transmittance**: Allows you to characterize the spectral transmittance of optical components such as filters.
- **EDFA**: Allows you to characterize the performance of an Erbium Doped Fiber Amplifier (EDFA) using the OSA module in field deployed systems (NB measurement assumes transmission conditions).

To select a test mode:

1. From the main menu, press **Mode**.



2. Select the desired test mode.



Once you select the mode, you will notice a against the selected mode and all the tabs on the main window and the main menu will change accordingly.

After selecting the test mode, you must configure it. You will find the instructions in Setting Up the Instrument in WDM Mode *on page 37*, Setting Up the Instrument in Drift Mode *on page 103*, Setting Up the Instrument in DFB Mode *on page 165*, Setting Up the Instrument in Spectral Transmittance Mode *on page 183* and Setting Up the Instrument in EDFA Mode *on page 209*.

Nulling Electrical Offsets

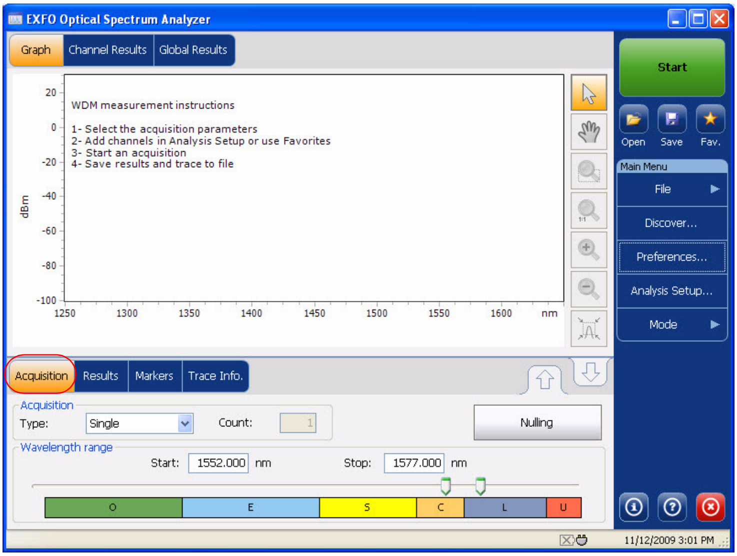
The offset nulling process provides a zero-power reference measurement, thus eliminating the effects of electronic offsets and dark current due to detectors.

Temperature and humidity variations affect the performance of electronic circuits and optical detectors. For this reason, EXFO recommends performing a nulling of the electrical offsets whenever environmental conditions change.

Nulling can be performed for all tests modes. In addition, a nulling is performed automatically each time you start the OSA application, and at regular intervals afterwards for WDM test mode.

To perform offset nulling:

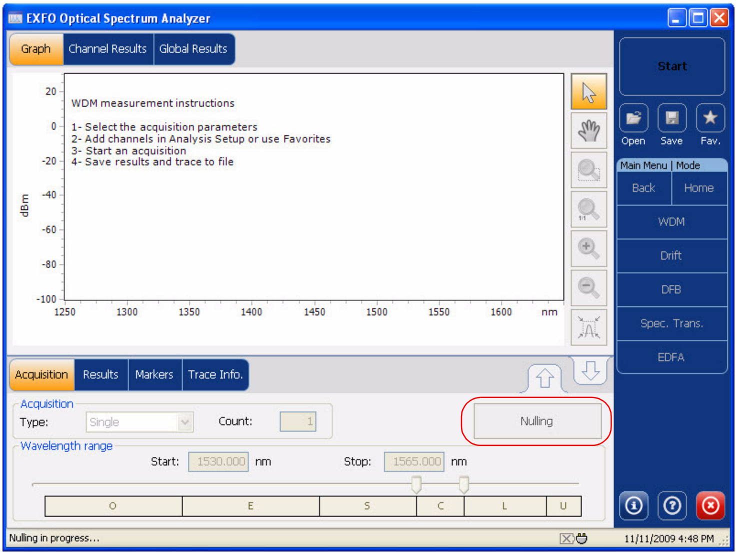
1. From the main window, select the **Acquisition** tab.



2. Disconnect any incoming signal to obtain an optimal accuracy.

3. Press **Nulling**.

You are notified that the nulling is in progress in the status bar. Nulling should be completed in a few seconds.



Note: *The* ***Start*** *and* ***Discover*** *options are not available during the nulling process.*

Performing User Calibration

Calibrating your module can help you achieve better results. It is particularly important when the measurement accuracy is critical or when your OSA has experienced unusual shock or vibrations. To reach the highest possible accuracy, you can perform a wavelength or power calibration. Your OSA allows you to modify and read the user calibration values, revert to the factory calibration, load and save the modified user calibration file. The user configuration file (*.txt) contains the reference and modified wavelength and power values.

You can perform user calibration for all tests modes. Select a test mode as explained in *Selecting a Test Mode* on page 18, and follow the procedures mentioned below for performing user calibration.

Note: *The procedure for performing user calibration is the same for all test modes. The procedure is explained with WDM mode only in this document.*

IMPORTANT

For optimal results, you must allow a minimum warm up period of two hours for your Optical Spectrum Analyzer before performing user calibration.

IMPORTANT

The correction factor list must be cleared before making new calibration measurements. If calibration measurements are made when user correction factors are inside the module, they will affect the measurements and the calibration results will be inapplicable.

A high level sequence of steps to perform user calibration are listed below, with the detailed instructions in the following procedures.

- Proper warm-up is required before performing user calibration.
- If user correction factors are present in the module (indicated by a message in calibration page),
  - save a backup copy of the correction factors (optional but strongly recommended).
  - clear user correction factors.
- Make measurements.
- Write measurement to text file according to following format.
  - First column: reference wavelength in nm
  - Second column: wavelength (in nm) read using the OSA application and module
  - Third column: reference power in dBm
  - Fourth column: power (in dBm) read using the OSA application and module

    100 calibration points limit in the application. Each value in the file must be separated using semi-colon (;) character.
- Write new correction factors inside the module
  - Load the text file built at preceding step
  - Press **Write to Module**.

    The application shows a warning message indicating to clear correction factors before making new measurements
- Verify that calibration points are properly applied.

An example of the calibration file format is given below.

1310.154; 1310.167; -1.34; -1.55

1490.000; 1490.000; 1.09; 1.15

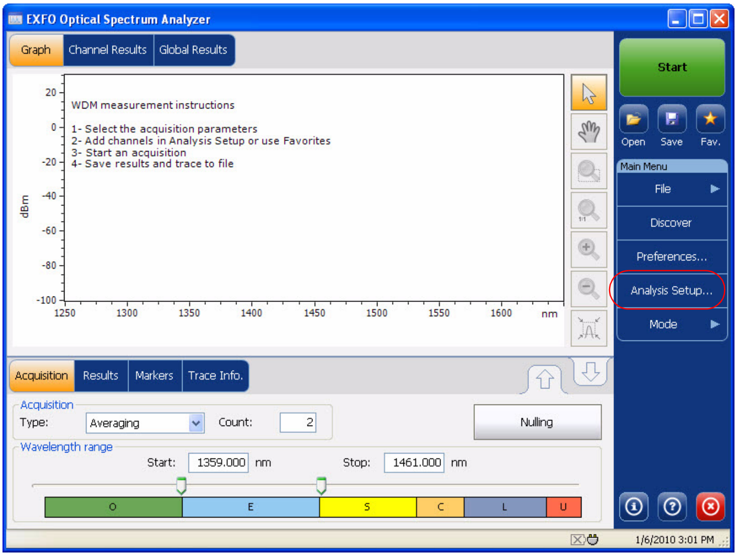
1551.334; 1551.298; -5.20; -5.45

1625.401; 1625.448; 0.00; 0.00

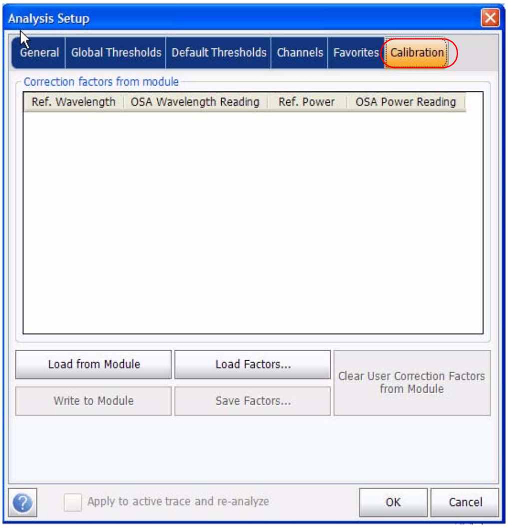
Note: *The decimal separator is a point (.). This decimal separator is constant independent of what the regional settings are.*

To perform a user calibration:

1. From the **Main Menu**, press **Analysis Setup**.

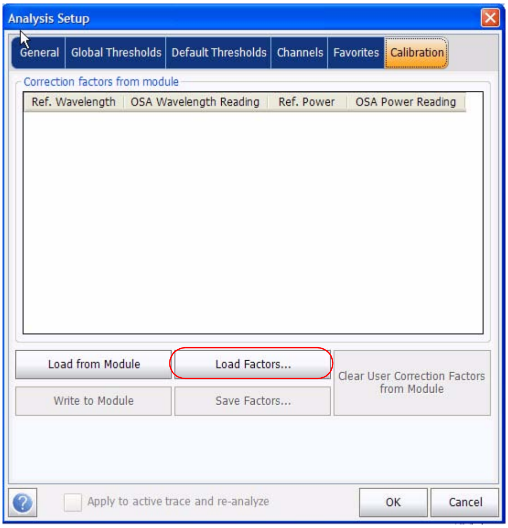


2. Select the **Calibration** tab.

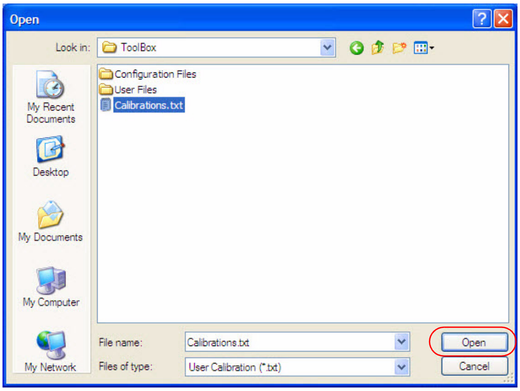


Note: *You cannot edit the power or wavelength values directly from the application. The modifications in the user calibration have to be made in a text file, and then it can be loaded in the application.*

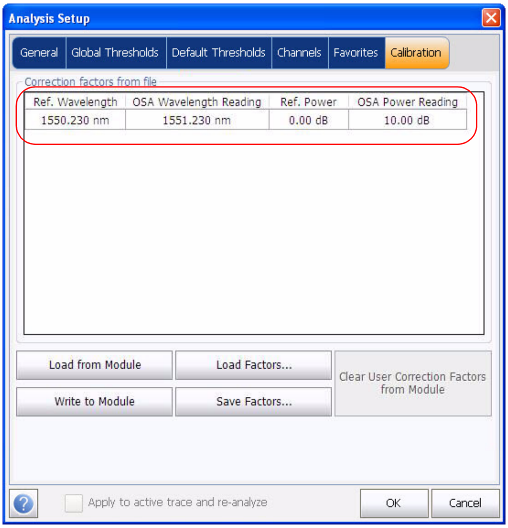
3. If a user calibration file exists, press **Load Factors** to load the file.



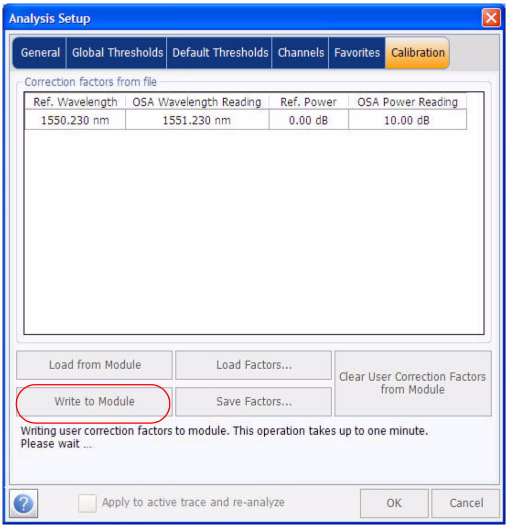
4. Select the modified user calibration file and press **Open**.



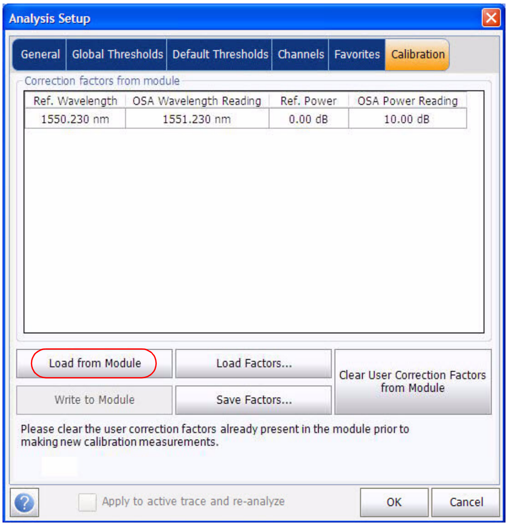
The calibration values will replace the Correction factors list in the **Analysis setup - Calibration** window.



5. Press **Write to Module** to apply the modified calibration values to the module.



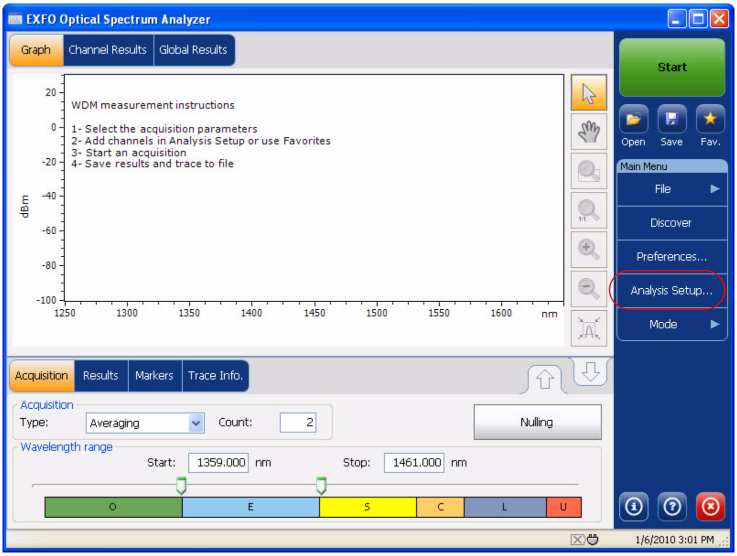
6. To verify that the calibration changes are properly applied to the module, press **Load from Module**.



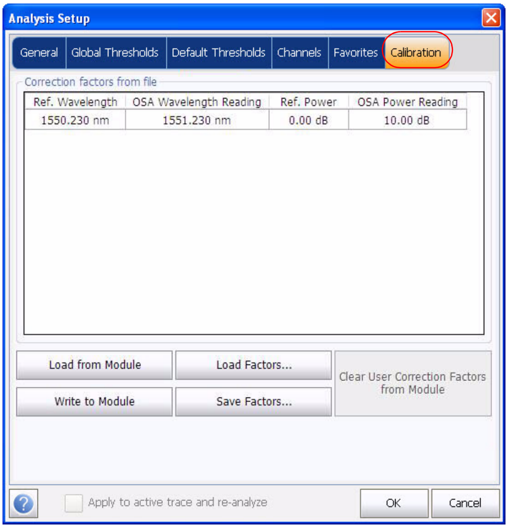
Note: *OK and Cancel buttons do not have any impact on the calibration page or the correction factors inside the module.*

To clear user correction factors:

1. From the **Main Menu**, press **Analysis Setup**.

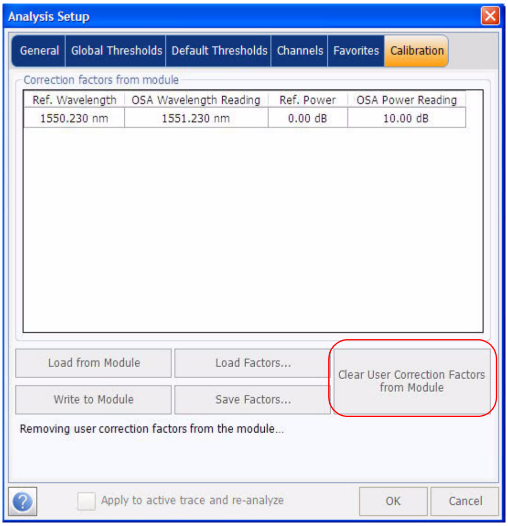


2. Select the **Calibration** tab.



3. Press **Clear user correction factors**, to load the factory calibration values.

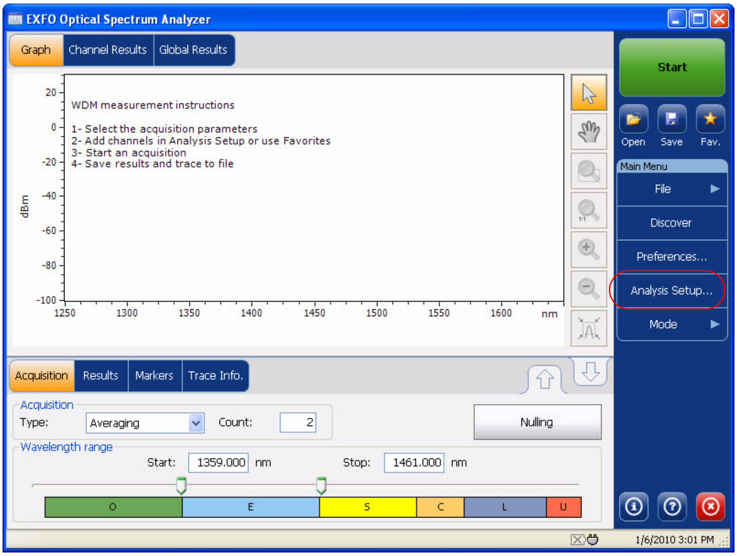
Note: *The application will ask if you want to clear user correction factors on the module and the user calibration will be removed. Press* ***Yes*** *or* ***No*** *as required.*



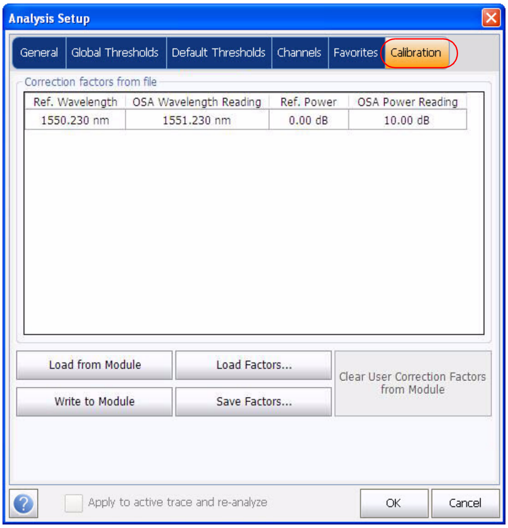
Note: ***OK*** *and* ***Cancel*** *buttons do not have any impact on the calibration page or the correction factors inside the module.*

To save a user calibration:

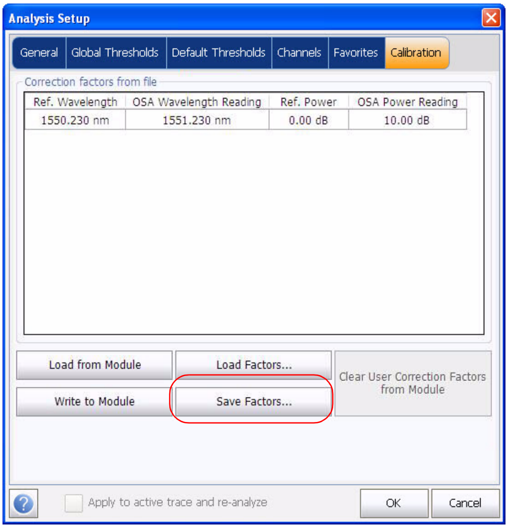
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Calibration** tab.



3. Press **Save Factors**, to save the modified user calibration values.



5 Setting Up the Instrument in WDM Mode

Before performing a spectral analysis in the WDM mode, you must set up your OSA module and the test application with the appropriate parameters, as explained in this chapter.

Select the WDM test mode as explained in *Selecting a Test Mode* on page 18 before setting up the WDM test parameters.

➤ The *preferences* are the result displayed in the graph and tables, as well as the job information and related comments saved with each file (See *Defining Preferences* on page 39 for more details).

➤ The *analysis parameters* include the channel list details, pass-fail threshold settings and allows you to select the noise and power calculation methods (See *Setting Up WDM Analysis Parameters* on page 58 for more details).

➤ The *acquisition parameters* include the type of measurement you want to perform and the wavelength range (See *Setting Up Acquisition Parameters* on page 99 for more details).

You can set up your unit in different manners, depending on your testing needs. The four possible ways for WDM mode are preferred, easy, efficient and post-processing.

➤ The preferred way is to use the complete analysis setup parameters and complete the information in all tables, as explained in *Setting Up WDM Analysis Parameters* on page 58. This setup will be used for the next acquisition.

➤ The easiest way to set up the instrument, especially when the operator does not know in advance what to expect at the input of the OSA is to use the **Discover** button. After **Discover** button has been pressed, a measurement and analysis will be performed according to the best setup determined by the instrument and this setup will be used for the next scan. This is explained in *Using the Discover Feature* on page 269.

- The most efficient way to setup the instrument is to use one of the favorites configurations, uploading a pre-customized acquisition and analysis setup configuration. The operator in the field only has to press the star button, select the appropriate configuration and press **Start**. As an example, a pre-customized configuration could be: “32 channels DWDM 50GHz”; “Toronto-Montreal CWDM” or “Vendor ABC DWDM ROADM 40Gb”. This is explained in *Managing Favorites* on page 86.
- The post-processing optimized way to setup the instrument is to change the active trace analysis parameters until satisfactory from the **Channel Results** and **Global Results** tabs, apply those changes to the analysis setup to use it for future measurements, and re-analyze the trace with those parameters. This is explained in *Changing Active Trace Analysis Parameters and Re-analyzing* on page 306.

Defining Preferences

The preferences window allows you to set general information and comments on trace, set display parameters and customize the WDM results table. This information is saved with all the traces.

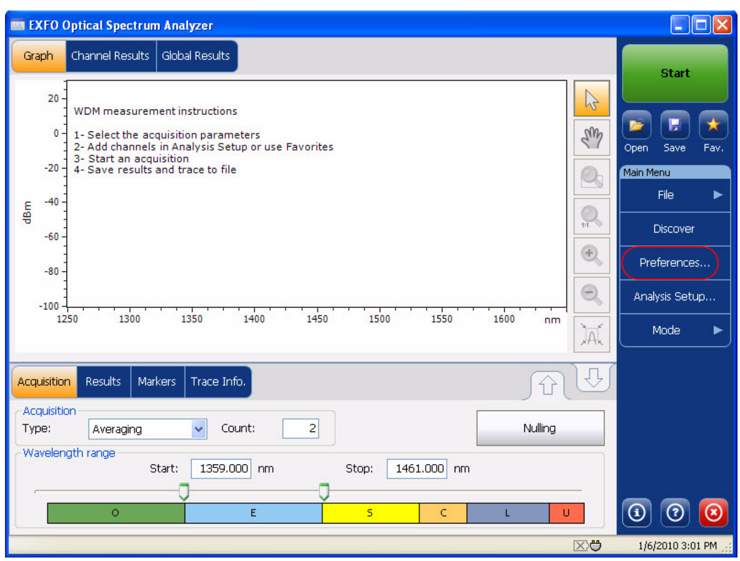
Defining Trace Information

The trace information relates to the description of the job to be done, cable and job IDs, and any relevant information about what is being tested.

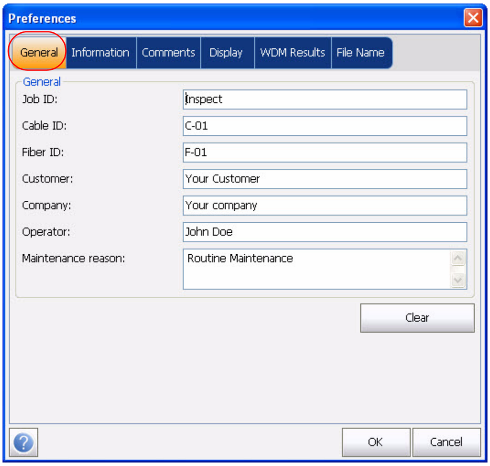
The Link ID is used by the application to propose a file name when you want to save the current acquisition. The link parameters are prefix and suffix values (file names) for the link IDs.

To enter general information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **General** tab.



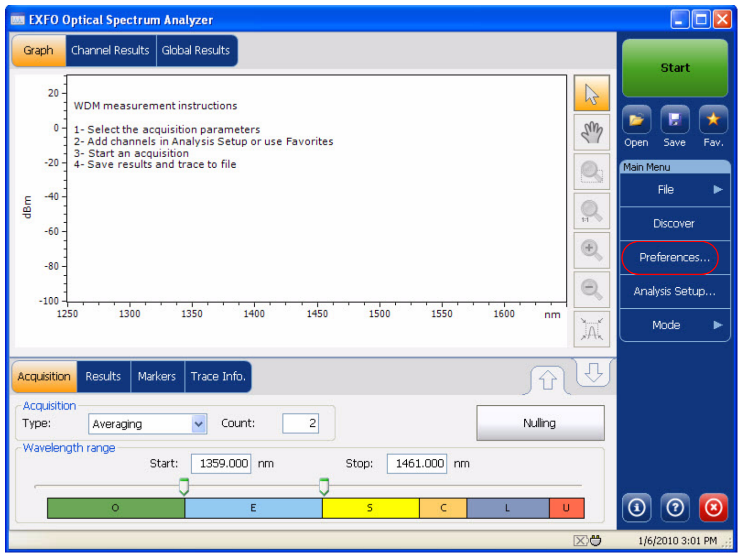
3. Define the general parameters as needed.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

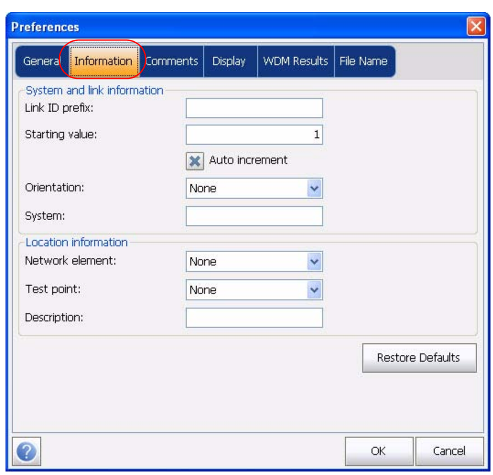
Press **Clear** to clear all the changes made in the **General** tab.

To enter link and location information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Information** tab.



3. Under **System and link information**, define the following parameters as needed:

   ➤ **Link ID prefix**: Sets the prefix value for the link ID. You can enter any alphanumeric value.

   ➤ **Starting value**: Sets the suffix increment starting value for the link ID.

   This value is incremented each time a new file is saved provided the **Auto Increment** option is selected.

IMPORTANT

When the **Auto Increment** option is not selected, while saving the trace file, you have to manually change the file name, else the application will overwrite the previously saved files every time you save a new trace.

   ➤ **Orientation**: Sets the orientation of the link.

   ➤ **System**: Sets the system information.

4. Under **Location Information**, define the following parameters as needed:

   ➤ **Network element**: Sets the type of network element.

   ➤ **Test point**: Sets the test point.

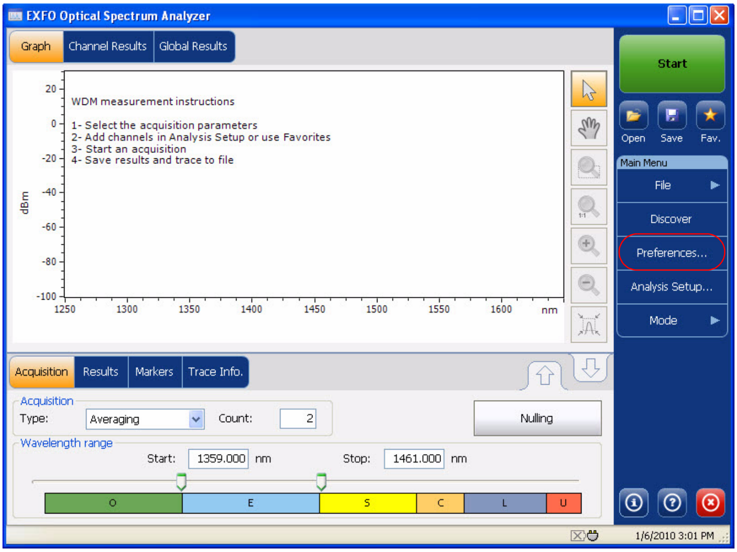
   ➤ **Description**: Enter the description of location if required.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

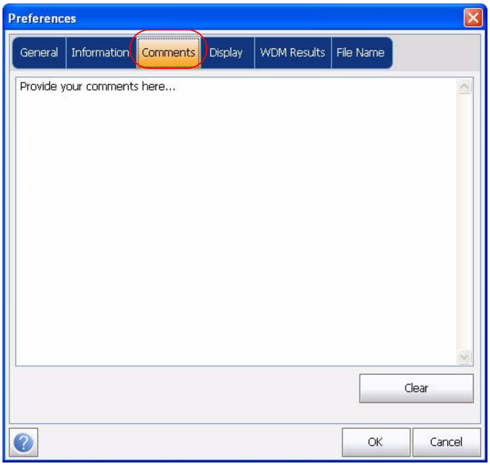
Press **Restore Defaults** to remove all the changes and apply the default values.

To enter comments:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Comments** tab.



3. Enter your comments for the current trace.
4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

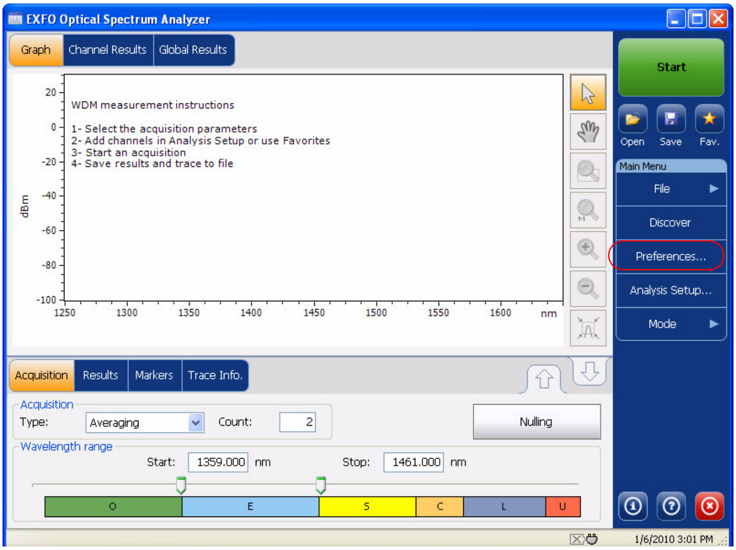
Press **Clear** to clear all the changes made in the **Comments** tab.

Defining Display Parameters

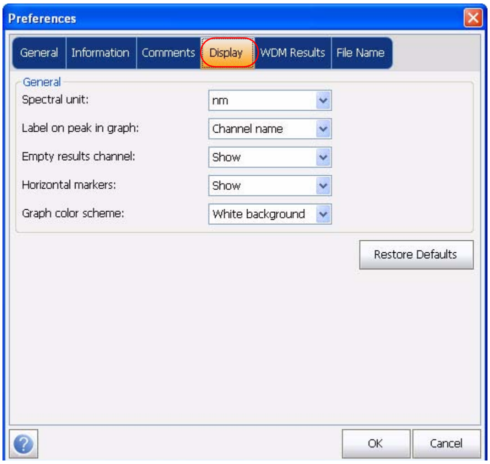
The application allows you to set display settings for the acquisition trace. You can set the spectral unit for the trace and the results table. You can also select the label that should appear on the peaks of the trace.

To define display parameters:

1. From the **Main Menu**, press **Preferences**.

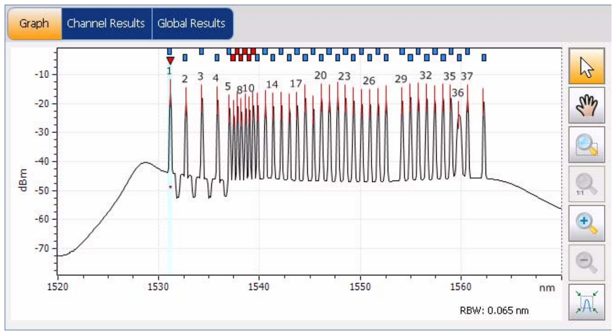


2. Select the **Display** tab.

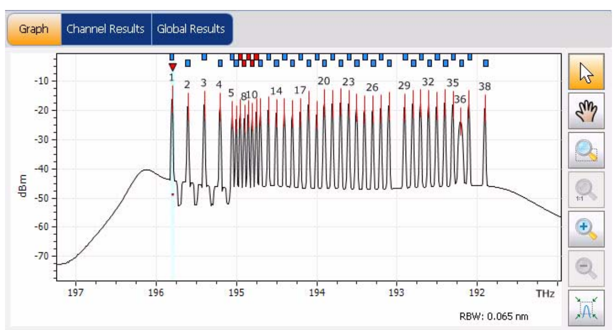


3. Select the spectral unit you want to work with.

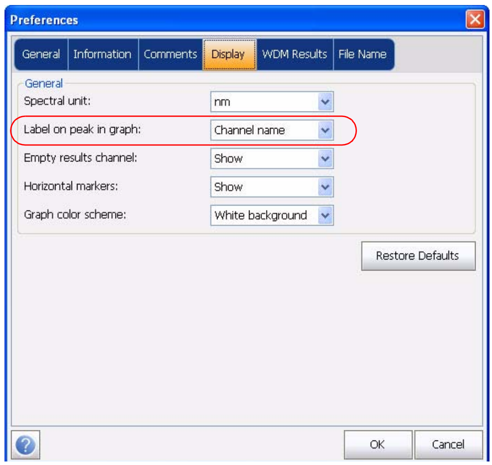
With the nanometer (nm) spectral unit, the trace will appear as shown below:



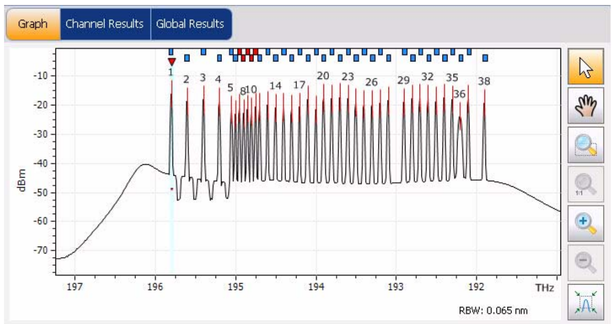
With the terahertz (THz) spectral unit, the trace will appear as shown below:



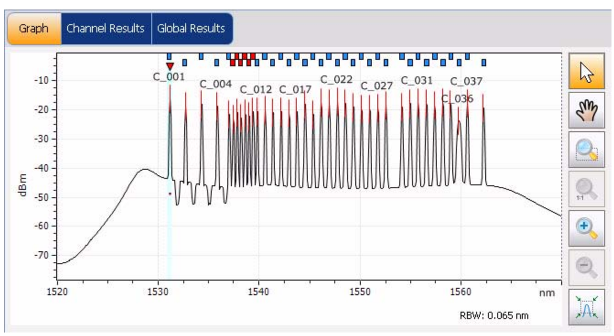
4. Select the label that will appear on the peaks in the graph.



Note: *The channel name and channel number cannot be shown at the same time. If a number is displayed on the peak, it means that the channel name for that peak is not defined. If a channel name is defined for the peak, it will be displayed at the top of the peak.*

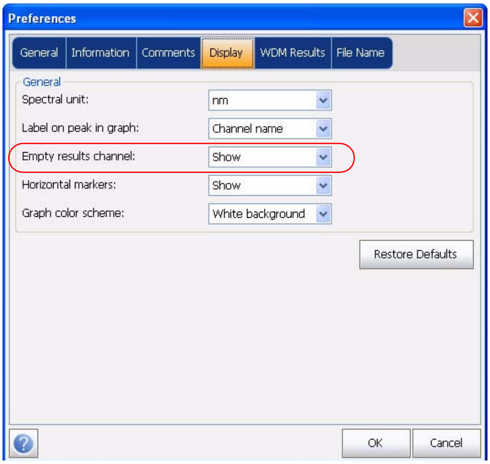


Default channel names

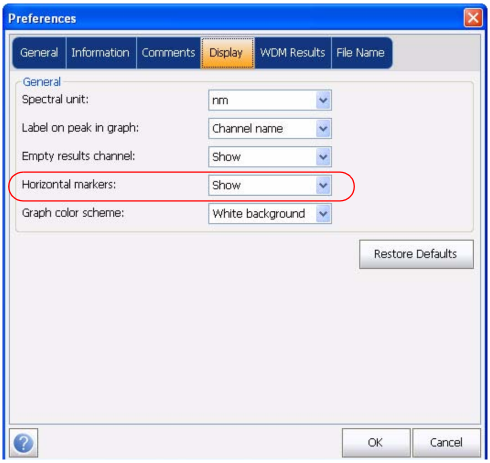


Defined channel names

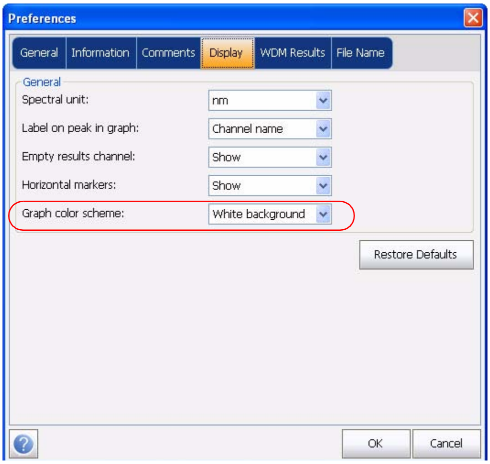
5. Select **Show** to display the empty channels from the channel list in the **Results** tab. If **Hide** is selected, it will not display the empty channels in the **Results** tab.



6. Select **Show** to enable horizontal markers in the **Markers** tab. If **Hide** is selected, horizontal markers will be disabled.



7. Select the background color scheme for Graph as desired.



8. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

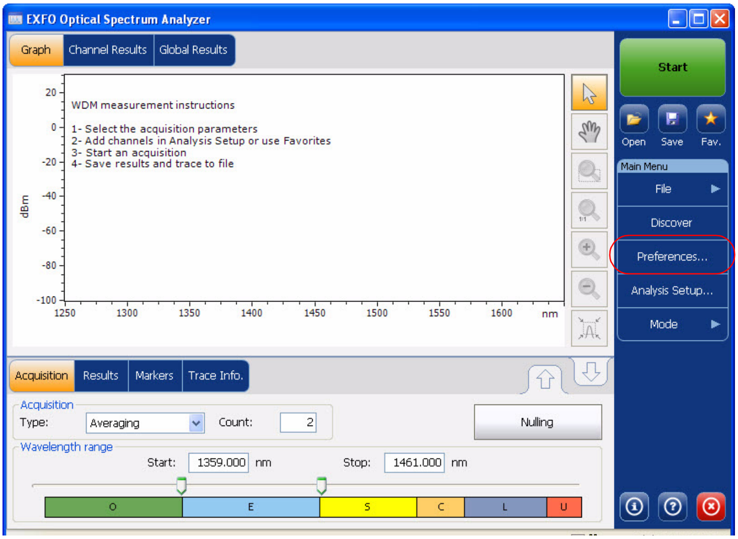
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing WDM Results Table

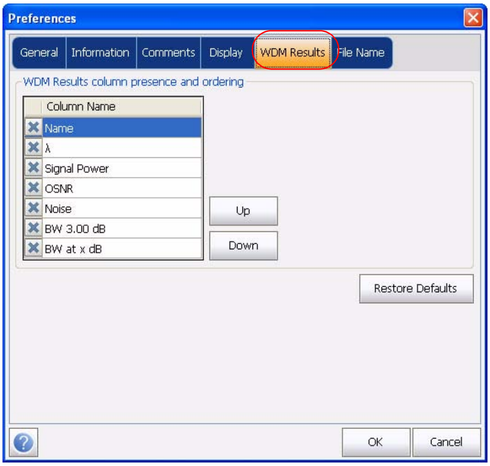
It is possible to select which results you would like to be displayed in the **Results** tab of your WDM tests.

To customize the results table:

1. From the **Main Menu**, press **Preferences**.



2. Select the **WDM Results** tab.



3. Select which parameters you want to display in the **Results** tab from the list of available choices:

- **Name**: indicates the name of channel.
- **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.
- **Signal Power**: indicates the signal power for the selected channel (excludes noise).
- **OSNR**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).
- **Noise**: indicates the noise level for the selected channel.
- **BW 3.00 dB**: indicates bandwidth measured by taking the width of a signal at 50 % linear power of the peak or -3 dB from the peak.
- **BW at x dB**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.

4. Press **Up** or **Down** to change the order in which the columns will appear in the **Results** tab.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

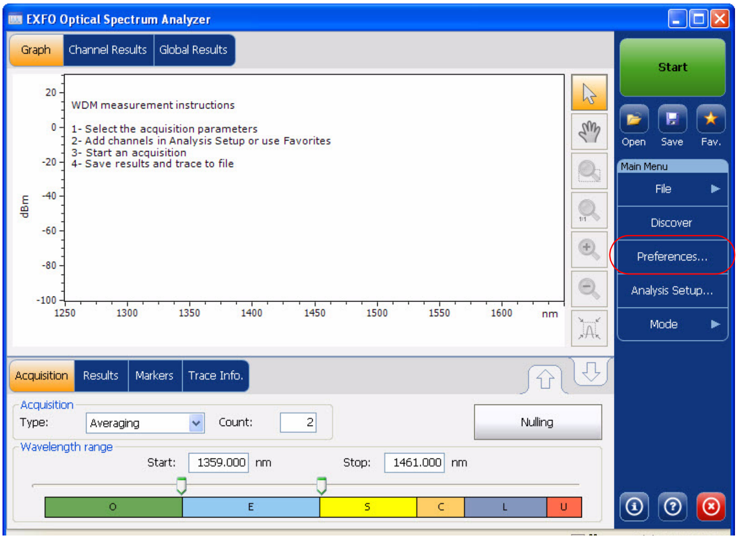
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing WDM File Name

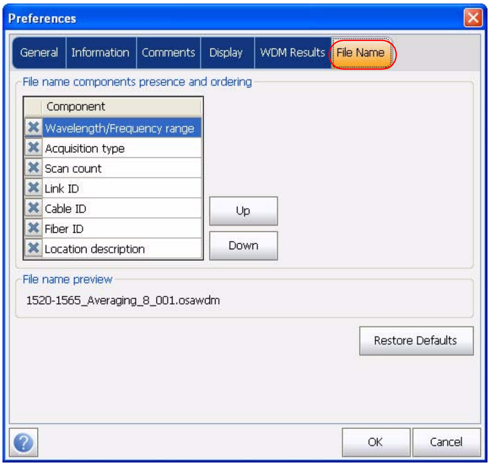
The application shall provide a way to define the name of the next file to be saved. Defining a file autonaming format will allow you to quickly and automatically name traces in a sequential order. The customized name appears when the file is saved using the **Save As** option. It is possible to select which fields you want to include in the file name and the order in which it should be displayed.

To customize the file name:

1. From the **Main Menu**, press **Preferences**.



2. Select the **File Name** tab.

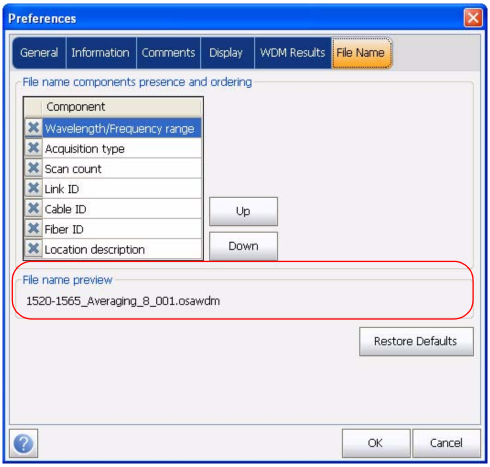


3. Select which parameters you want to include in the file name from the list of available choices:

- **Wavelength/frequency range**: indicates the current wavelength/frequency acquisition range.
- **Acquisition type**: indicates the current acquisition type selected.
- **Scan count**: indicates the current number of scans in the acquisition tab.
- **Link ID**: indicates the prefix value for the link ID configured in the **Preferences-Information** tab.
- **Cable ID**: indicates the prefix value for the cable ID configured in the **Preferences-General** tab.
- **Fiber ID**: indicates the prefix value for the fiber ID configured in the **Preferences-General** tab.
- **Location description**: indicates the location description provided in the **Preferences-Information** tab.

4. Press **Up** or **Down** to change the order in which the field values will appear in the file name.

   Based on the fields selected, a preview of the file name shall be displayed under **File name preview**. The field values are separated with an underscore (_).



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default settings.

Setting Up WDM Analysis Parameters

This section presents the various analysis settings for the application, particularly the channel list and settings. These settings are applied on subsequent acquisitions. You can set the channel list, global thresholds, default channel thresholds, channel parameters, manage favorite configurations and perform user calibration.

Note: *The analysis setup parameters will be applied to the global results and channel results, upon the next acquisition. However, you can also apply the modifications to the active trace in order to re-analyze it.*

Defining General Settings

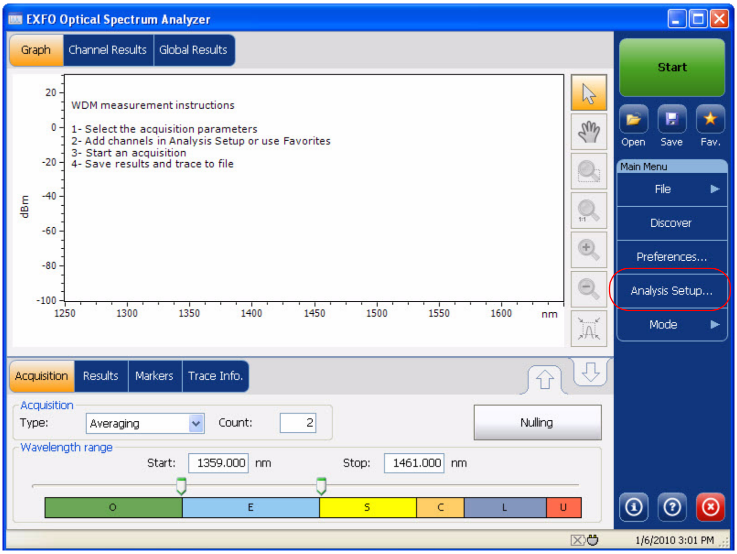
The general analysis parameters for WDM acquisitions affect the calculation of the results. These calculations take place after an acquisition. If these settings are modified, they will be applied to the next acquisition. However, any modifications made in the general parameters can also be applied to the active trace for re-analysis.

IMPORTANT

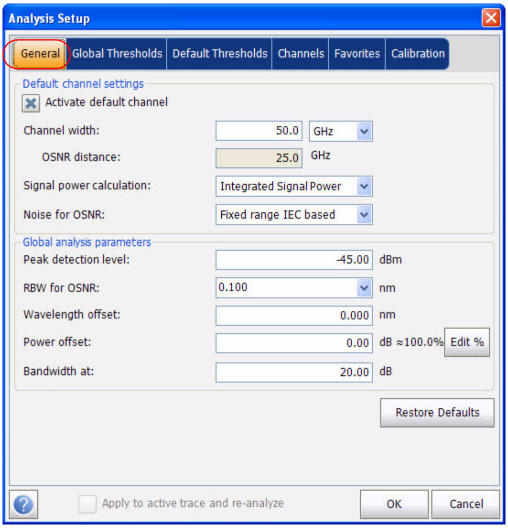
In the General tab, you can set the default channel parameters. Any channel found during an acquisition that is not defined in the channel list will be analyzed according to the default channel settings.

To define general settings:

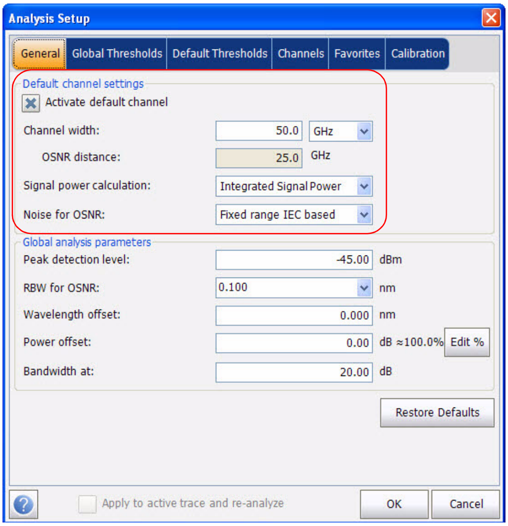
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **General** tab.



3. Under **Default channel settings**, define the following parameters as needed:



➤ Clear the **Activate default channel** selection, to use the currently defined channel for analysis. This reduces the analysis time by eliminating the peak detection over the complete spectral range. The peaks outside the defined channel will not be detected or analyzed.

- **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.

  For default channels, the channel width that sets the limits of the channel, should be the same as the channel distance or smaller (channel distance is defined while creating a channel list). If the channel width is not compatible with the channel spacing, either a single peak may be found for two distinct channels and two analysis would be performed and displayed for that peak, or, it is possible that two peaks may be found within the same channel and be considered as one multi-peak signal. With this result, you can use markers to find the spacing between adjacent channels or to find the channel width.

- **OSNR distance (GHz or nm)**: OSNR distance is automatically set at the channel edge, that is, at half of the channel width from the center wavelength.

- **Signal power calculation**: indicates which calculation method to apply for signal power value.

  Integrated signal power: The integrated signal power represents the sum of the power values included between the channel limits of this channel, minus the estimated noise contribution between the same boundaries. In some cases, for instance CATV signals, signals with high-frequency modulation, or signals with an inherent line width similar or larger than the OSA's resolution bandwidth, this calculation becomes a better estimation of the true signal power.

  Peak signal power: The peak signal power represents the maximum power value inside the channel, but it differs a little from the peak measurement on the spectrum due to the fact that the estimated noise is subtracted to get the peak signal power.

  Total channel power: The total channel power represents the sum of the power values included inside the channel width (including noise).

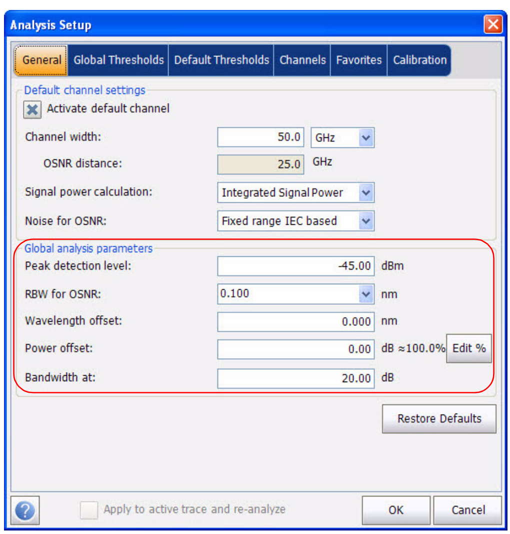
➤ **Noise for OSNR**: indicates which calculation method to use for OSNR value.

Fixed range IEC based (IEC): The IEC method uses the interpolation of noise measured on both sides of the signal to estimate the noise level. The position at which the noise is estimated from the center wavelength is given by the OSNR distance.

In-Band (InB): The In-Band method uses a series of scans having different polarization states to calculate the noise level under the peak (In-Band).

In-Band narrow filter (InB nf): The In-Band narrow filter method uses additional processing to provide an accurate OSNR value for the narrow carved noise. This is because with narrow filters, the noise level under the peak is not uniform and the OSNR value depends on the processing width selected.

4. Under **Global analysis parameters**, define the following parameters as needed:



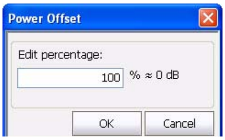
➤ **Peak detection level (dBm)**: indicates the minimum power level from where the peak can be considered as a signal.

➤ **RBW for OSNR (nm)**: indicates the resolution bandwidth for the selected OSNR value. The resolution bandwidth of an OSA determines its ability to deal with close optical channel spacing. It is measured as the width of the response curve at half peak power (i.e., 3 dB down) of the instrument to a monochromatic test signal. The instrument's RBW value is written below the graph on the right (below wavelength offset).

➤ **Wavelength offset (nm)**: indicates the offset value applied on the wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.

➤ **Power offset (dB)**: indicates the offset value applied on the power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. The power offset appears in the graph, on the left, above the ↕ arrow.

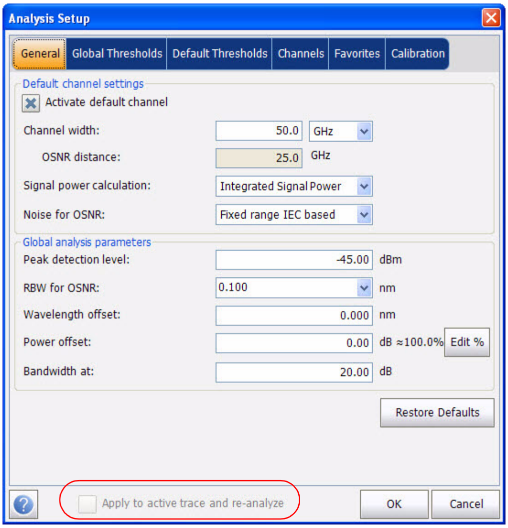
To edit the power offset value in percentage, press **Edit %** button.



The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

➤ **Bandwidth at (dB)**: Set the power level used, relative to the channel peak power, to compute the bandwidth.

5. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



6. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

You can set your pass/fail threshold limits in different ways depending on the type of test you are performing.

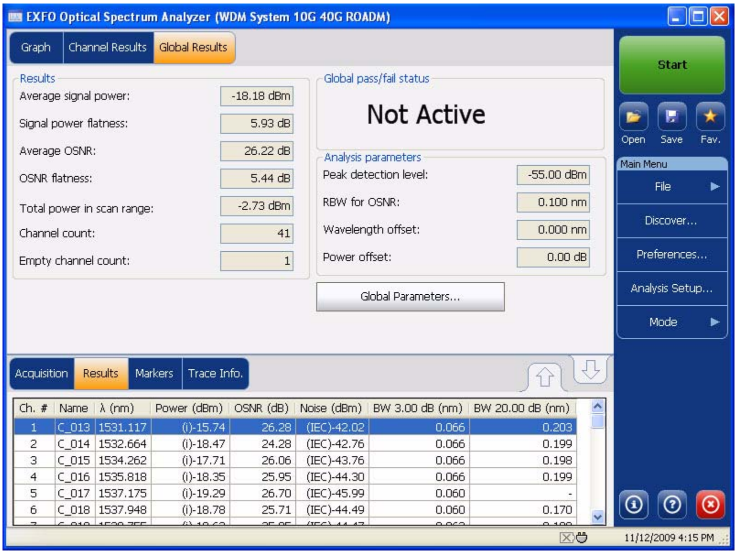
Threshold Limit	Definition
None	No threshold limit is set. The results will be displayed without a Pass/Fail verdict.
Min. only	The threshold limit is set for a minimum value only. The Pass/Fail verdict is declared as Pass (in green), when the value is equal to or greater than the minimum threshold set. The verdict is declared as Fail (in red), when the value is below the minimum threshold set.
Max. only	The threshold limit is set for a maximum value only. The Pass/Fail verdict is declared as Pass (in green), when the value is equal to or less than the maximum threshold set. The verdict is declared as Fail (in red), when the value is above the maximum threshold set.
Min. and Max.	The threshold limit is set for the minimum and maximum value. The Pass/Fail verdict is declared as Pass (in green), when the value is equal to or within the minimum and maximum thresholds set. The Pass/Fail verdict is declared as Fail (in red), when the value is beyond the minimum or maximum thresholds set.
Use Default	This threshold limit option is available in the **Channel Parameters** tab of **Channel Results** window only. When this limit is set, the defaults set for the default channels in the **Analysis Setup** tab will be applied to the channel.

Defining Global Thresholds

Global thresholds will be applied to the global results during the next acquisition. However, any modifications made in the global thresholds parameters can also be applied to the active trace for re-analysis.

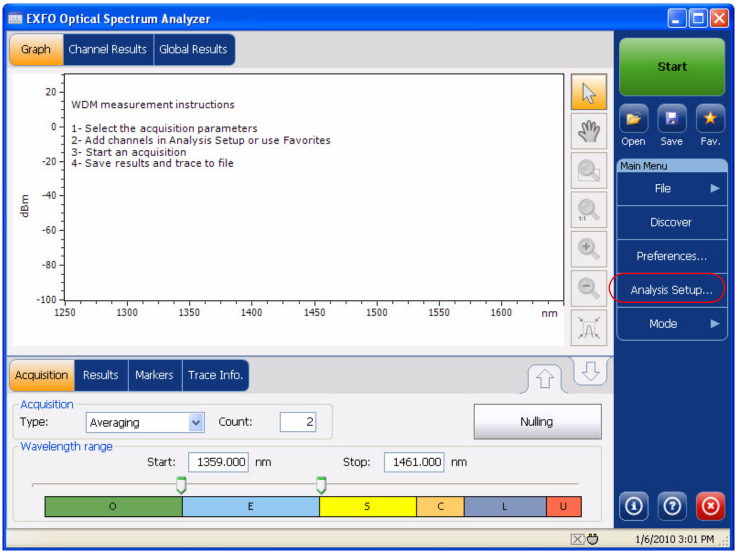
The application allows you to activate and deactivate the threshold functionality with a single control. When thresholds are globally enabled, the results are displayed with the Pass/Fail status based on various settings (global results, channel results). In addition, a global pass/fail status is also displayed in the **Global Results** tab (See *Global Results Tab* on page 304).

When thresholds are globally disabled, results are displayed without a Pass/Fail status and the **Global pass/fail status** will not be active in the **Global Results** tab. The **P/F** column under the results table will not be displayed.



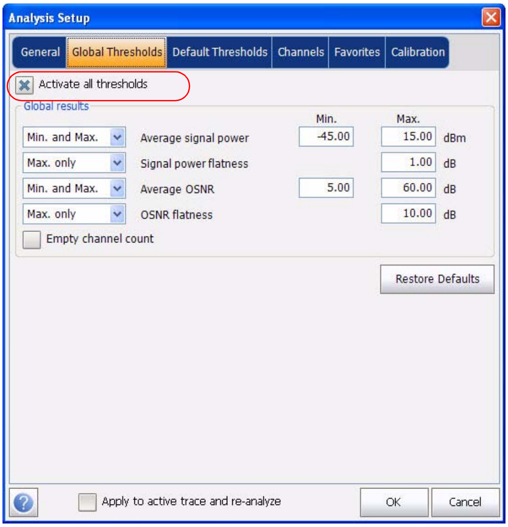
To define global thresholds:

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Global Thresholds** tab.

3. Select the **Activate all thresholds** option, to manually set the global threshold values. If this option is not selected, all the thresholds will be deactivated, results are displayed without a Pass/Fail status and **Global pass/fail status** will not be active in the **Global Results** tab.



4. Enter values in the boxes as explained below:

➤ **Average signal power (dBm)**: indicates the sum of the signal powers of all the peaks detected in the current acquisition, divided by the total number of peaks.

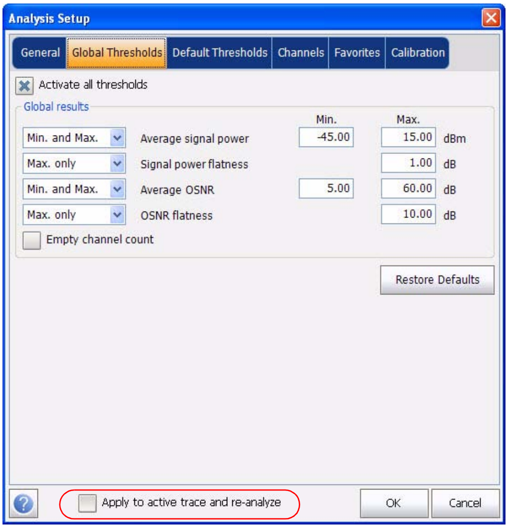
➤ **Signal power flatness (dB)**: indicates the difference between the maximum and minimum signal power values of the detected peaks, in dB.

➤ **Average OSNR (dB)**: indicates the sum of the entire OSNR of the peaks detected in the current acquisition, divided by the total number of peaks.

➤ **OSNR flatness (dB)**: indicates the difference between the maximum and minimum OSNR values of the detected peaks, in dB.

➤ **Empty channel count**: Select this option to obtain the number of empty channels from the channel list in the **Global Results** tab. If this option is not selected, it will not display the Pass/Fail verdict in the **Results** tab for the empty channels.

5. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



6. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

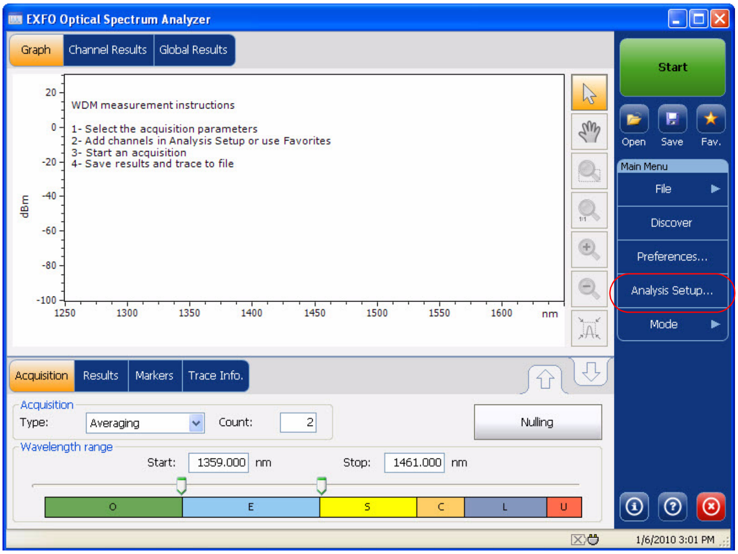
Defining Default Thresholds

Default thresholds will be applied to any channel found outside the channel list during the next acquisition.

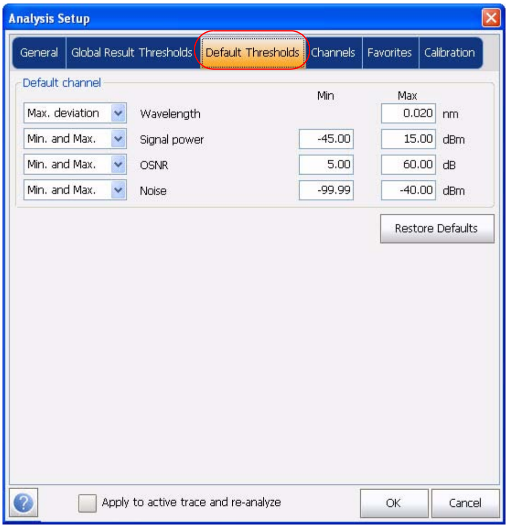
Note: *The default thresholds settings are enabled only when the* ***Activate all thresholds*** *option is selected in the* ***Global Thresholds*** *tab. For more information, see* Defining Global Thresholds *on page 67.*

To define Default Thresholds:

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Default Thresholds** tab.



3. Enter values in the boxes as explained below:

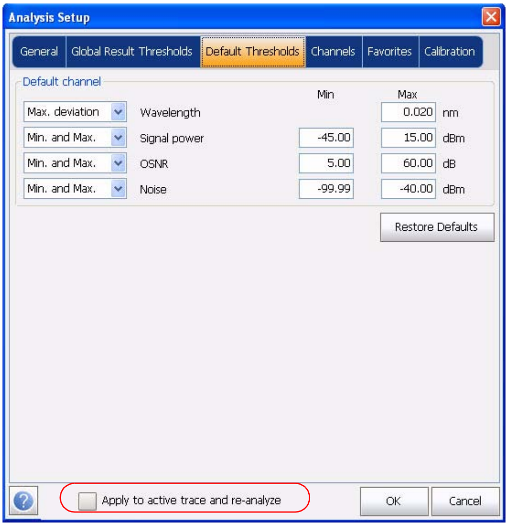
➤ **Wavelength/Frequency (nm/GHz)**: indicates the channel's central wavelength/frequency.

➤ **Signal power (dBm)**: indicates the signal power for the selected channel (excludes noise).

➤ **OSNR (dB)**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).

➤ **Noise (dBm)**: indicates the level of the noise for the selected channel.

4. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Managing Channels

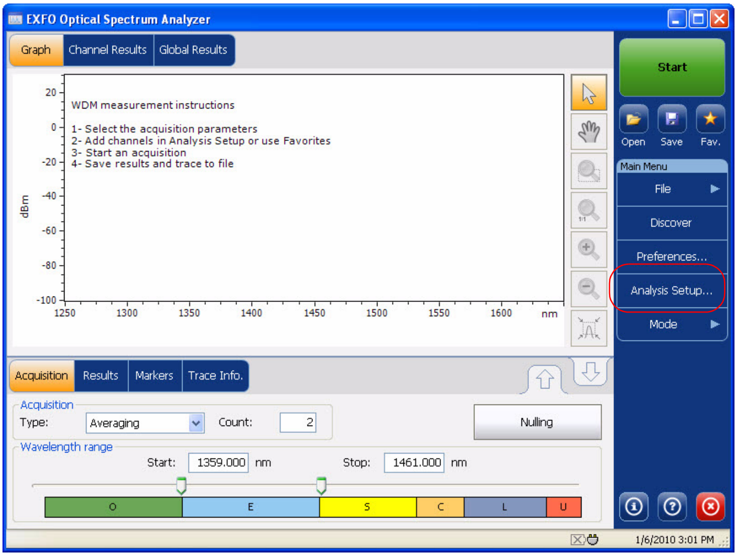
Testing DWDM systems involves characterizing multiple signals in a link. The application allows you to define channels using a channel editor or quickly generate them from the current data. You can also rapidly create a list of equally spaced channels. Once a channel list is created, you can modify it as needed. You can edit the analysis parameters for one channel or multiple channels.

Adding Channels

While creating the channel list, some channels may overlap. When the channel widths are specified in nm, two channels are considered to be overlapping when more than 0.001 nm (approximately) of wavelength range is common between the two channels.

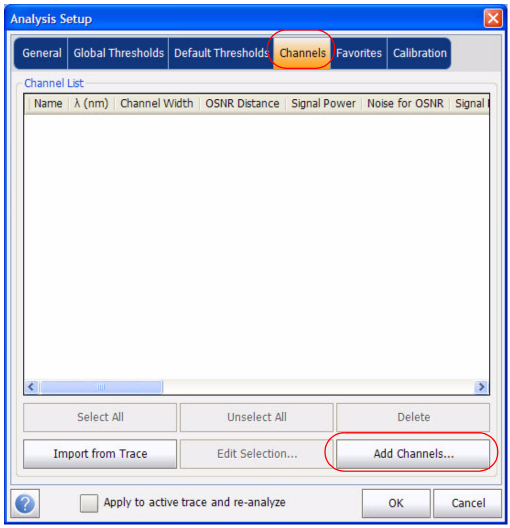
To add a channel list:

1. From the **Main Menu**, press **Analysis Setup**.

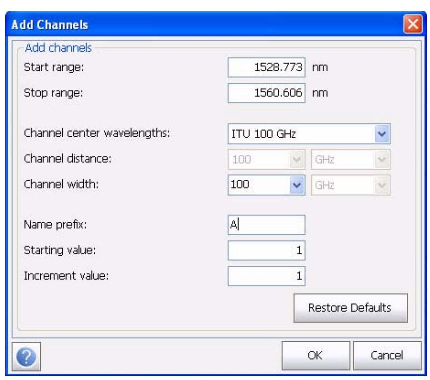


2. Select the **Channels** tab.

3. By default, the channel list is empty. Press **Add Channels**.



4. Enter values in the boxes as explained below:



➤ **Start range (nm or THz)**: indicates the starting range of the channel list.

- **Stop range (nm or Thz)**: indicates the ending range of the channel list.
- **Channel center wavelength/frequency:** indicates the spectral center-of-mass for the peak in that channel.

Note: *When using the custom channel center wavelength option, the first channel will be centered at the Start Range, and the list will be created using channel distance and channel width.*

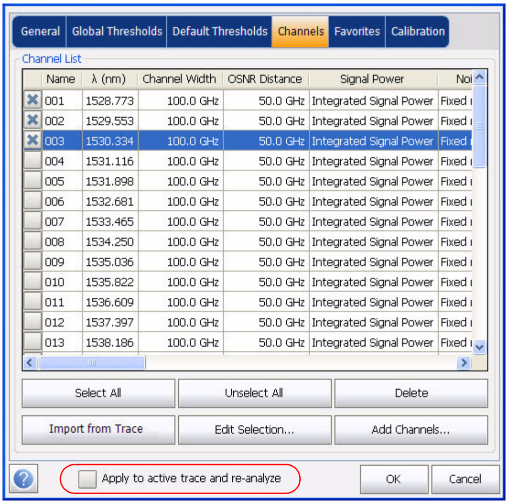
- **Channel distance (nm or GHz)**: indicates distance between the channels. The value of channel distance will be set depending on the selection made for the channel center wavelength option. The channel distance box will be enabled only when the channel center wavelength option is set to custom.
- **Channel width (nm or GHz)**: indicates the limit inside which the power values will be considered in the channel. Integrated power is calculated on channel width.
- **Name prefix:** Adds prefix to the channel names.
- **Starting Value:** Sets the increment starting value for the channel name in the channel list.
- **Increment value:** Sets the increment value for the channel name in the channel list.

5. Press **OK** to return to the **Channels** window, which now lists the added channels.

Note: *When new channels are added, the Use Default thresholds will be applied to the channel parameters.*

Note: *A warning message will be displayed if any channels are overlapping, but the analysis can still be performed on overlapping channels. If any duplicate channels are added, a confirmation message will be displayed to overwrite the existing channels with the duplicate channels.*

6. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option.

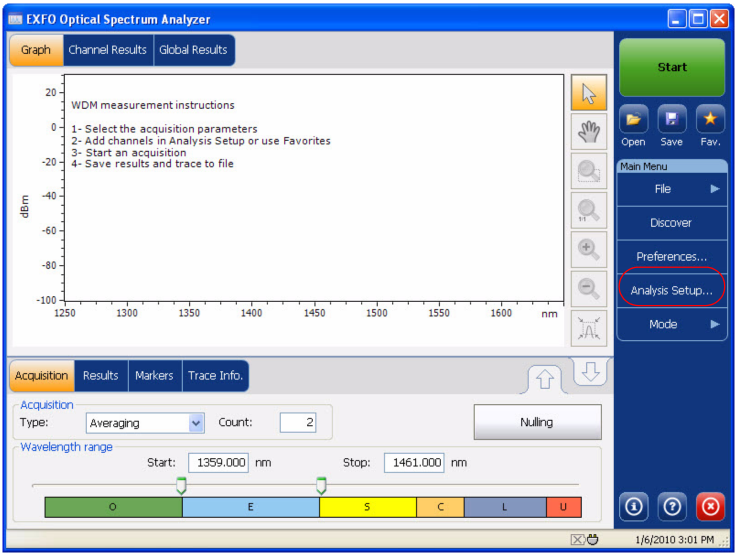


7. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

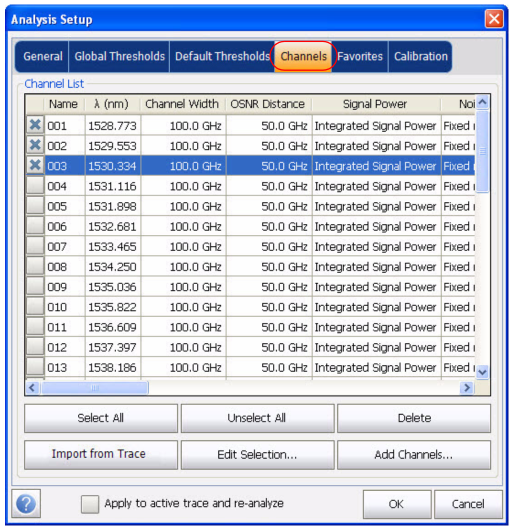
Note: *The application displays a message if more than 1000 channels are added. You can exit the* ***Analysis Setup*** *window only after deleting the extra channels from the channel list. You can delete the channels manually as required.*

To edit the parameters of a specific channel:

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.

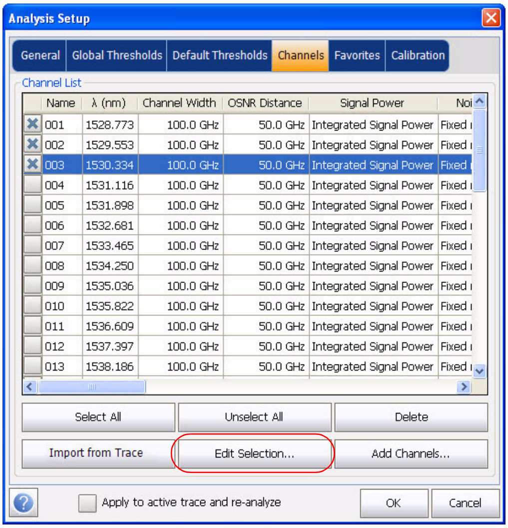


3. Select the channel or channels to be modified in the channel list.

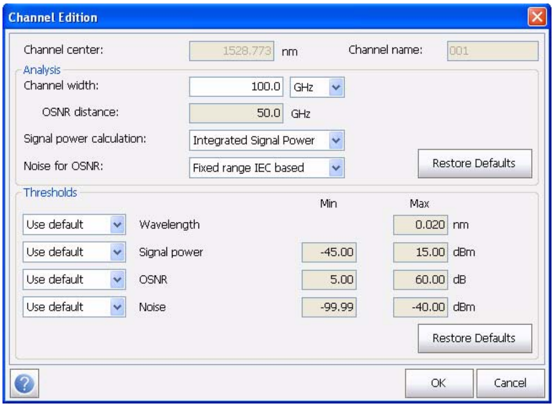
Selected channels have a  in the first column of the channels table.

If you want the changes to be applied to all of your channels, press **Select All**. Channels can be selected one by one or all together. You can press **Unselect All** to clear all channel selections. To delete the selected channels, press **Delete**.

4. Press **Edit Selection**.

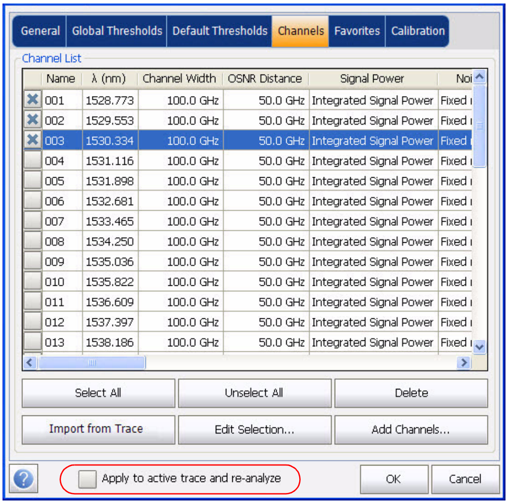


5. Modify the settings as needed. For more information about the settings, see Adding Channels *on page 74*, Changing Default Channel Parameters *on page 309* and Defining Default Thresholds *on page 71*. If you leave a box empty, it will remain as it was before your changes. Modify appropriate settings.



6. Press **OK** to return to the **Channels** tab, which now contains the modified settings.

7. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option in the **Channels** window and press **OK**.

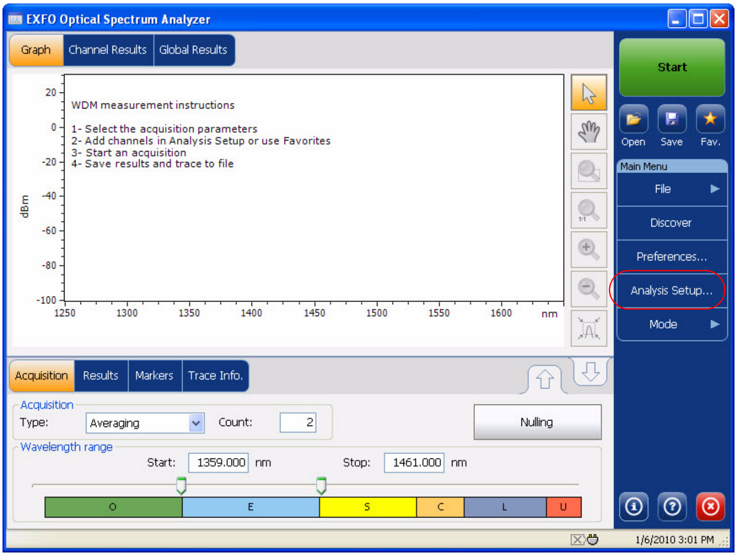


8. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

To add current peaks:

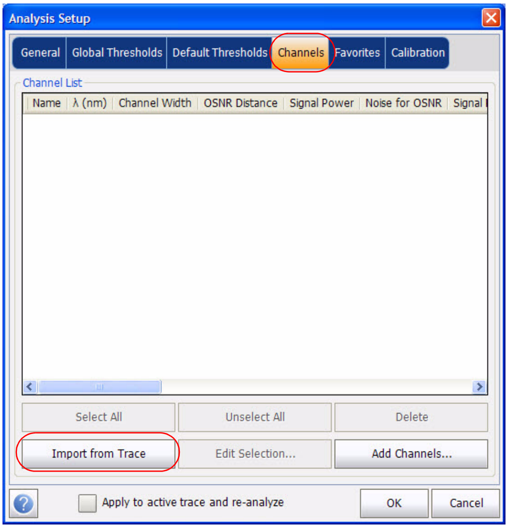
Note: *You can add current peaks to the channel list only if an acquisition has already been performed.*

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.

3. Press **Import from Trace**. All peaks from the current trace will be added to the channel list.

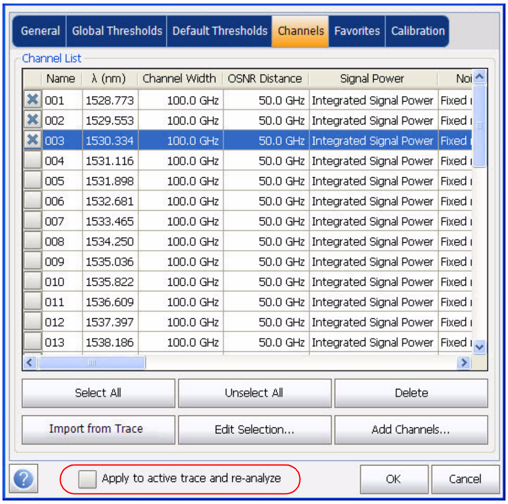


A warning message will be displayed if any channels are overlapping. Press **OK** to close the warning window.

Note: *Changes can be applied to any channel at any time.*

Note: *If some channels were already present in the channel list, the new channels created with the Add Current Peaks button will be added to the list.*

4. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Managing Favorites

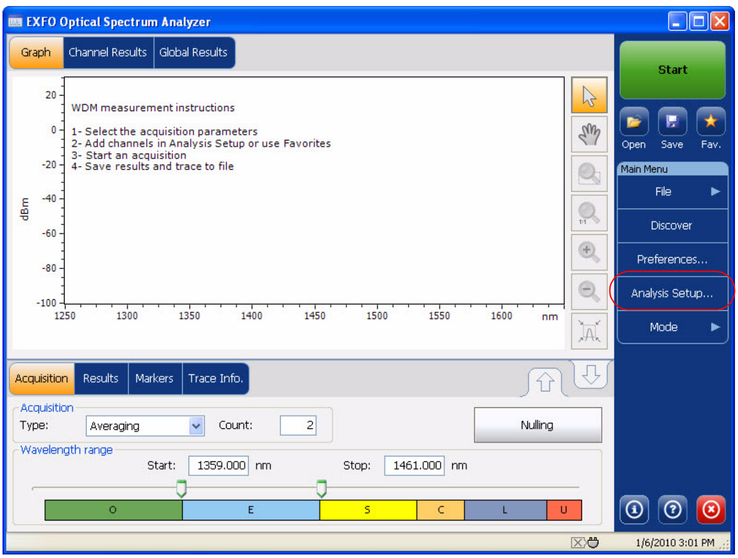
Favorites are configurations files that contain all of the parameters from the **Analysis Setup** tab and **Acquisition** tab. When you often use the same settings, you can save them as a favorite, then recall them for future acquisitions.

To apply a test configuration to the current acquisition:

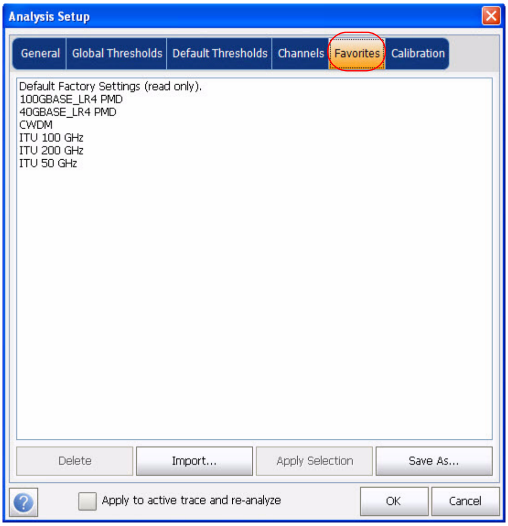
1. From the **Main Menu**, press **Analysis Setup**.

OR

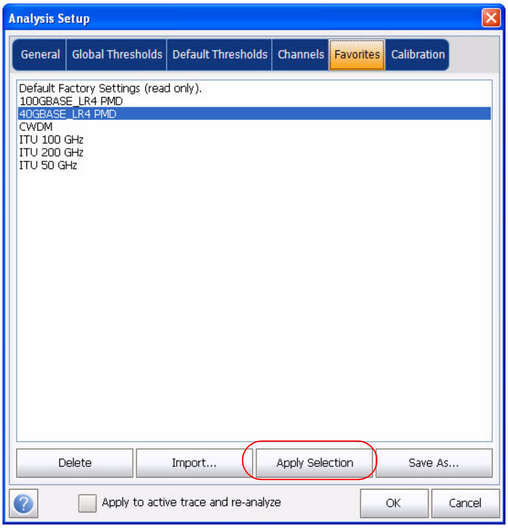
From the main window, press .



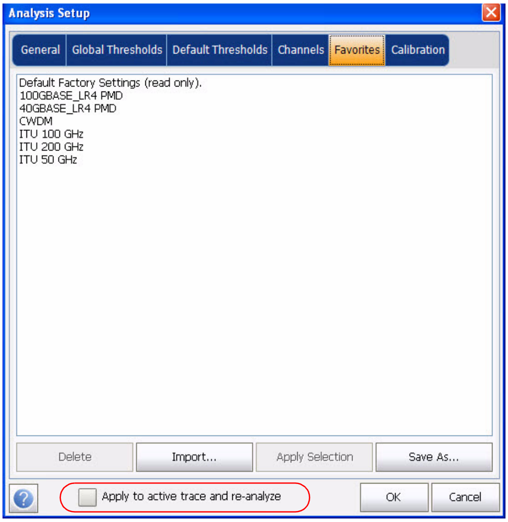
2. Select the **Favorites** tab.



3. To apply the settings from a favorite file to the current analysis setup, select a file from the favorites list and press **Apply Selection**. The apply selection button will be enabled only when a file is selected from the favorites list. When the **Apply Selection** button is pressed, the content of the file will be loaded in the other tabs of this page. To make these parameters effective for the next acquisition you need to press the **OK** button.



4. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



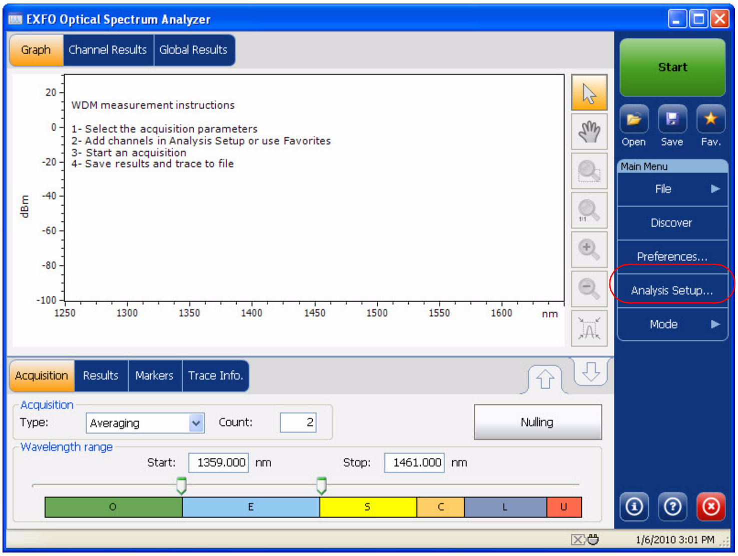
5. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To save a test configuration:

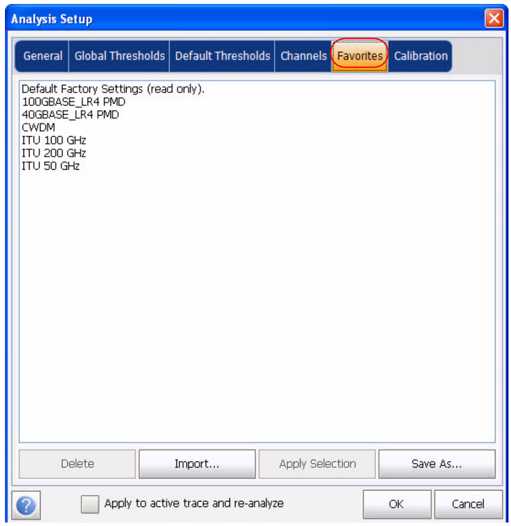
1. From the **Main Menu**, press **Analysis Setup**.

OR

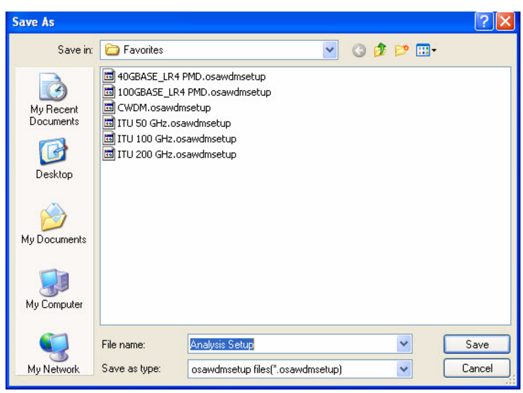
From the main window, press .



2. Select the **Favorites** tab.

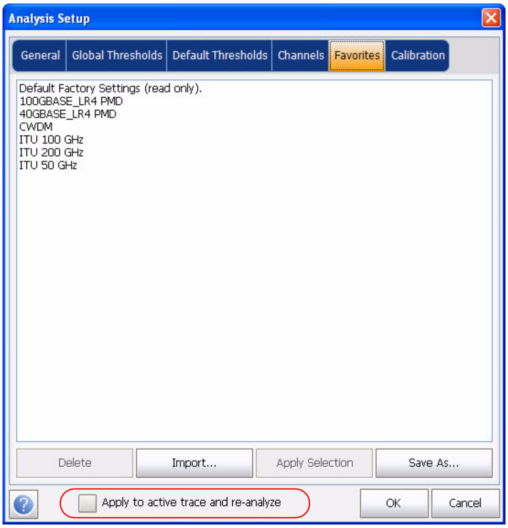


3. To save an analysis setup to a file, press **Save As**. The default directory where the file will be saved is the favorites folder. You should use this folder unless you save the file on a USB stick.



4. In the **Save As** window, enter a file name and press **Save**. The file will be added to the favorites list in the **Analysis setup – Favorites** window.

5. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



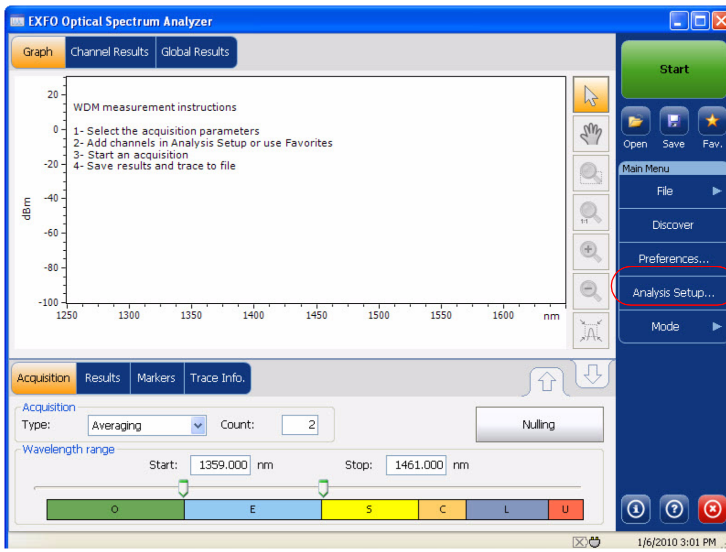
6. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To import a test setup:

1. From the **Main Menu**, press **Analysis Setup**.

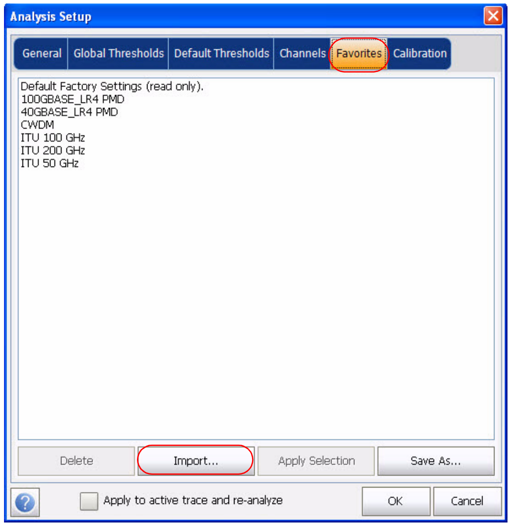
   OR

   From the main window, press .

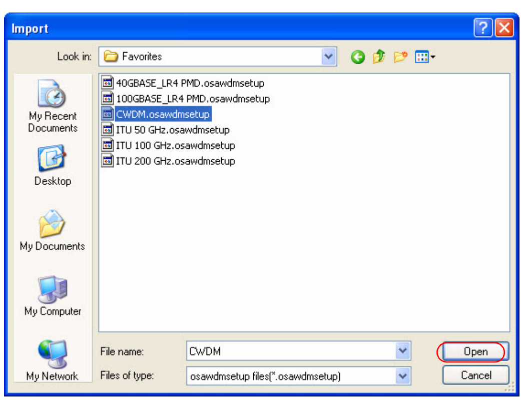


2. Select the **Favorites** tab.

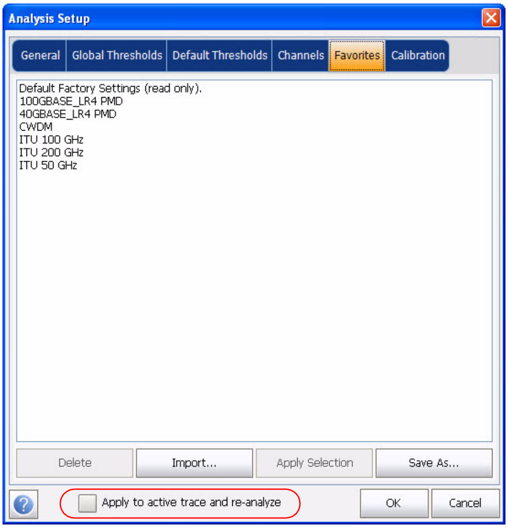
3. Press **Import** to import an analysis setup from a file.



4. From the Import window, select the file you want to import and press **Open**. The file will be added to the favorites list in the **Analysis setup – Favorites** window.



5. If you want to apply the above settings to the active trace, select the **Apply to active trace and re-analyze** option and press **OK**.



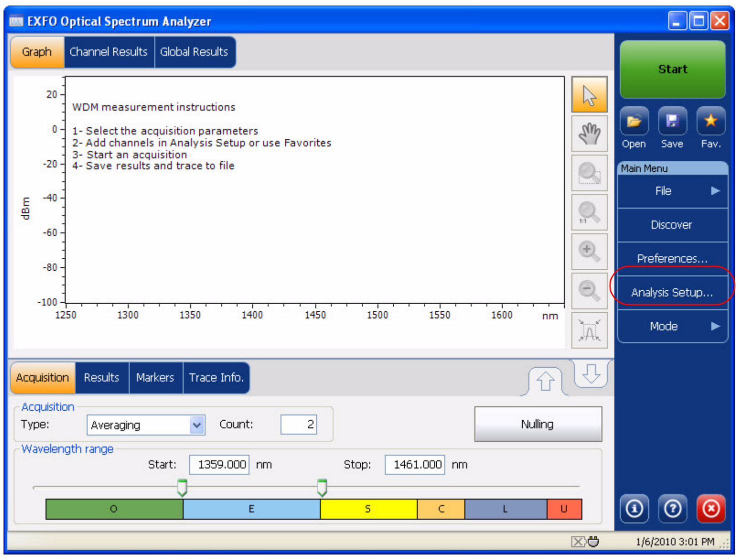
6. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To delete a test configuration:

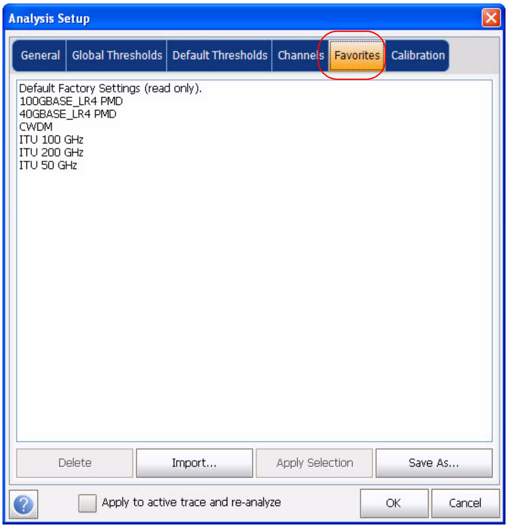
1. From the **Main Menu**, press **Analysis Setup**.

   OR

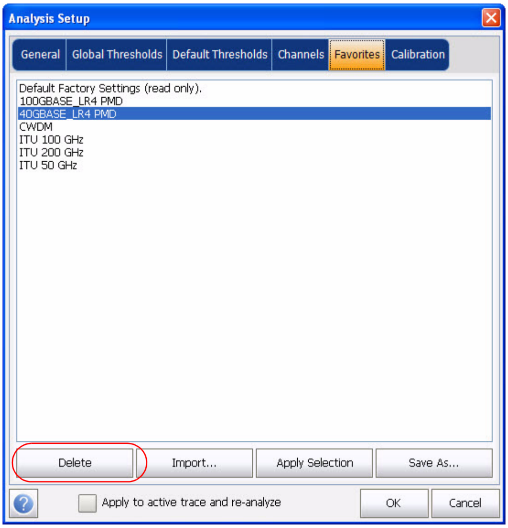
   From the main window, press  .



2. Select the **Favorites** tab.



3. To delete a configuration file from the favorites list, select a file from the favorites list and press **Delete**. Press **Yes** to delete the file, else press **No**.



Setting Up Acquisition Parameters

Before performing your test, you must set the acquisition type and parameters.

There are four types of acquisitions in WDM mode: Single, Averaging, Real-Time and In-Band.

- **Single:** Spectral measurement is performed once. The results are displayed according to this measurement. In single mode, noise measurement can be performed In-Band if the option is available.
- **Averaging:** Spectral measurements are performed based on the number of scans that you have entered for this parameter. The trace will be displayed after each acquisition and averaged with the previous traces.
- **Real-Time**: In Real-Time acquisition, spectral measurements are performed continuously until you press **Stop**. No averaging is done for spectral measurements. The graph and results are refreshed after each acquisition.
- **In-Band**: The In-Band method can be used to measure noise in some or all channels. This mode will perform a series of scans in different polarization conditions in order to accurately calculate the In-Band OSNR measurements for each signal.

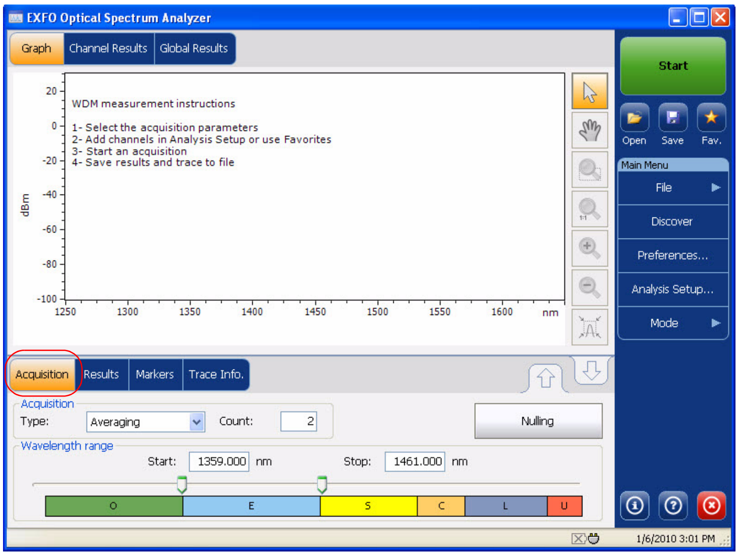
Note: *The In-Band option is available only if the module supports it.*

Before performing measurements on an optical spectrum, you must select the wavelength/frequency range to use. You can perform the scan on the full range, on spectral bands, or select a custom range.

Note: *The shorter the wavelength or frequency range, the faster the acquisition.*

To set parameters in the Acquisition tab:

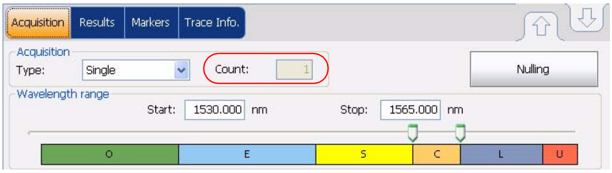
1. From the main window, select the **Acquisition** tab.



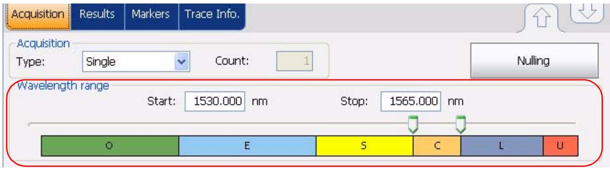
2. Select the acquisition type.

➤ If you are performing a **Single** or **Real-Time** acquisition, you cannot modify the number of scans count value.

➤ If you are performing an **Averaging** or **In-Band** type acquisition, enter the number of scans the unit will perform.



3. Select the wavelength range for your acquisition.



You can select the wavelength range by entering the start and stop values or by selecting a range on the double slider.

To select the wavelength range using the double slider, move the left and right handles on the double slider or simply click on any band.

The wavelength range covered within these bands of the spectra are listed below.

➤ O band (original): 1259 to 1361 nm

➤ E band (extended): 1359 to 1461 nm

➤ S band (short wavelengths): 1459 to 1531 nm

➤ C band (conventional “erbium window”): 1529 to 1566 nm

➤ L band (long wavelengths): 1564 to 1626 nm

➤ U band (ultralong wavelengths): 1624 to 1650 nm.

6 Setting Up the Instrument in Drift Mode

Before performing a spectral analysis in the Drift mode, you must set up your OSA module and the test application with the appropriate parameters, as explained in this chapter.

Select the Drift test mode as explained in *Selecting a Test Mode* on page 18 before setting up the Drift test parameters.

- The *preferences* are the result displayed in the graph and tables, as well as the job information and related comments saved with each file (See *Defining Preferences* on page 105 for more details).
- The *analysis parameters* include the channel list details, pass-fail threshold settings and allows you to select the noise and power calculation methods (See *Setting Up Drift Analysis Parameters* on page 123 for more details).
- The *acquisition parameters* include the type of measurement you want to perform and the wavelength range (See *Setting Up Acquisition Parameters* on page 159 for more details).

You can set up your unit in different manners, depending on your testing needs. The three possible ways for Drift mode are preferred, easy and efficient.

- The preferred way is to use the complete analysis setup parameters and complete the information in all tables, as explained in *Setting Up Drift Analysis Parameters* on page 123. This setup will be used for the next acquisition.

- The easiest way to set up the instrument, especially when the operator does not know in advance what to expect at the input of the OSA is to use the **Discover** button. After **Discover** button has been pressed, a measurement and analysis will be performed according to the best setup determined by the instrument and this setup will be used for the next scan. This is explained in *Using the Discover Feature* on page 269.
- The most efficient way to setup the instrument is to use one of the favorites configurations, uploading a pre-customized acquisition and analysis setup configuration. The operator in the field only has to press the star button, select the appropriate configuration and press **Start**. As an example, a pre-customized configuration could be: “32 channels DWDM 50GHz”; “Toronto-Montreal CWDM” or “Vendor ABC DWDM ROADM 40Gb”. This is explained in *Managing Favorites* on page 147.

Defining Preferences

The preferences window allows you to set general information and comments on trace, set display parameters and customize the drift results table. This information is saved with all the traces.

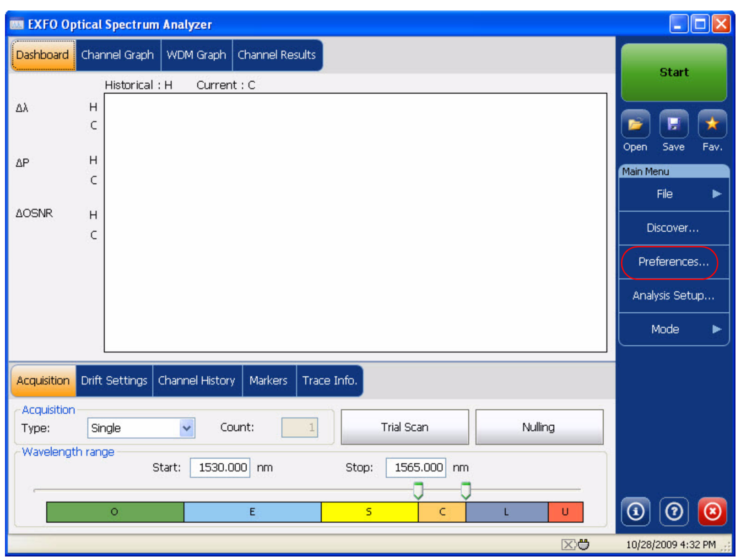
Defining Trace Information

The trace information relates to the description of the job to be done, cable and job IDs, and any relevant information about what is being tested.

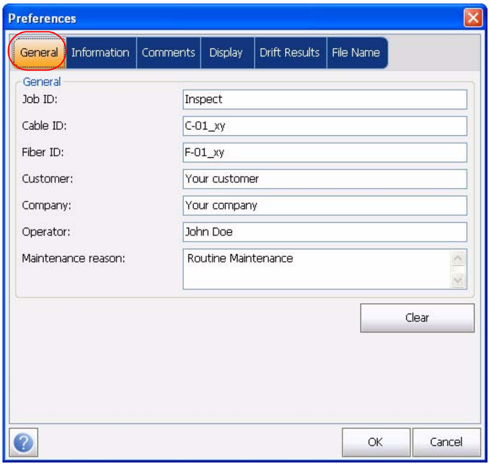
The Link ID is used by the application to propose a file name when you want to save the current acquisition. The link parameters are prefix and suffix values (file names) for the link IDs.

To enter general information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **General** tab.



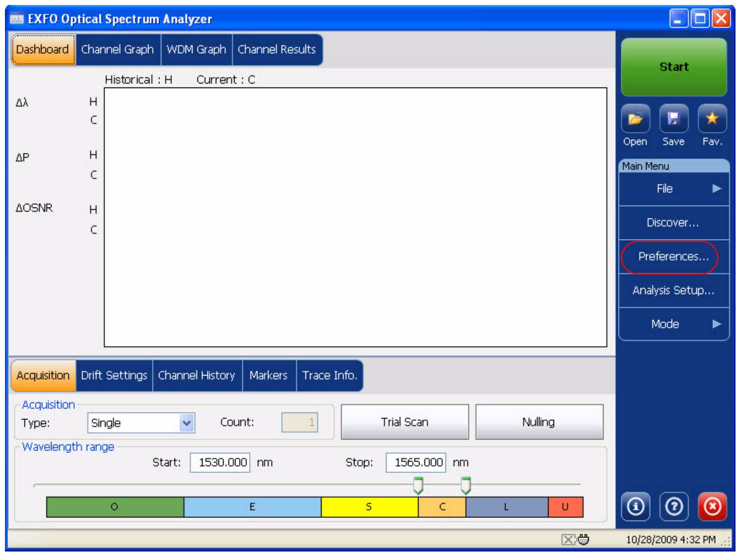
3. Define the general parameters as needed.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

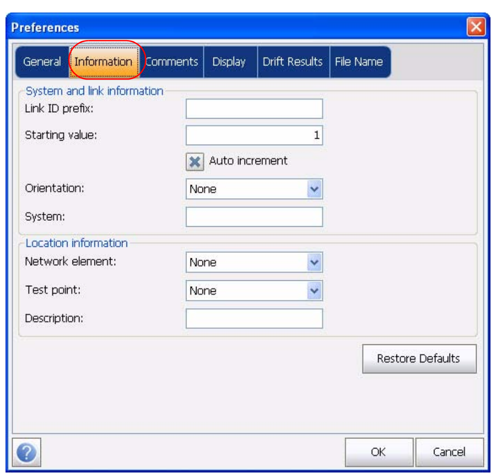
Press **Clear** to clear all the changes made in the **General** tab.

To enter link and location information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Information** tab.



3. Under **System and link information**, define the following parameters as needed:

   ➤ **Link ID prefix**: Sets the prefix value for the link ID. You can enter any alphanumeric value.

   ➤ **Starting value**: Sets the suffix increment starting value for the link ID.

   This value is incremented each time a new file is saved provided the **Auto Increment** option is selected.

IMPORTANT

When the **Auto Increment** option is not selected, while saving the trace file, you have to manually change the file name, else the application will overwrite the previously saved files every time you save a new trace.

   ➤ **Orientation**: Sets the orientation of the link.

   ➤ **System**: Sets the system information.

4. Under **Location Information**, define the following parameters as needed:

   ➤ **Network element**: Sets the type of network element.

   ➤ **Test point**: Sets the test point.

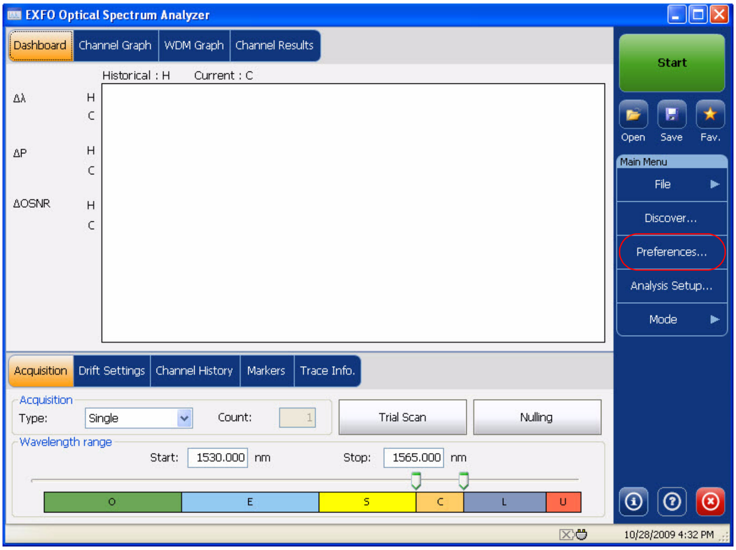
   ➤ **Description**: Enter the description of location if required.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

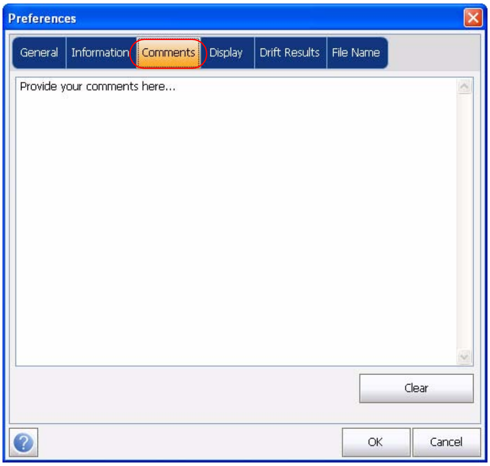
Press **Restore Defaults** to remove all the changes and apply the default values.

To enter comments:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Comments** tab.



3. Enter your comments for the current trace.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

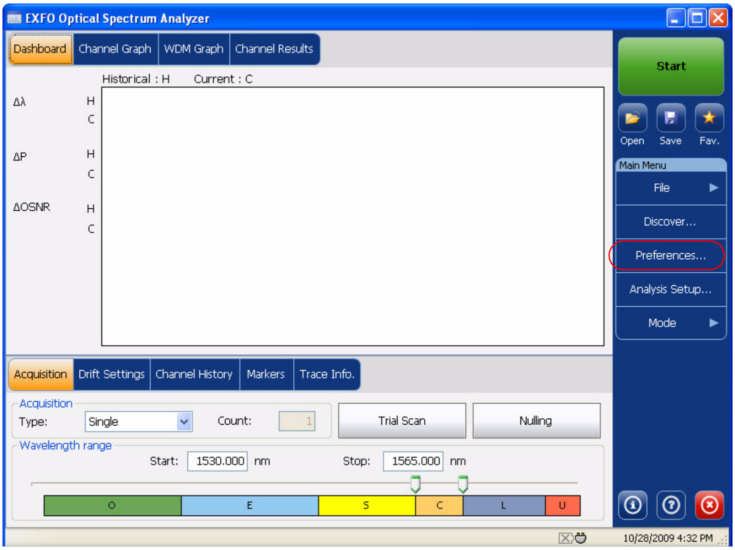
Press **Clear** to clear all the changes made in the **Comments** tab.

Defining Display Parameters

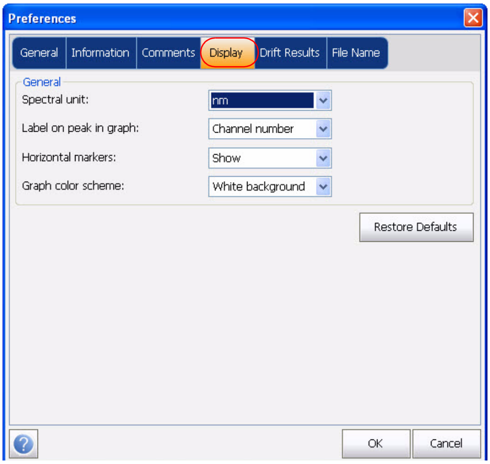
The application allows you to set display settings for the acquisition trace. You can set the spectral unit for the trace and the results table. You can also select the label that should appear on the peaks of the trace.

To define display parameters:

1. From the **Main Menu**, press **Preferences**.

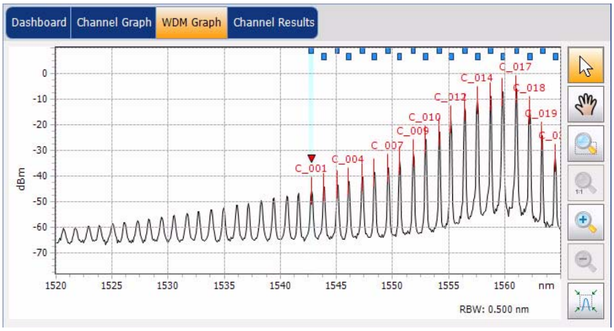


2. Select the **Display** tab.

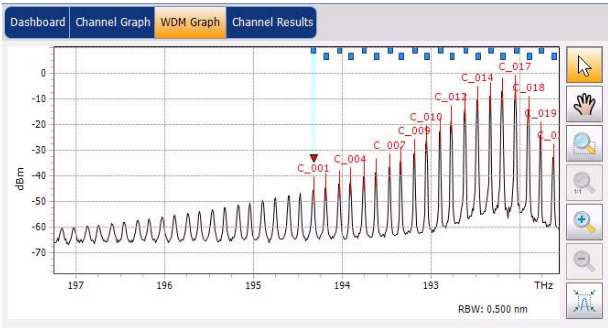


3. Select the spectral unit you want to work with.

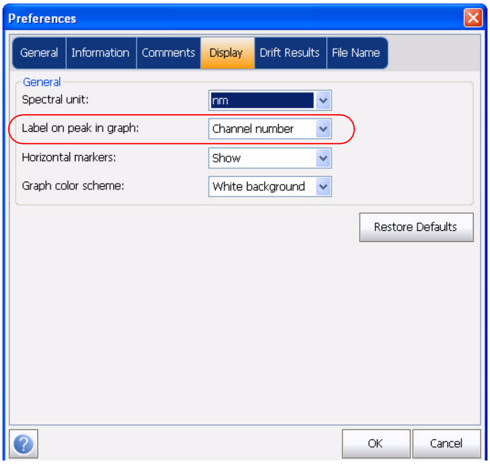
With the nanometer (nm) spectral unit, the trace will appear as shown below:



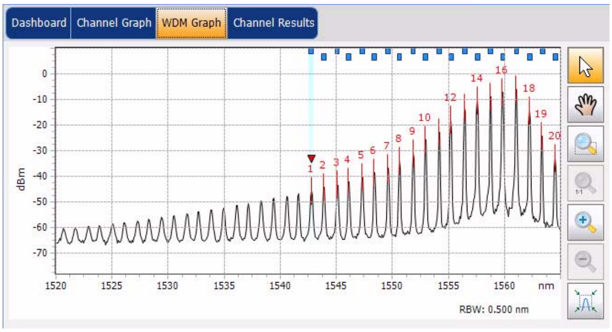
With the terahertz (THz) spectral unit, the trace will appear as shown below:



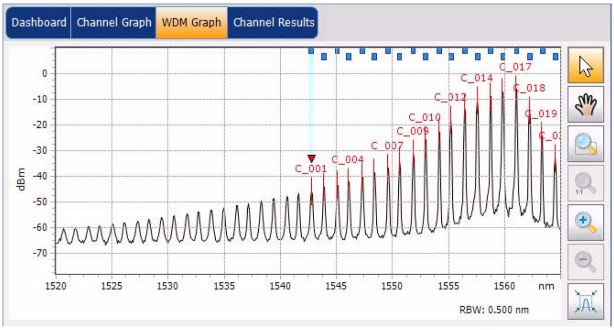
4. Select the label that will appear on the peaks in the graph.



Note: *The channel name and channel number cannot be shown at the same time. If a number is displayed on the peak, it means that the channel name for that peak is not defined. If a channel name is defined for the peak, it will be displayed at the top of the peak.*

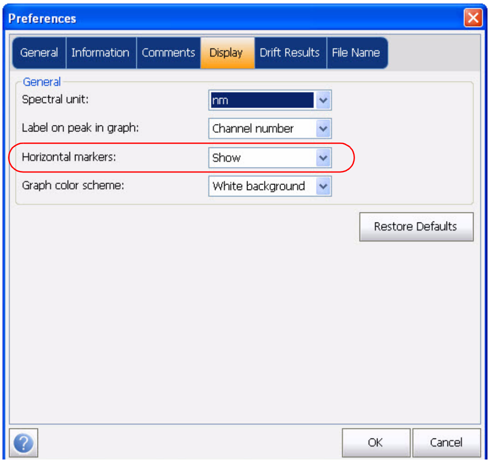


Default channel names

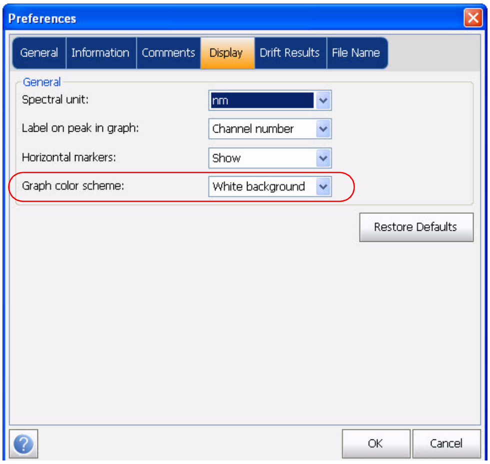


Defined channel names

5. Select **Show** to enable horizontal markers in the **Markers** tab. If **Hide** is selected, horizontal markers will be disabled.



6. Select the background color scheme for Graph as desired.



7. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

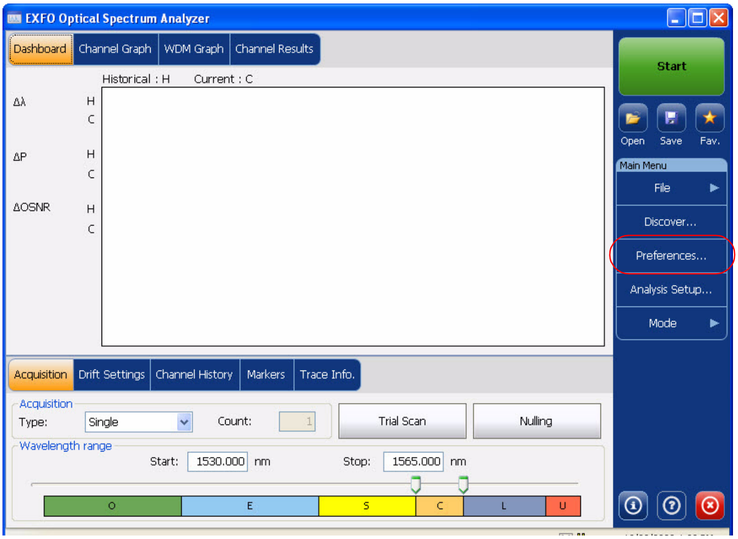
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing Drift Results Table

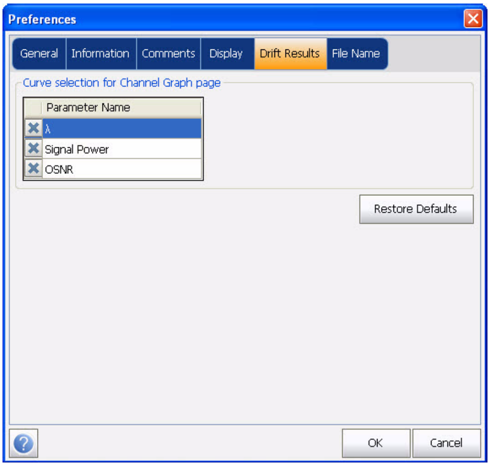
It is possible to select which results you would like to be displayed in the **Results** tab of your Drift tests.

To customize the results table:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Drift Results** tab.



3. Select which parameters you want to display in the **Channel Graph** tab from the list of available choices:

➤ **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.

➤ **Signal Power**: indicates the signal power for the selected channel (excludes noise).

➤ **OSNR**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

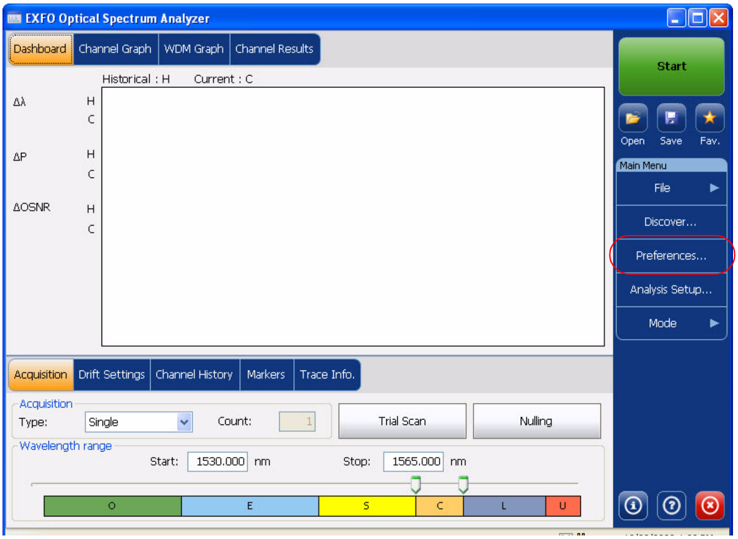
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing Drift File Name

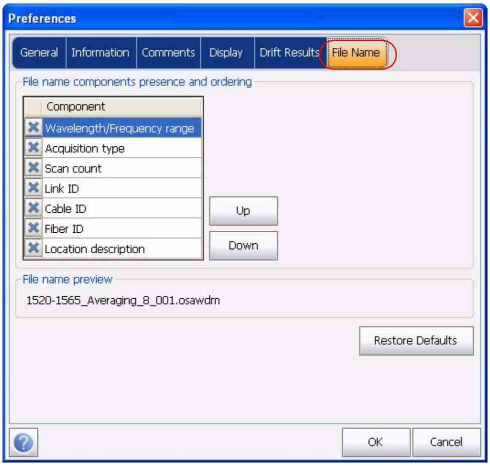
The application shall provide a way to define the name of the next file to be saved. Defining a file autonaming format will allow you to quickly and automatically name traces in a sequential order. The customized name appears when the file is saved using the **Save As** option. It is possible to select which fields you want to include in the file name and the order in which it should be displayed.

To customize the file name:

1. From the **Main Menu**, press **Preferences**.



2. Select the **File Name** tab.

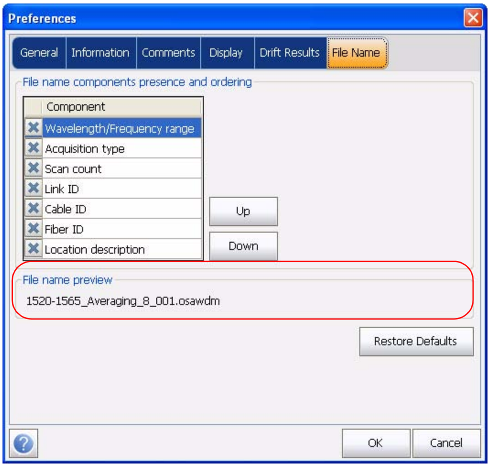


3. Select which parameters you want to include in the file name from the list of available choices:

- **Wavelength/frequency range**: indicates the current wavelength/frequency acquisition range.
- **Acquisition type**: indicates the current acquisition type selected.
- **Scan count**: indicates the current number of scans in the acquisition tab.
- **Link ID**: indicates the prefix value for the link ID configured in the **Preferences-Information** tab.
- **Cable ID**: indicates the prefix value for the cable ID configured in the **Preferences-General** tab.
- **Fiber ID**: indicates the prefix value for the fiber ID configured in the **Preferences-General** tab.
- **Location description**: indicates the location description provided in the **Preferences-Information** tab.

4. Press **Up** or **Down** to change the order in which the field values will appear in the file name.

Based on the fields selected, a preview of the file name shall be displayed under **File name preview**. The field values are separated with an underscore (_).



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default settings.

Setting Up Drift Analysis Parameters

This section presents the various analysis settings for the application, particularly the channel list and settings. These settings are applied on subsequent acquisitions. You can set the channel list, global thresholds, default channel thresholds, channel parameters, manage favorite configurations and perform user calibration.

Note: *The analysis setup parameters will be applied to the global results and channel results, upon the next acquisition. However, you can also apply the modifications to the active trace in order to re-analyze it.*

Defining General Settings

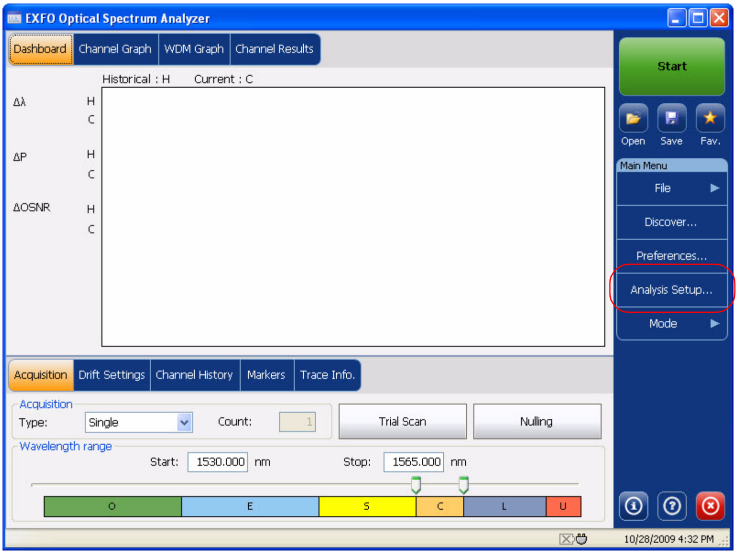
The general analysis parameters for drift acquisitions affect the calculation of the results. These calculations take place after an acquisition. If these settings are modified, they will be applied to the next acquisition.

IMPORTANT

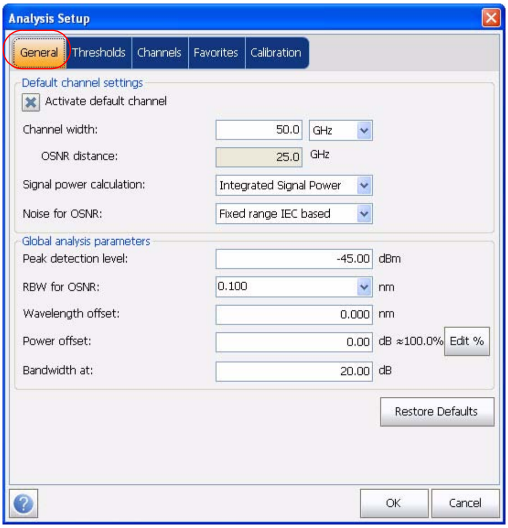
In the General tab, you can set the default channel parameters. Any channel found during an acquisition that is not defined in the channel list will be analyzed according to the default channel settings.

To define general settings:

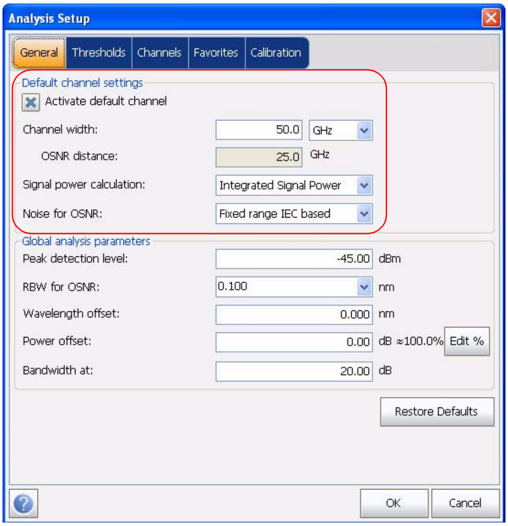
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **General** tab.



3. Under **Default channel settings**, define the following parameters as needed:



➤ Clear the **Activate default channel** selection, to use the currently defined channel for analysis. This reduces the analysis time by eliminating the peak detection over the complete spectral range. The peaks outside the defined channel will not be detected or analyzed.

➤ **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.

For default channels, the channel width that sets the limits of the channel, should be the same as the channel distance or smaller (channel distance is defined while creating a channel list). If the channel width is not compatible with the channel spacing, either a single peak may be found for two distinct channels and two analysis would be performed and displayed for that peak, or, it is possible that two peaks may be found within the same channel and be considered as one multi-peak signal. With this result, you can use markers to find the spacing between adjacent channels or to find the channel width.

➤ **OSNR distance (GHz or nm)**: OSNR distance is automatically set at the channel edge, that is, at half of the channel width from the center wavelength.

➤ **Signal power calculation**: indicates which calculation method to apply for signal power value.

Integrated signal power: The integrated signal power represents the sum of the power values included between the channel limits of this channel, minus the estimated noise contribution between the same boundaries. In some cases, for instance CATV signals, signals with high-frequency modulation, or signals with an inherent line width similar or larger than the OSA's resolution bandwidth, this calculation becomes a better estimation of the true signal power.

Peak signal power: The peak signal power represents the maximum power value inside the channel, but it differs a little from the peak measurement on the spectrum due to the fact that the estimated noise is subtracted to get the peak signal power.

Total channel power: The total channel power represents the sum of the power values included inside the channel width (including noise).

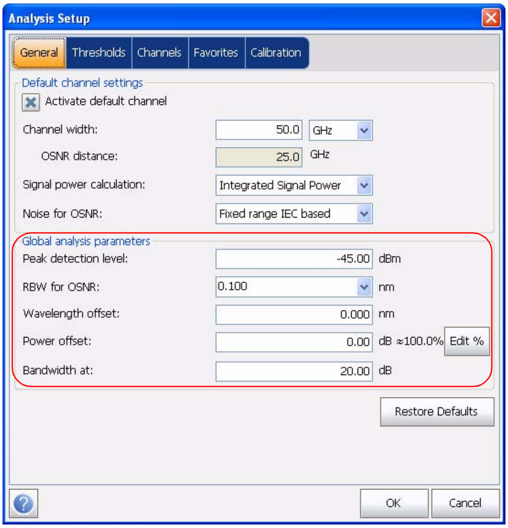
➤ **Noise for OSNR**: indicates which calculation method to use for OSNR value.

Fixed range IEC based (IEC): The IEC method uses the interpolation of noise measured on both sides of the signal to estimate the noise level. The position at which the noise is estimated from the center wavelength is given by the OSNR distance.

In-Band (InB): The In-Band method uses a series of scans having different polarization states to calculate the noise level under the peak (In-Band).

In-Band narrow filter (InB nf): The In-Band narrow filter method uses additional processing to provide an accurate OSNR value for the narrow carved noise. This is because with narrow filters, the noise level under the peak is not uniform and the OSNR value depends on the processing width selected.

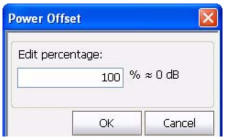
4. Under **Global analysis parameters**, define the following parameters as needed:



- **Peak detection level (dBm)**: indicates the minimum power level from where the peak can be considered as a signal.
- **RBW for OSNR (nm)**: indicates the resolution bandwidth for the selected OSNR value. The resolution bandwidth of an OSA determines its ability to deal with close optical channel spacing. It is measured as the width of the response curve at half peak power (i.e., 3 dB down) of the instrument to a monochromatic test signal. The instrument's RBW value is written below the graph on the right (below wavelength offset).
- **Wavelength offset (nm)**: indicates the offset value applied on the wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.

- **Power offset (dB)**: indicates the offset value applied on the power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. The power offset appears in the graph, on the left, above the arrow.

  To edit the power offset value in percentage, press **Edit %** button.

  The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

- **Bandwidth at (dB)**: Set the power level used, relative to the channel peak power, to compute the bandwidth.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

You can set your pass/fail threshold limits in different ways depending on the type of test you are performing.

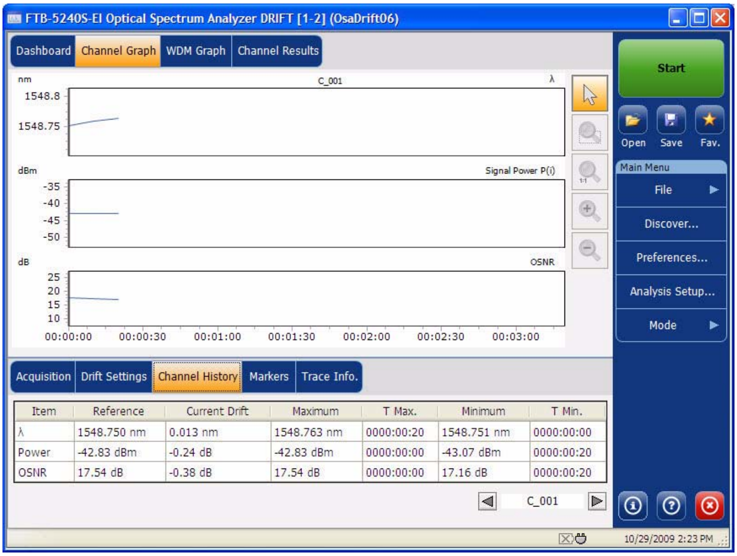
Threshold Limit	Definition
None	No threshold limit is set. The results will be displayed without a Pass/Fail verdict.
Min only	The threshold limit is set for a minimum value only. The Pass/Fail verdict is declared as Pass (in green), when the value is equal to or greater than the minimum threshold set. The verdict is declared as Fail (in red), when the value is below the minimum threshold set.
Max only	The threshold limit is set for a maximum value only. The Pass/Fail verdict is declared as Pass (in green), when the value is equal to or less than the maximum threshold set. The verdict is declared as Fail (in red), when the value is above the maximum threshold set.
Min and Max	The threshold limit is set for the minimum and maximum value. The Pass/Fail verdict is declared as Pass (in green), when the value is equal to or within the minimum and maximum thresholds set. The Pass/Fail verdict is declared as Fail (in red), when the value is beyond the minimum or maximum thresholds set.
Use Default	This threshold limit option is available in the **Channel Parameters** tab of **Channel Results** window only. When this limit is set, the defaults set for the default channels in the **Analysis Setup** tab will be applied to the channel.

Defining Channel Thresholds

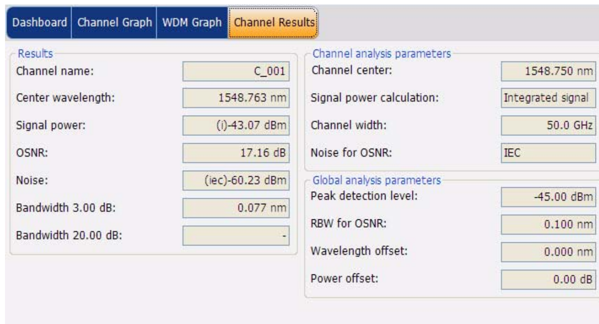
The thresholds will be applied to any channel found outside the channel list during the next acquisition. Thresholds will be applied to the channel results during the next acquisition.

The application allows you to activate and deactivate the threshold functionality with a single control. When thresholds are globally enabled, the results are displayed with the Pass/Fail status based on various settings.

When thresholds are globally disabled, results are displayed without a Pass/Fail status in the Channel Graph and Channel History tabs.

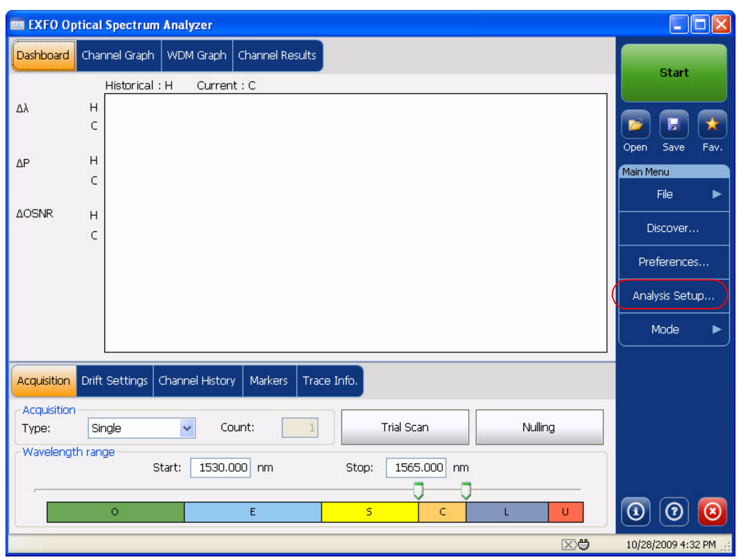


When thresholds are globally disabled, the results in the **Channel Results** tab are also displayed without a Pass/Fail status.

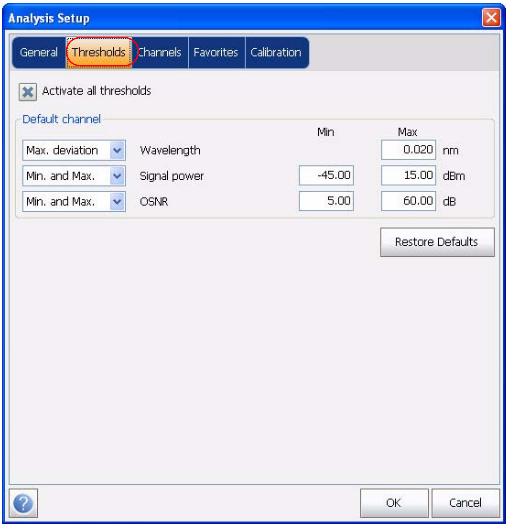


To define channel results thresholds:

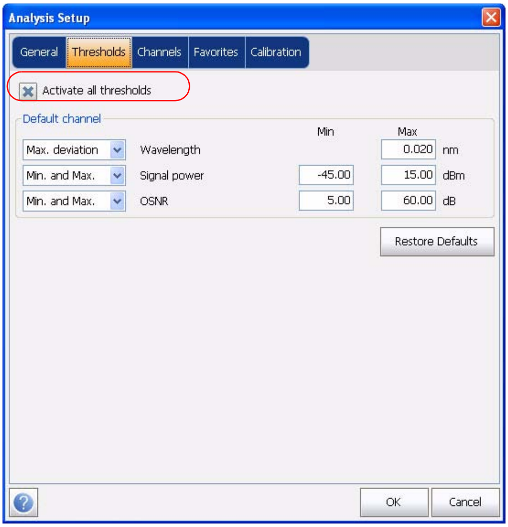
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Thresholds** tab.



3. Select the **Activate all thresholds** option, to manually set the channel threshold values. If this option is not selected, all the thresholds will be deactivated, results are displayed without a Pass/Fail status in the **Channel Graph**, **Channel History** and **Channel Results** tabs.



4. Enter values in the boxes as explained below:
   - **Wavelength/Frequency (nm/GHz)**: indicates the channel's central wavelength/frequency.
   - **Signal power (dBm)**: indicates the signal power for the selected channel (excludes noise).
   - **OSNR (dB)**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Managing Channels

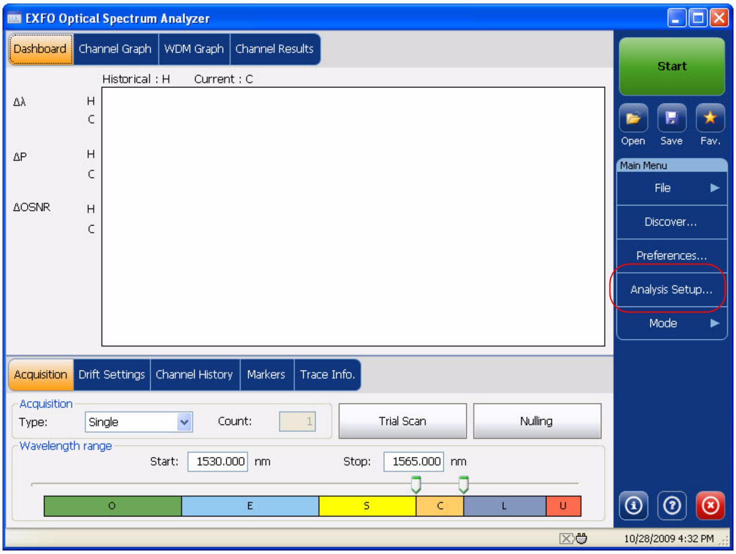
Testing DWDM systems involves characterizing multiple signals in a link. The application allows you to define channels using a channel editor or quickly generate them from the current data. You can also rapidly create a list of equally spaced channels. Once a channel list is created, you can modify it as needed. You can edit the analysis parameters for one channel or multiple channels.

Adding Channels

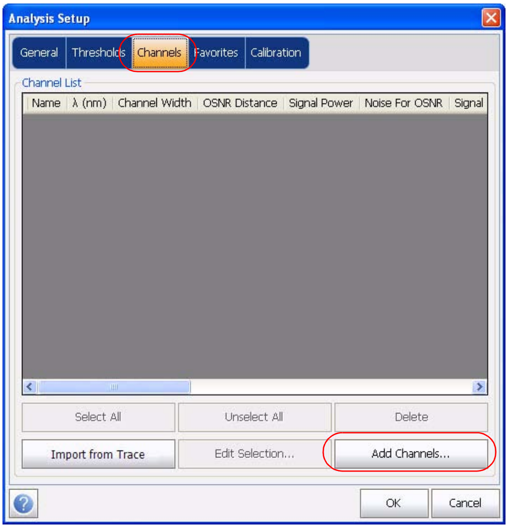
While creating the channel list, some channels may overlap. When the channel widths are specified in nm, two channels are considered to be overlapping when more than 0.001 nm of wavelength range is common between the two channels.

To add a channel list:

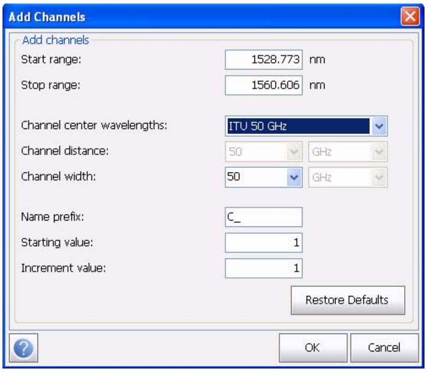
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.
3. By default, the channel list is empty. Press **Add Channels**.



4. Enter values in the boxes as explained below:



➤ **Start range (nm or THz)**: indicates the starting range of the channel list.

- **Stop range (nm or Thz)**: indicates the ending range of the channel list.
- **Channel center wavelength/frequency:** indicates the spectral center-of-mass for the peak in that channel.

Note: *When using the custom channel center wavelength option, the first channel will be centered at the Start Range, and the list will be created using channel distance and channel width.*

- **Channel distance (nm or GHz)**: indicates distance between the channels. The value of channel distance will be set depending on the selection made for the channel center wavelength option. The channel distance box will be enabled only when the channel center wavelength option is set to custom.
- **Channel width (nm or GHz)**: indicates the limit inside which the power values will be considered in the channel. Integrated power is calculated on channel width.
- **Name prefix:** Adds prefix to the channel names.
- **Starting Value:** Sets the increment starting value for the channel name in the channel list.
- **Increment value:** Sets the increment value for the channel name in the channel list.

5. Press **OK** to return to the **Channels** window, which now lists the added channels.

Note: *When new channels are added, the Use Default thresholds will be applied to the channel parameters.*

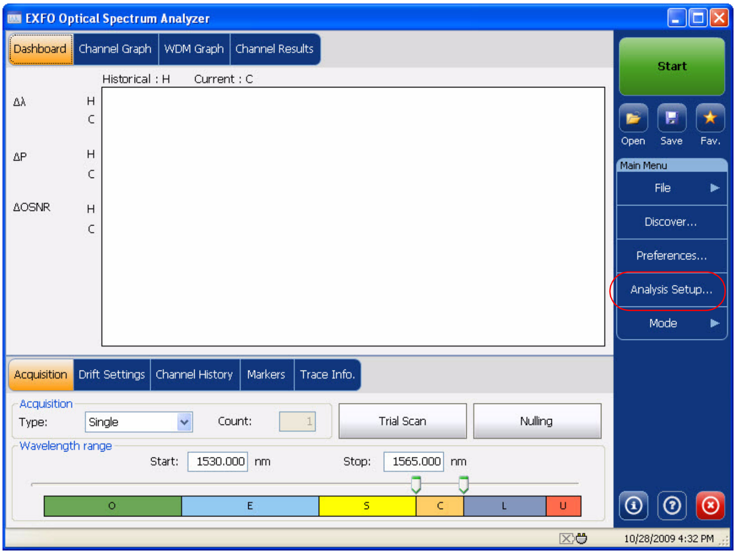
Note: *A warning message will be displayed if any channels are overlapping, but the analysis can still be performed on overlapping channels. If any duplicate channels are added, a confirmation message will be displayed to overwrite the existing channels with the duplicate channels.*

6. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

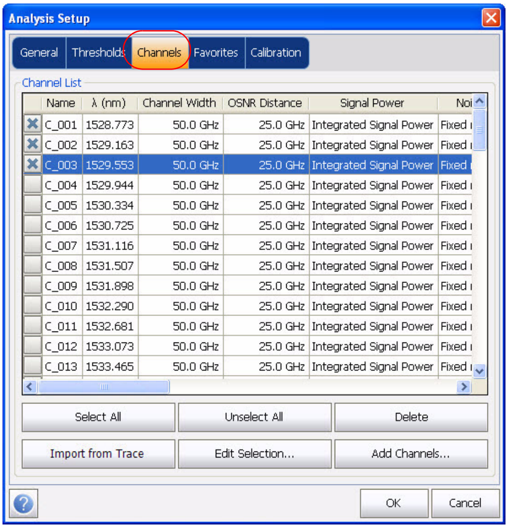
Note: *The application displays a message if more than 1000 channels are added. You can exit the **Analysis Setup** window only after deleting the extra channels from the channel list. You can delete the channels manually as required.*

To edit the parameters of a specific channel:

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.

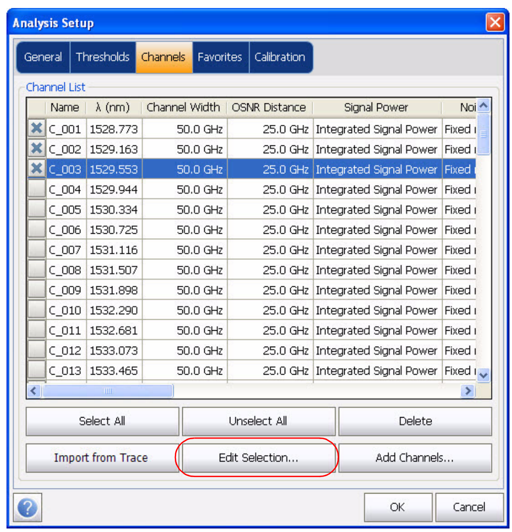


3. Select the channel or channels to be modified in the channel list.

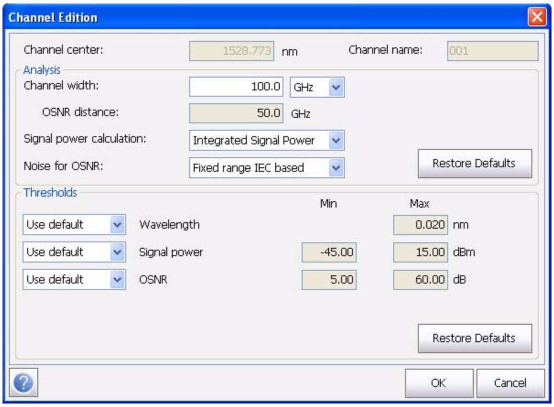
Selected channels have a  in the first column of the channels table.

If you want the changes to be applied to all of your channels, press **Select All**. Channels can be selected one by one or all together. You can press **Unselect All** to clear all channel selections. To delete the selected channels, press **Delete**.

4. Press **Edit Selection**.



5. Modify the settings as needed. For more information about the settings, see Adding Channels *on page 137*, Changing Default Channel Parameters *on page 309* and Defining Channel Thresholds *on page 132*. If you leave a box empty, it will remain as it was before your changes. Modify appropriate settings.



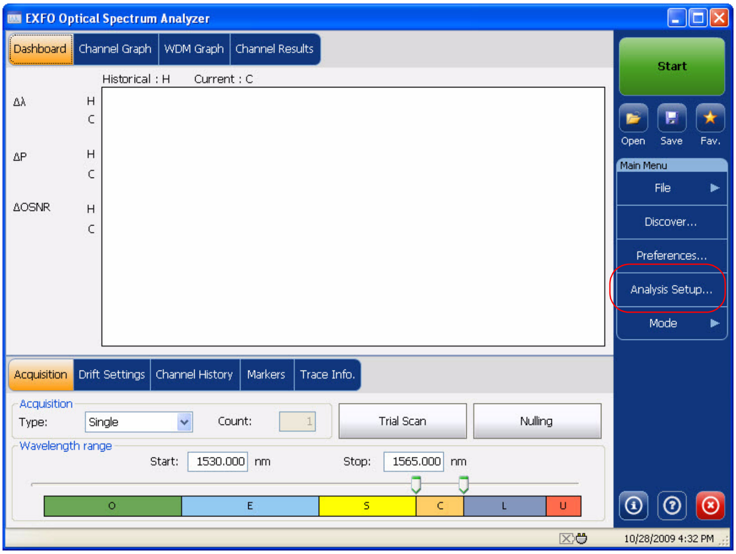
6. Press **OK** to return to the **Channels** tab, which now contains the modified settings.

7. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

To add current peaks:

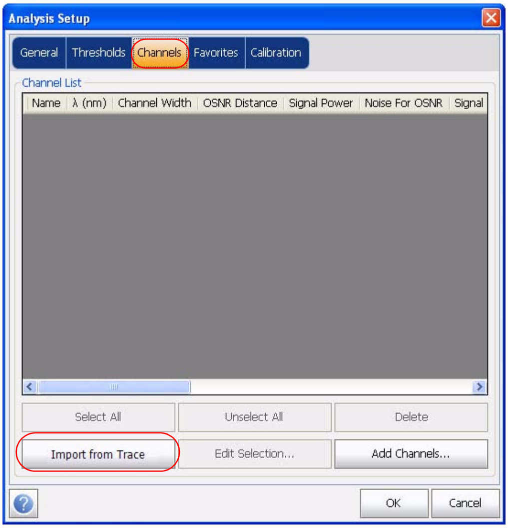
Note: *You can add current peaks to the channel list only if an acquisition has already been performed.*

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.

3. Press **Import from Trace**. All peaks from the current trace will be added to the channel list.



A warning message will be displayed if any channels are overlapping. Press **OK** to close the warning window.

Note: *Changes can be applied to any channel at any time.*

Note: *If some channels were already present in the channel list, the new channels created with the Add Current Peaks button will be added to the list.*

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Managing Favorites

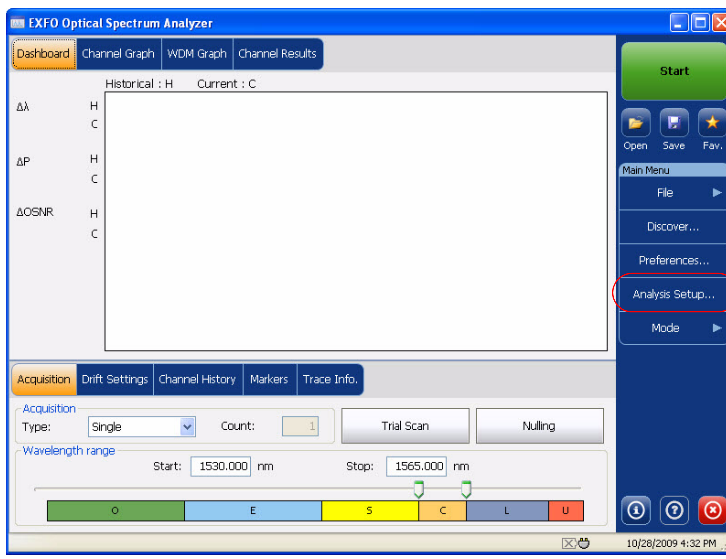
Favorites are configurations files that contain all of the parameters from the **Analysis Setup** tab and **Acquisition** tab. When you often use the same settings, you can save them as a favorite, then recall them for future acquisitions.

To apply a test configuration to the current acquisition:

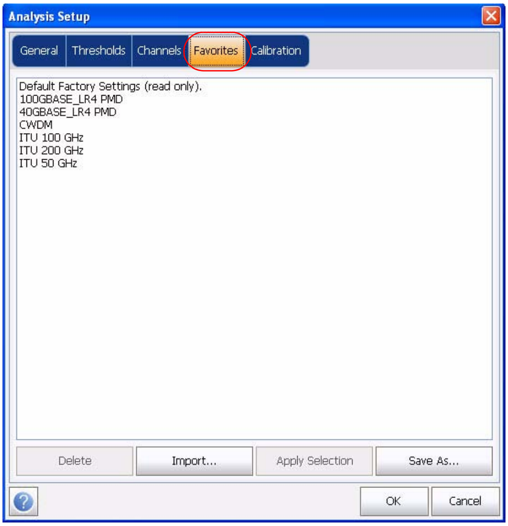
1. From the **Main Menu**, press **Analysis Setup**.

   OR

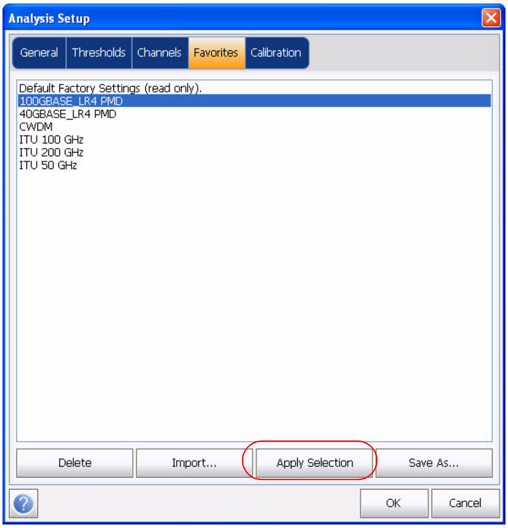
   From the main window, press .



2. Select the **Favorites** tab.



3. To apply the settings from a favorite file to the current analysis setup, select a file from the favorites list and press **Apply Selection**. The apply selection button will be enabled only when a file is selected from the favorites list. When the **Apply Selection** button is pressed, the content of the file will be loaded in the other tabs of this page. To make these parameters effective for the next acquisition you need to press the **OK** button.



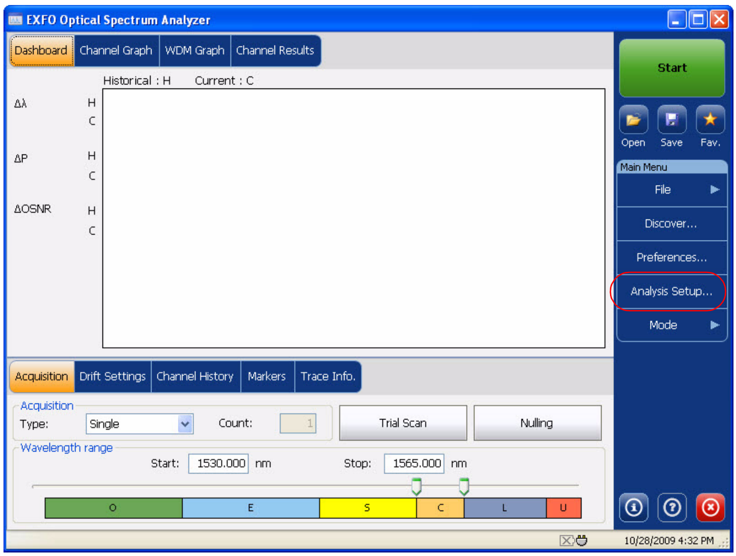
4. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To save a test configuration:

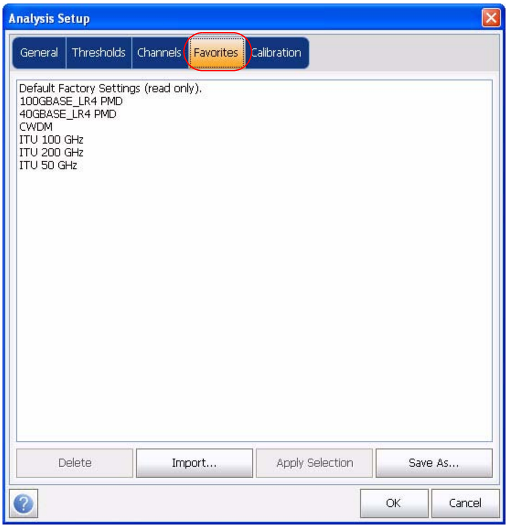
1. From the **Main Menu**, press **Analysis Setup**.

OR

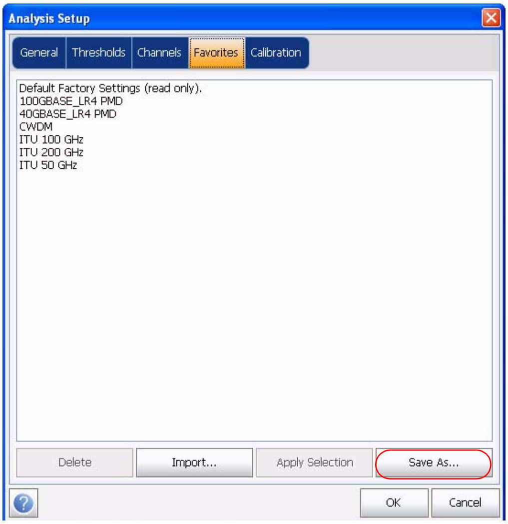
From the main window, press  .



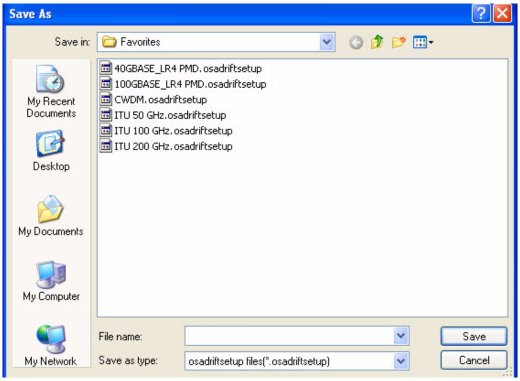
2. Select the **Favorites** tab.



3. To save an analysis setup to a file, press **Save As**. The default directory where the file will be saved is the favorites folder. You should use this folder unless you save the file on a USB stick.



4. In the **Save As** window, enter a file name and press **Save**. The file will be added to the favorites list in the **Analysis setup – Favorites** window.



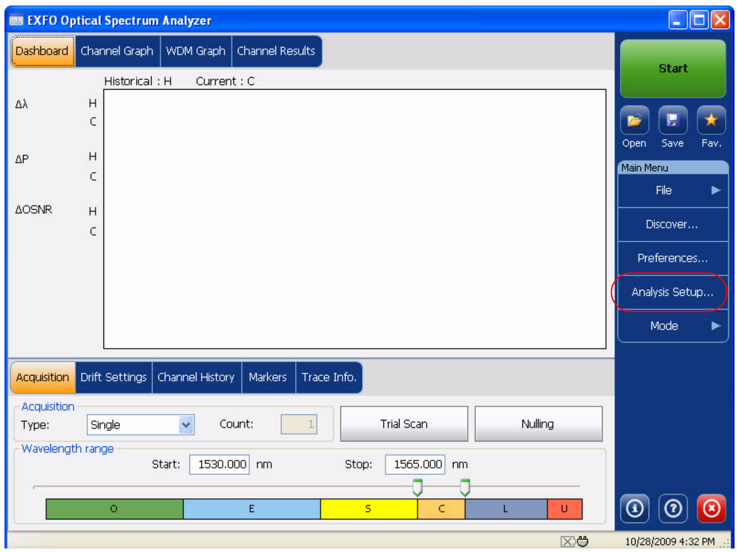
5. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To import a test setup:

1. From the **Main Menu**, press **Analysis Setup**.

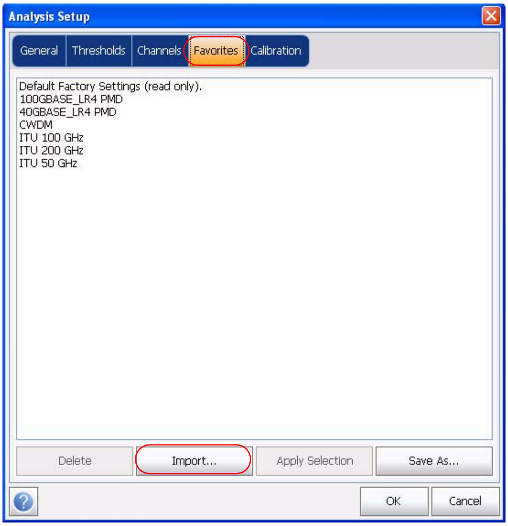
   OR

   From the main window, press .

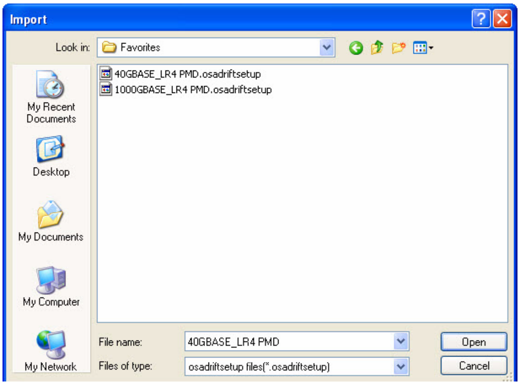


2. Select the **Favorites** tab.

3. Press **Import** to import an analysis setup from a file.



4. From the Import window, select the file you want to import and press **Open**. The file will be added to the favorites list in the **Analysis setup – Favorites** window.



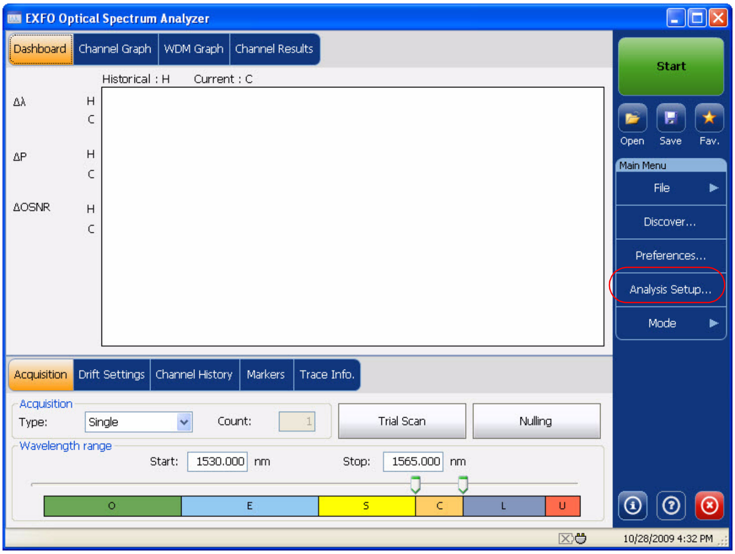
5. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To delete a test configuration:

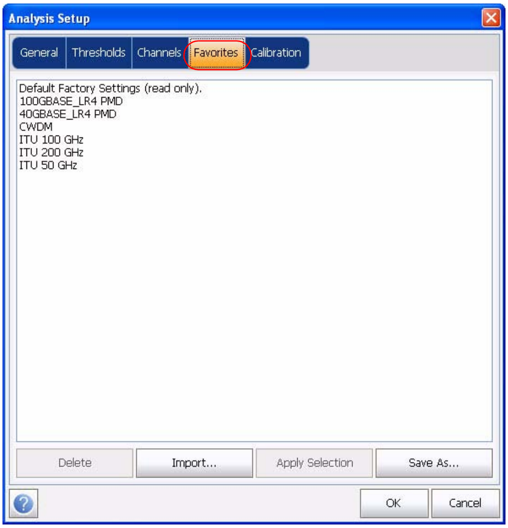
1. From the **Main Menu**, press **Analysis Setup**.

   OR

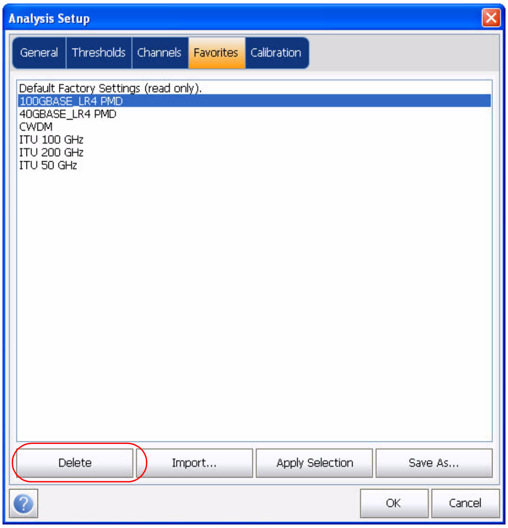
   From the main window, press .



2. Select the **Favorites** tab.



3. To delete a configuration file from the favorites list, select a file from the favorites list and press **Delete**. Press **Yes** to delete the file, else press **No**.



Setting Up Acquisition Parameters

Before performing your test, you must set the acquisition type and parameters from the **Acquisition** tab and other parameters from the **Drift Settings** tab.

There are three types of acquisitions in Drift mode: Single, Averaging and In-Band.

- **Single:** Spectral measurement is performed once. The results are displayed according to this measurement. In single mode, noise measurement can be performed In-Band if the option is available.
- **Averaging:** Spectral measurements are performed based on the number of scans that you have entered for this parameter. The trace will be displayed after each acquisition and averaged with the previous traces.
- **In-Band**: The In-Band method can be used to measure noise in some or all channels. This mode will perform a series of scans in different polarization conditions in order to accurately calculate the In-Band OSNR measurements for each signal.

Note: *The In-Band option is available only if the module supports it.*

The application allows you to perform a trial scan while setting up the drift measurement.

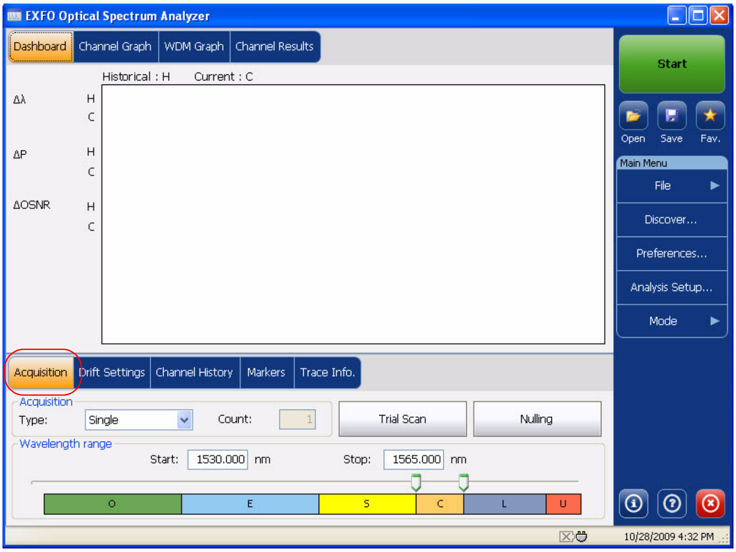
Before performing measurements on an optical spectrum, you must select the wavelength/frequency range to use. You can perform the scan on the full range, on spectral bands, or select a custom range.

Note: *The shorter the wavelength or frequency range, the faster the acquisition.*

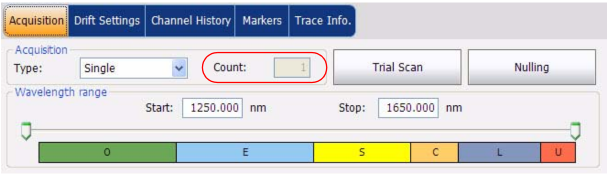
You can configure the **Delay**, **Sampling** and total **Duration** for a drift measurement. You can also configure the drift files name and select a location where it should be saved.

To set parameters in the Acquisition tab:

1. From the main window, select the **Acquisition** tab.



2. Press Trial Scan to perform a trial acquisition.

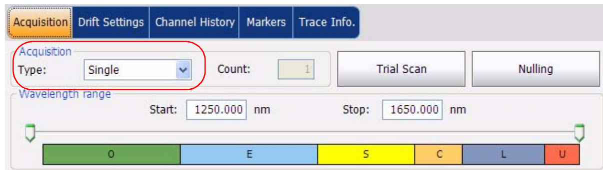


When a trial acquisition is running in the module, the Start button is disabled. You are notified that the acquisition is in progress in the status bar.

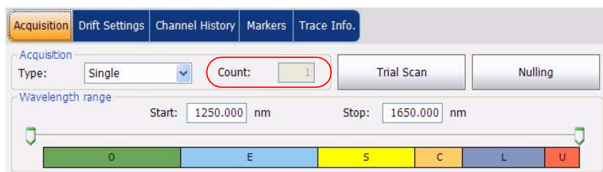
The trial scan is performed using the analysis setup parameters. When the acquisition is complete, the resulting acquisition shall be displayed in the **WDM Graph** tab and **Channel Results** tab. The **Channel History** tab shall display results as if only time 0:00 was available. The other drift mode tabs shall be empty (**Dashboard**, **Channel Graph**).

3. Select the acquisition type.

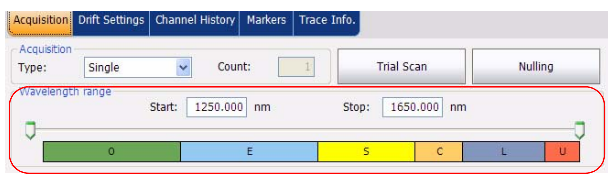
➤ If you are performing a **Single** acquisition, you cannot modify the number of scans count value.



➤ If you are performing an **Averaging** or **In-Band** type acquisition, enter the number of scans the unit will perform.



4. Select the wavelength range for your acquisition.



You can select the wavelength range by entering the start and stop values or by selecting a range on the double slider.

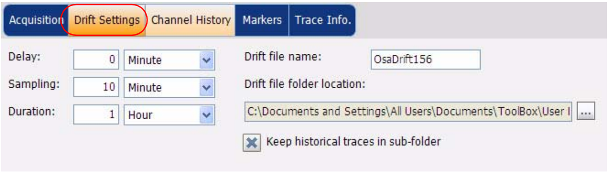
To select the wavelength range using the double slider, move the left and right handles on the double slider or simply click on any band.

The wavelength range covered within these bands of the spectra are listed below.

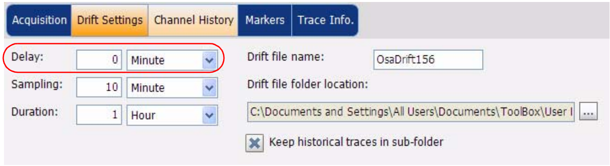
- O band (original): 1259 to 1361 nm
- E band (extended): 1359 to 1461 nm
- S band (short wavelengths): 1459 to 1531 nm
- C band (conventional “erbium window”): 1529 to 1566 nm
- L band (long wavelengths): 1564 to 1626 nm
- U band (ultralong wavelengths): 1624 to 1650 nm.

To set parameters in the Drift Settings tab:

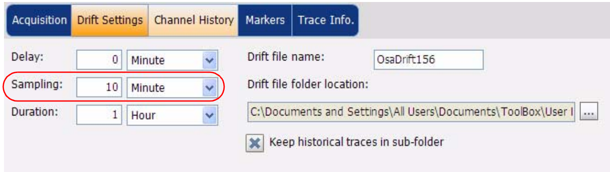
1. From the main window, select the **Drift Settings** tab.



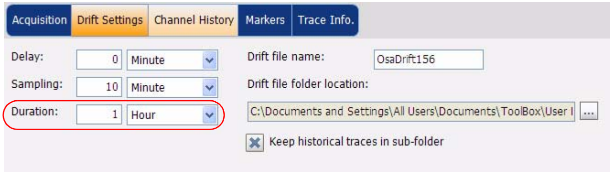
2. Select a delay unit and enter a delay count required to establish a delay before taking the first acquisition in a drift measurement. The application shall wait for this time before taking the first acquisition of a drift measurement.



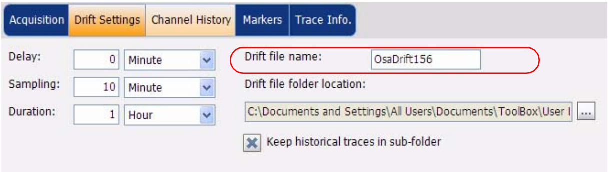
3. Select a sampling unit and enter a sampling count to configure a time that should be elapsed between the start of each acquisition during a drift measurement.



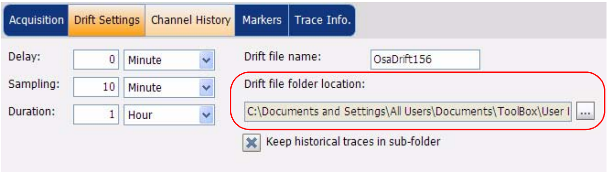
4. Select a duration unit and enter a duration count to configure the total duration of a drift measurement.



5. Enter a drift file name that should be used to save a drift file.



6. Select a location where the drift file should be saved.

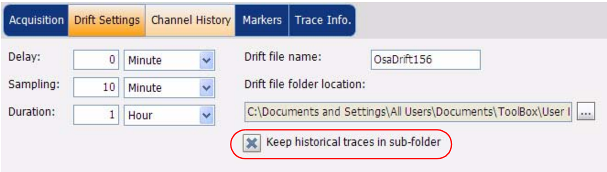


7. Select the **Keep historical traces in subfolder** option to allow the storage of current acquisition in a separate *.osawdm file, when a significant event occurs during the drift measurement.

A significant event is when

- a value from a given channel has crossed its threshold (going from pass to fail).
- there is no signal power in a given channel.

These historical files are stored in a dedicated folder having the same name as the associated drift measurement file name.



7 Setting Up the Instrument in DFB Mode

Before performing a spectral analysis in the DFB mode, you must set up your OSA module and the test application with the appropriate parameters, as explained in this chapter.

Select the DFB test mode as explained in *Selecting a Test Mode* on page 18 before setting up the DFB test parameters.

➤ The *preferences* are the result displayed in the graph and tables, as well as the job information and related comments saved with each file (See *Defining Preferences* on page 166 for more details).

➤ The *acquisition parameters* include the type of measurement you want to perform and the wavelength range (See *Setting Up Acquisition Parameters* on page 179 for more details).

Defining Preferences

The preferences window allows you to set general information and comments on trace, set display parameters and customize the WDM results table. This information is saved with all the traces.

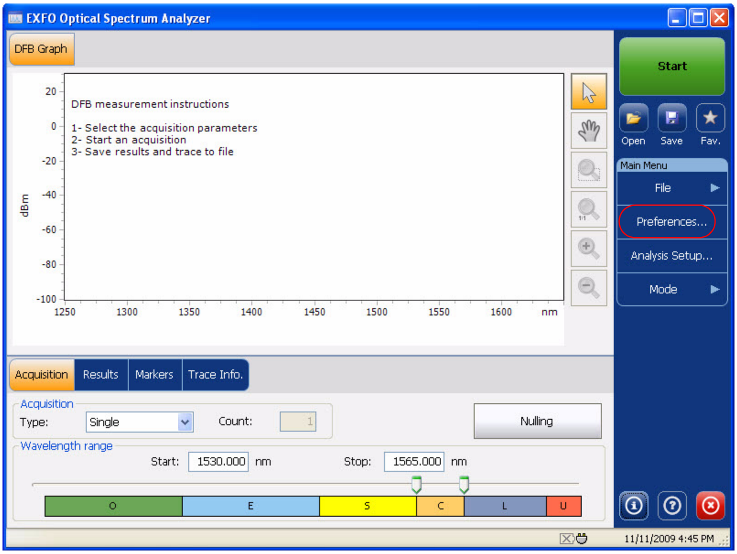
Defining Trace Information

The trace information relates to the description of the job to be done, cable and job IDs, and any relevant information about what is being tested.

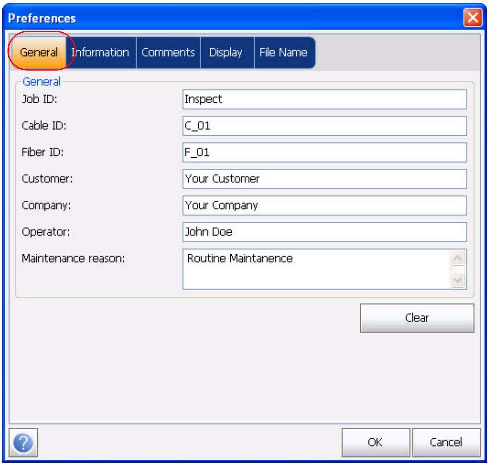
The Link ID is used by the application to propose a file name when you want to save the current acquisition. The link parameters are prefix and suffix values (file names) for the link IDs.

To enter general information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **General** tab.



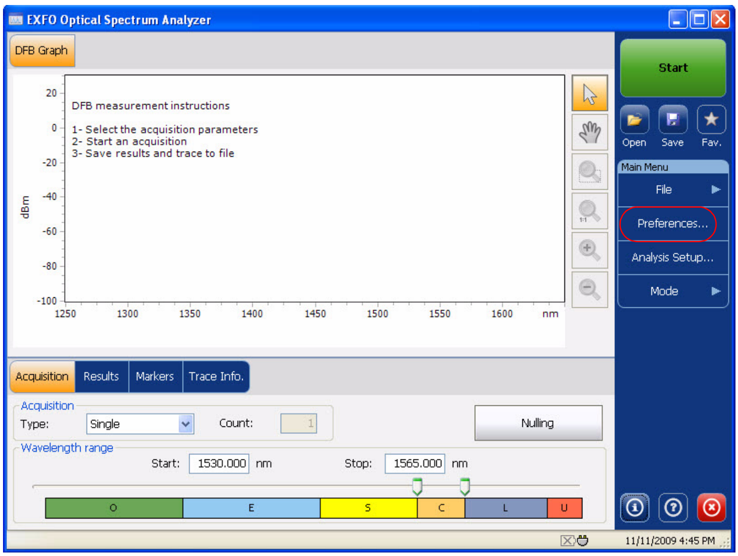
3. Define the general parameters as needed.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

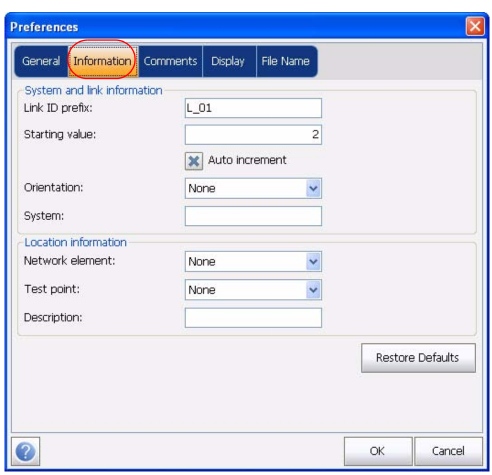
Press **Clear** to clear all the changes made in the **General** tab.

To enter link and location information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Information** tab.



3. Under **System and link information**, define the following parameters as needed:

   ➤ **Link ID prefix**: Sets the prefix value for the link ID. You can enter any alphanumeric value.

   ➤ **Starting value**: Sets the suffix increment starting value for the link ID.

   This value is incremented each time a new file is saved provided the **Auto Increment** option is selected.

IMPORTANT

When the **Auto Increment** option is not selected, while saving the trace file, you have to manually change the file name, else the application will overwrite the previously saved files every time you save a new trace.

   ➤ **Orientation**: Sets the orientation of the link.

   ➤ **System**: Sets the system information.

4. Under **Location Information**, define the following parameters as needed:

   ➤ **Network element**: Sets the type of network element.

   ➤ **Test point**: Sets the test point.

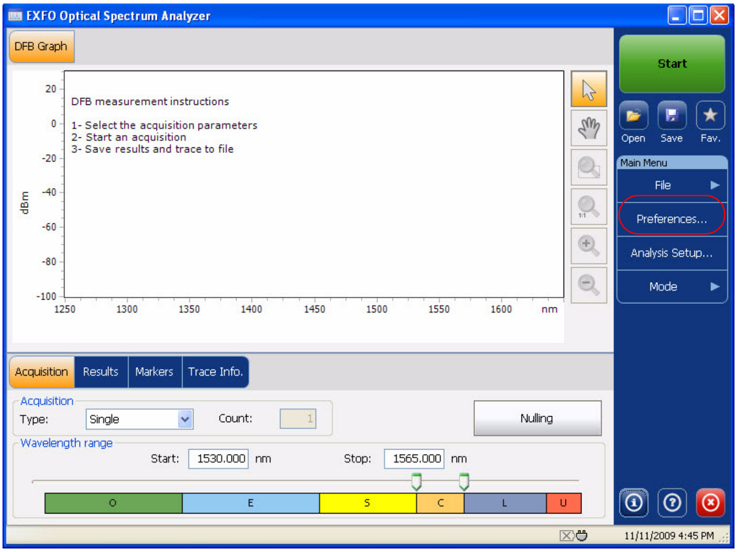
   ➤ **Description**: Enter the description of location if required.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

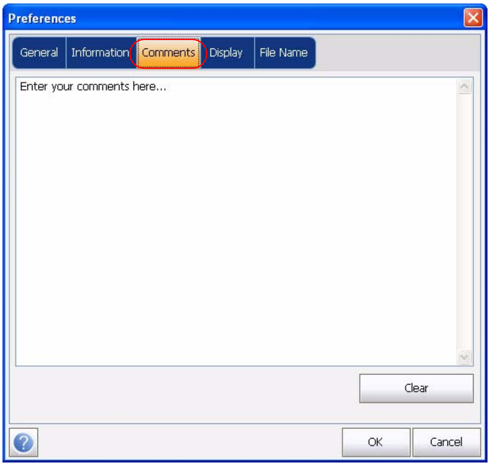
Press **Restore Defaults** to remove all the changes and apply the default values.

To enter comments:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Comments** tab.



3. Enter your comments for the current trace.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

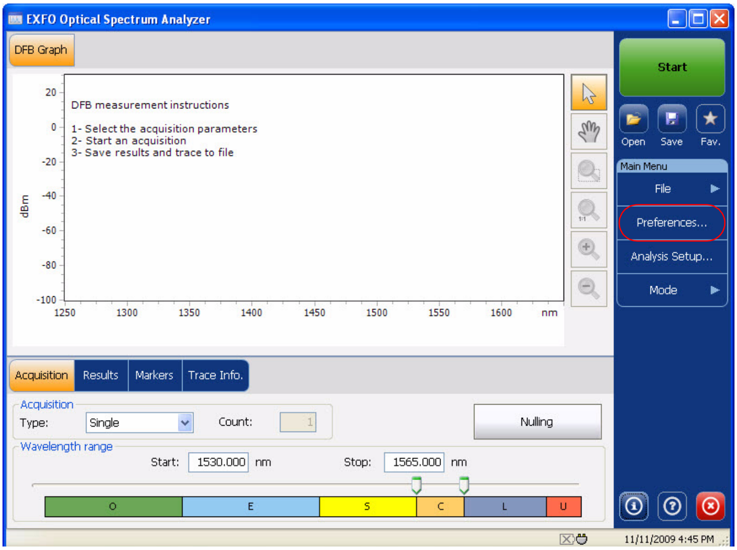
Press **Clear** to clear all the changes made in the **Comments** tab.

Defining Display Parameters

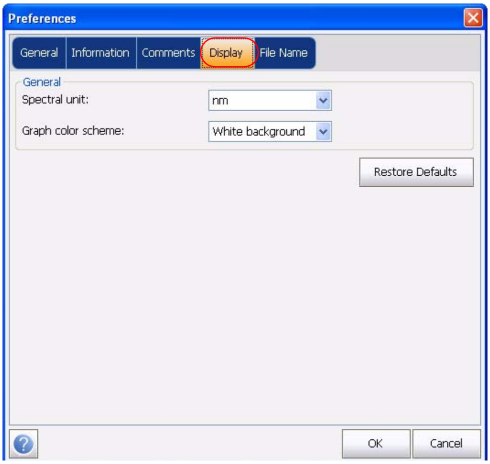
The application allows you to set display settings for the acquisition trace. You can set the spectral unit for the trace and the results table. You can also select the label that should appear on the peaks of the trace.

To define display parameters:

1. From the **Main Menu**, press **Preferences**.

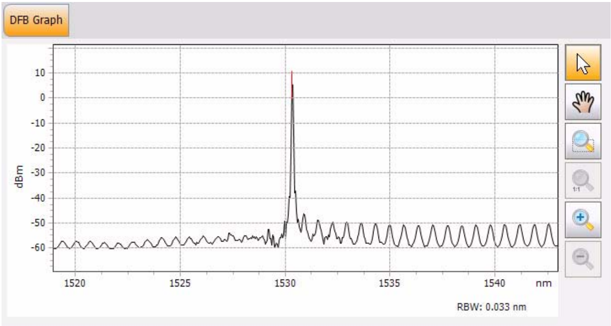


2. Select the **Display** tab.

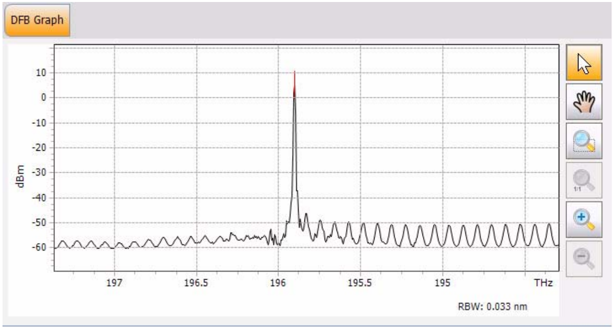


3. Select the spectral unit you want to work with.

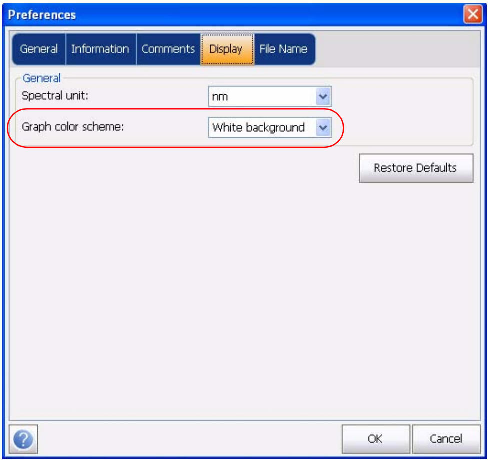
With the nanometer (nm) spectral unit, the trace will appear as shown below:



With the terahertz (THz) spectral unit, the trace will appear as shown below:



4. Select the background color scheme for Graph as desired.



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

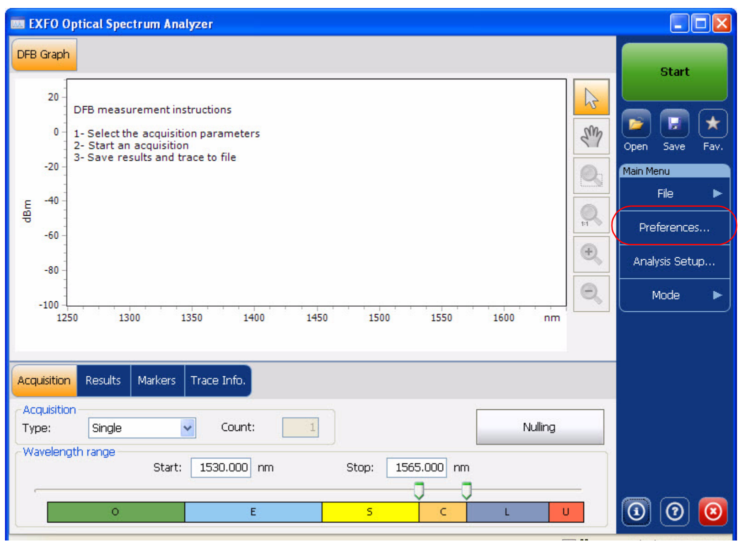
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing DFB File Name

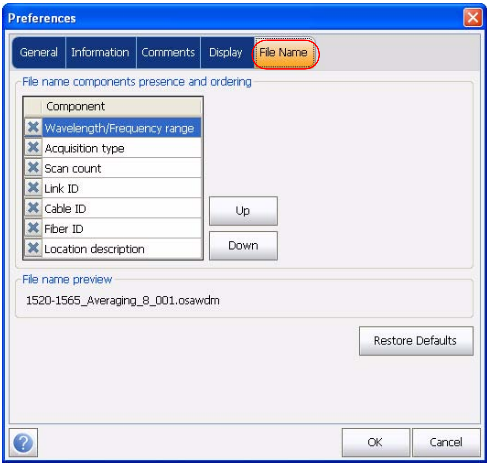
The application shall provide a way to define the name of the next file to be saved. Defining a file autonaming format will allow you to quickly and automatically name traces in a sequential order. The customized name appears when the file is saved using the **Save As** option. It is possible to select which fields you want to include in the file name and the order in which it should be displayed.

To customize the file name:

1. From the **Main Menu**, press **Preferences**.



2. Select the **File Name** tab.

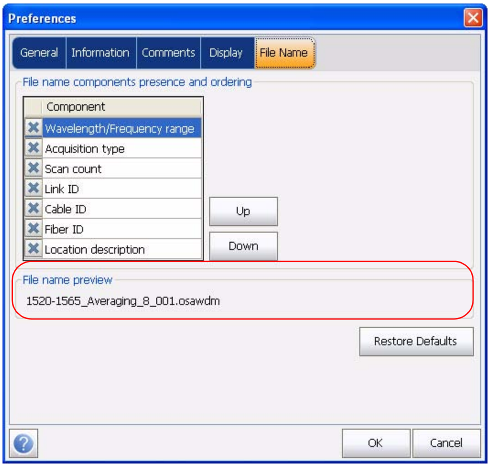


3. Select which parameters you want to include in the file name from the list of available choices:

- **Wavelength/frequency range**: indicates the current wavelength/frequency acquisition range.
- **Acquisition type**: indicates the current acquisition type selected.
- **Scan count**: indicates the current number of scans in the acquisition tab.
- **Link ID**: indicates the prefix value for the link ID configured in the **Preferences-Information** tab.
- **Cable ID**: indicates the prefix value for the cable ID configured in the **Preferences-General** tab.
- **Fiber ID**: indicates the prefix value for the fiber ID configured in the **Preferences-General** tab.
- **Location description**: indicates the location description provided in the **Preferences-Information** tab.

4. Press **Up** or **Down** to change the order in which the field values will appear in the file name.

   Based on the fields selected, a preview of the file name shall be displayed under **File name preview**. The field values are separated with an underscore (_).



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default settings.

Setting Up Acquisition Parameters

Before performing your test, you must set the acquisition type and parameters.

There are three types of acquisitions in DFB mode: Single, Averaging and Real-Time.

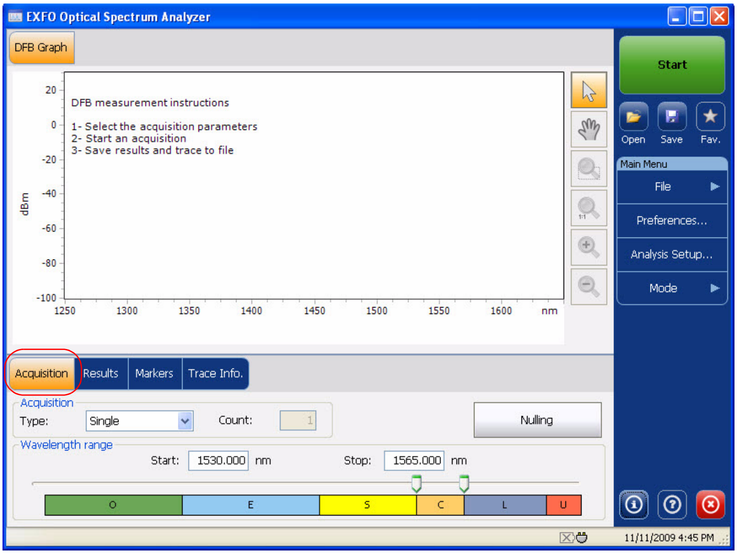
- **Single:** Spectral measurement is performed once. The results are displayed according to this measurement. In single mode, noise measurement can be performed In-Band if the option is available.
- **Averaging:** Spectral measurements are performed based on the number of scans that you have entered for this parameter. The trace will be displayed after each acquisition and averaged with the previous traces.
- **Real-Time**: In Real-Time acquisition, spectral measurements are performed continuously until you press **Stop**. No averaging is done for spectral measurements. The graph and results are refreshed after each acquisition.

Before performing measurements on an optical spectrum, you must select the wavelength/frequency range to use. You can perform the scan on the full range, on spectral bands, or select a custom range.

Note: *The shorter the wavelength or frequency range, the faster the acquisition.*

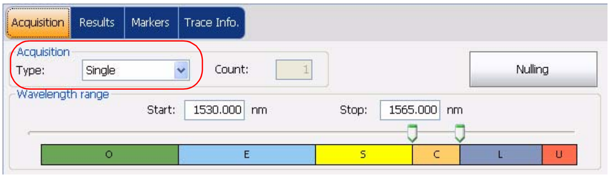
To set parameters in the Acquisition tab:

1. From the main window, select the **Acquisition** tab.

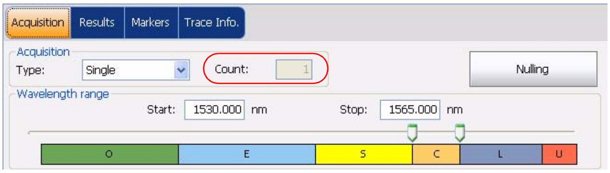


2. Select the acquisition type.

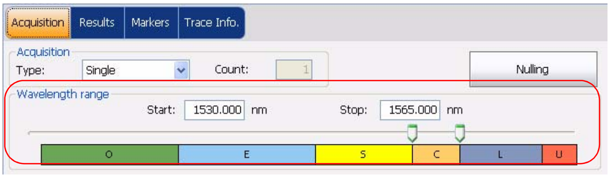
➤ If you are performing a **Single** or **Real-Time** acquisition, you cannot modify the number of scans count value.



➤ If you are performing an **Averaging** type acquisition, enter the number of scans the unit will perform.



3. Select the wavelength range for your acquisition.



You can select the wavelength range by entering the start and stop values or by selecting a range on the double slider.

To select the wavelength range using the double slider, move the left and right handles on the double slider or simply click on any band.

The wavelength range covered within these bands of the spectra are listed below.

➤ O band (original): 1259 to 1361 nm

➤ E band (extended): 1359 to 1461 nm

➤ S band (short wavelengths): 1459 to 1531 nm

➤ C band (conventional “erbium window”): 1529 to 1566 nm

➤ L band (long wavelengths): 1564 to 1626 nm

➤ U band (ultralong wavelengths): 1624 to 1650 nm.

8 Setting Up the Instrument in Spectral Transmittance Mode

Before performing a spectral analysis in the Spectral Transmittance mode, you must set up your OSA module and the test application with the appropriate parameters, as explained in this chapter.

Select the Spectral Transmittance test mode as explained in *Selecting a Test Mode* on page 18 before setting up the test parameters.

- The *preferences* are the result displayed in the graph and tables, as well as the job information and related comments saved with each file (See *Defining Preferences* on page 184 for more details).
- The *analysis parameters* include the channel list details, pass-fail threshold settings and allows you to select the noise and power calculation methods (See *Setting Up Spectral Transmittance Analysis Parameters* on page 198 for more details).
- The *acquisition parameters* include the type of measurement you want to perform and the wavelength range (See *Setting Up Acquisition Parameters* on page 205 for more details).

The preferred way to set up your unit is to use the complete analysis setup parameters and complete the information in all tables, as explained in *Setting Up Spectral Transmittance Analysis Parameters* on page 198. This setup will be used for the next acquisition.

Defining Preferences

The preferences window allows you to set general information and comments on trace, set display parameters and customize the Spectral Transmittance results table. This information is saved with all the traces.

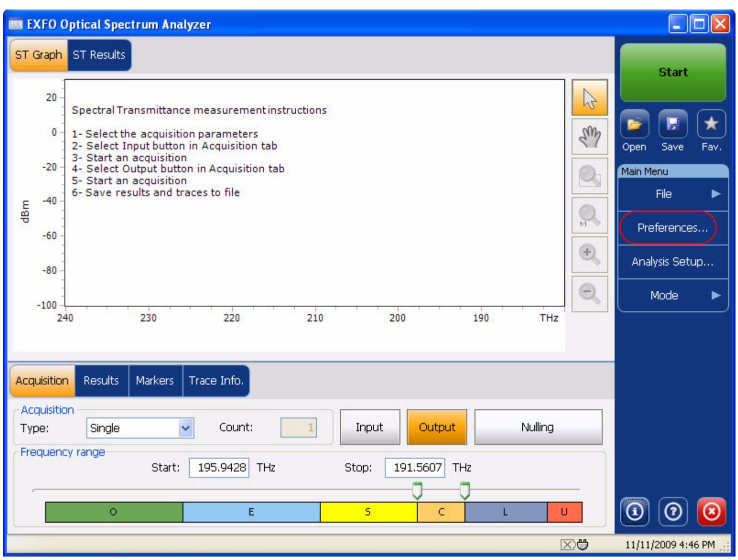
Defining Trace Information

The trace information relates to the description of the job to be done, cable and job IDs, and any relevant information about what is being tested.

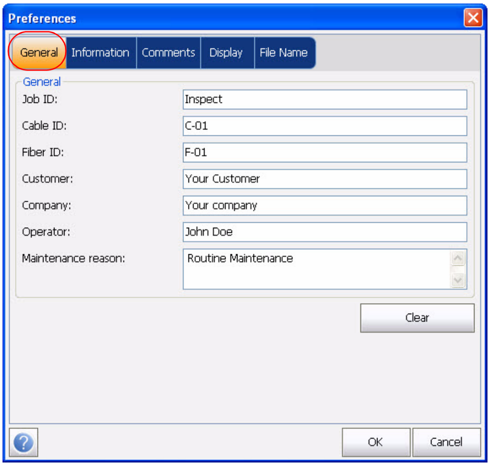
The Link ID is used by the application to propose a file name when you want to save the current acquisition. The link parameters are prefix and suffix values (file names) for the link IDs.

To enter general information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **General** tab.



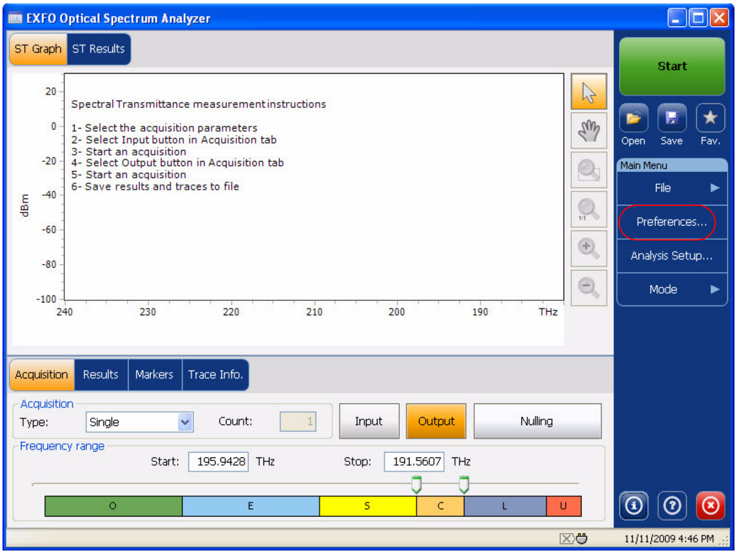
3. Define the general parameters as needed.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

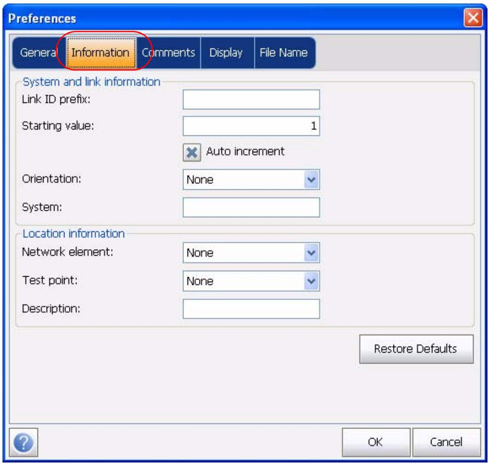
Press **Clear** to clear all the changes made in the **General** tab.

To enter link and location information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Information** tab.



3. Under **System and link information**, define the following parameters as needed:

➤ **Link ID prefix**: Sets the prefix value for the link ID. You can enter any alphanumeric value.

➤ **Starting value**: Sets the suffix increment starting value for the link ID.

This value is incremented each time a new file is saved provided the **Auto Increment** option is selected.

IMPORTANT

When the **Auto Increment** option is not selected, while saving the trace file, you have to manually change the file name, else the application will overwrite the previously saved files every time you save a new trace.

➤ **Orientation**: Sets the orientation of the link.

➤ **System**: Sets the system information.

4. Under **Location Information**, define the following parameters as needed:

➤ **Network element**: Sets the type of network element.

➤ **Test point**: Sets the test point.

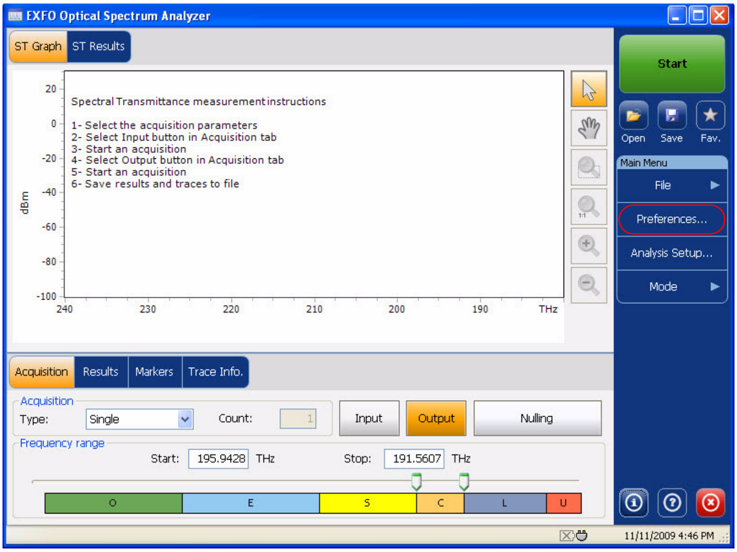
➤ **Description**: Enter the description of location if required.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

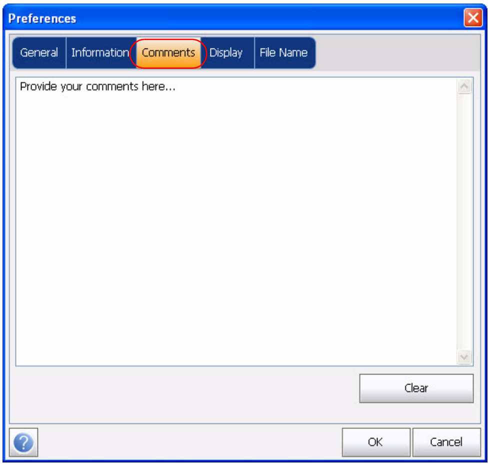
Press **Restore Defaults** to remove all the changes and apply the default values.

To enter comments:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Comments** tab.



3. Enter your comments for the current trace.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

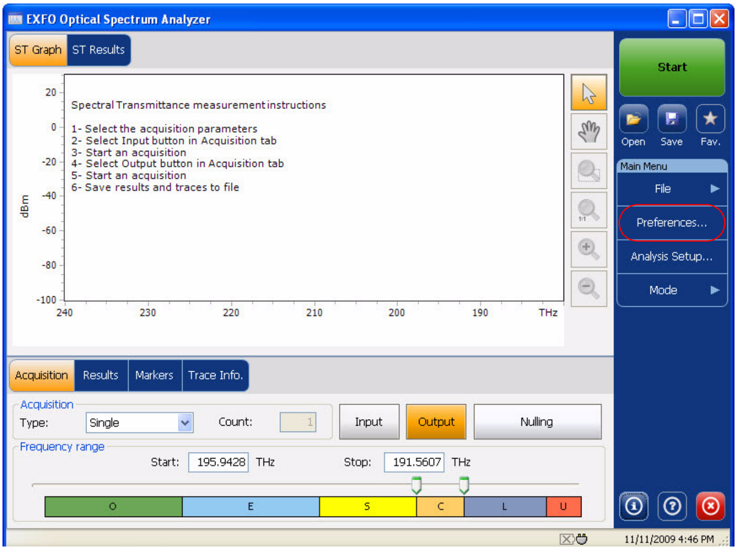
Press **Clear** to clear all the changes made in the **Comments** tab.

Defining Display Parameters

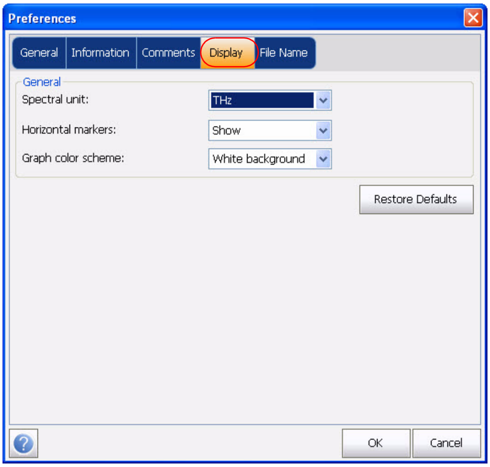
The application allows you to set display settings for the acquisition trace. You can set the spectral unit for the trace and the results table. You can also select the label that should appear on the peaks of the trace.

To define display parameters:

1. From the **Main Menu**, press **Preferences**.

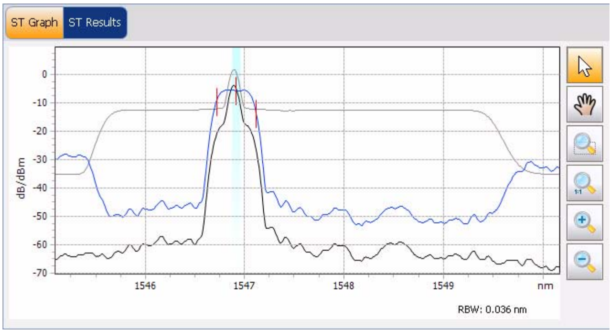


2. Select the **Display** tab.

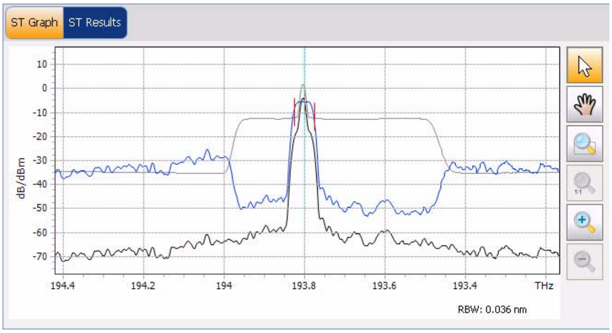


3. Select the spectral unit you want to work with.

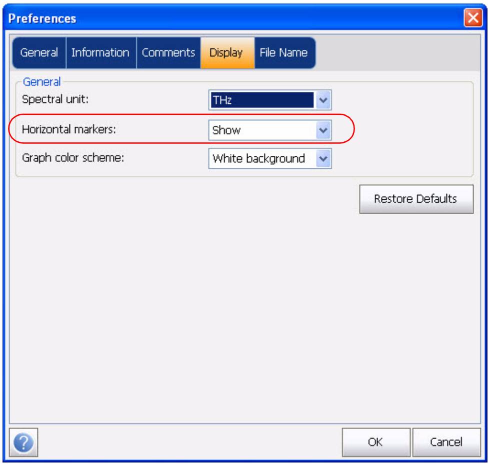
With the nanometer (nm) spectral unit, the trace will appear as shown below:



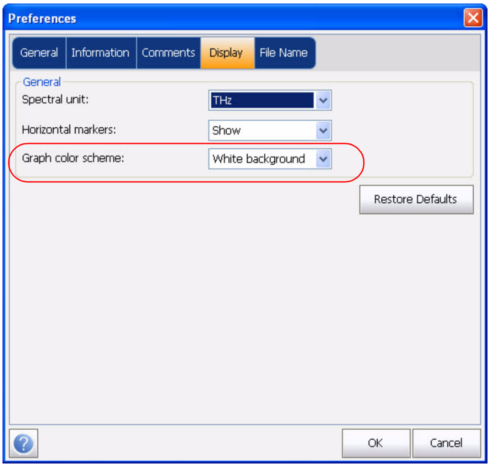
With the terahertz (THz) spectral unit, the trace will appear as shown below:



4. Select **Show** to enable horizontal markers in the **Markers** tab. If **Hide** is selected, horizontal markers will be disabled.



5. Select the background color scheme for Graph as desired.



6. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

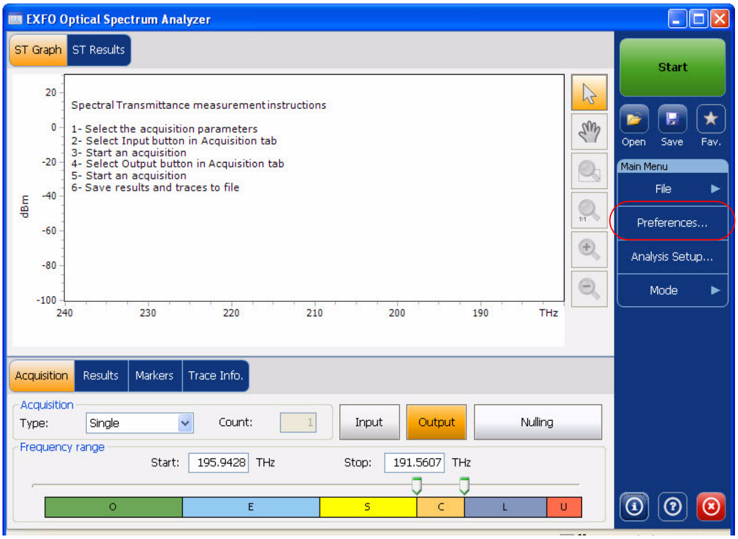
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing Spectral Transmittance File Name

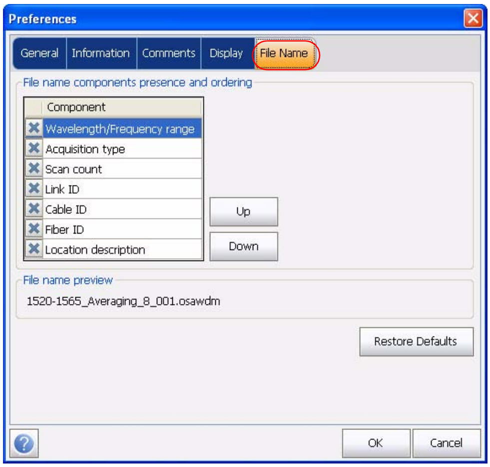
The application shall provide a way to define the name of the next file to be saved. Defining a file autonaming format will allow you to quickly and automatically name traces in a sequential order. The customized name appears when the file is saved using the **Save As** option. It is possible to select which fields you want to include in the file name and the order in which it should be displayed.

To customize the file name:

1. From the **Main Menu**, press **Preferences**.



2. Select the **File Name** tab.

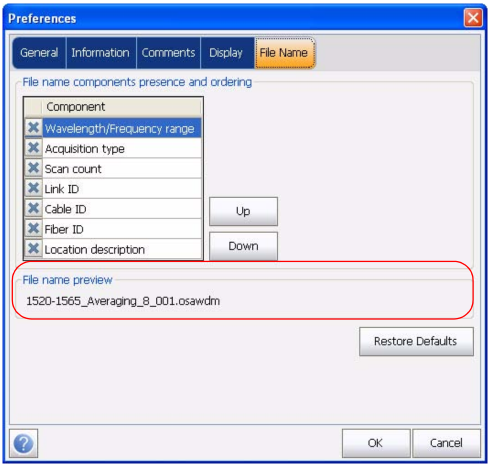


3. Select which parameters you want to include in the file name from the list of available choices:

- **Wavelength/frequency range**: indicates the current wavelength/frequency acquisition range.
- **Acquisition type**: indicates the current acquisition type selected.
- **Scan count**: indicates the current number of scans in the acquisition tab.
- **Link ID**: indicates the prefix value for the link ID configured in the **Preferences-Information** tab.
- **Cable ID**: indicates the prefix value for the cable ID configured in the **Preferences-General** tab.
- **Fiber ID**: indicates the prefix value for the fiber ID configured in the **Preferences-General** tab.
- **Location description**: indicates the location description provided in the **Preferences-Information** tab.

4. Press **Up** or **Down** to change the order in which the field values will appear in the file name.

Based on the fields selected, a preview of the file name shall be displayed under **File name preview**. The field values are separated with an underscore (_).



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default settings.

Setting Up Spectral Transmittance Analysis Parameters

This section presents the various analysis settings for the application, particularly the channel list and settings. These settings are applied on subsequent acquisitions.

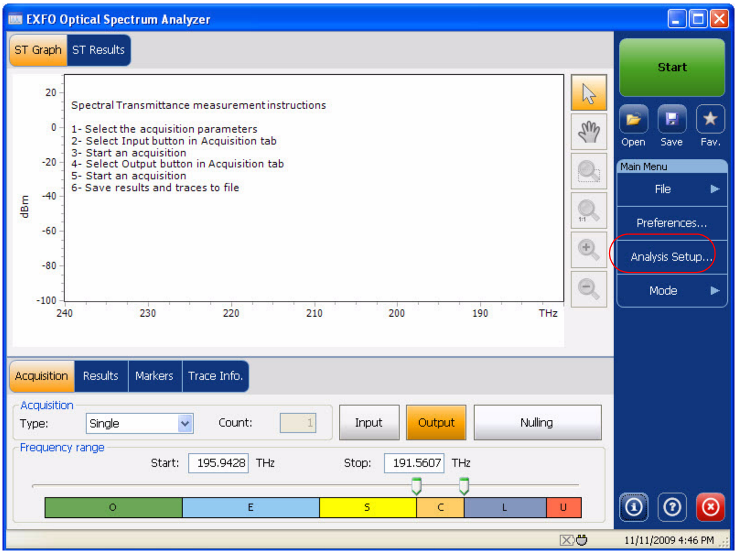
Note: *The analysis setup parameters will be applied to the ST results, upon the next acquisition. However, you can also apply the modifications to the active trace in order to re-analyze it.*

Defining ST Analysis Settings

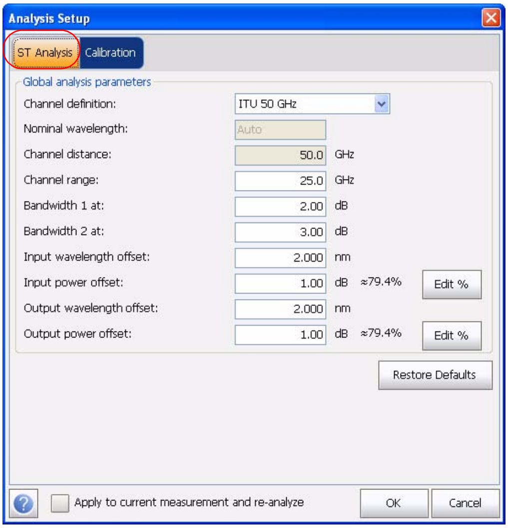
The global analysis parameters for spectral transmittance acquisitions affect the calculation of the results. These calculations take place after an acquisition. If these settings are modified, they will be applied to the next acquisition. However, any modifications made in the general parameters can also be applied to the active trace for re-analysis.

To define ST analysis parameters:

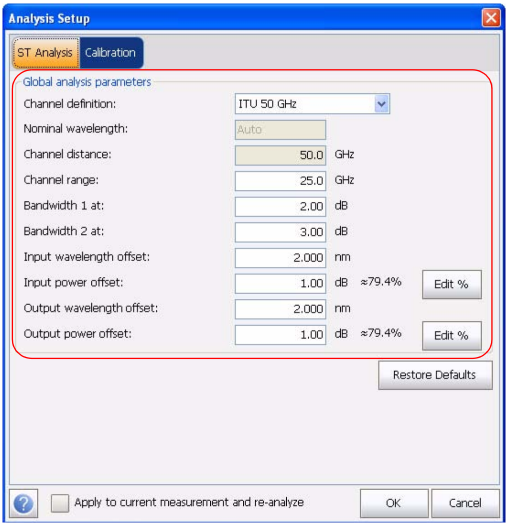
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **ST Analysis** tab.



3. Under **Global analysis parameters**, define the following parameters as needed:



➤ **Channel definition (GHz)**: indicates the limit inside which the power values will be considered in the channel.

Centred on max peak: Channel is centered on the lowest insertion loss peak.

ITU Grid: Select the nearest ITU channel from the peak with lowest insertion loss.

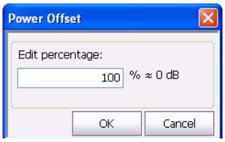
CWDM: Select the nearest CWDM channel from the peak with lowest insertion loss.

Custom: Channel is centered on value specified by the user.

➤ **Nominal wavelength or frequency (nm or Thz)**: indicates a single value that represent the channels center wavelength (in nm) or frequency (in THz). This field is editable only when **Channel definition** is selected as **Custom**.

- **Channel distance (GHz or nm)**: indicates distance between the channels. The value of channel distance will be set depending on the selection made for the channel definition option. The channel distance box will be enabled only when the channel center wavelength option is set to custom.
- **Channel range (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel. Integrated power is calculated on channel width.
- **Bandwidth 1 at (dB)**: Set the power level used, relative to the channel peak power, to compute the bandwidth.
- **Bandwidth 2 at (dB)**: Set the power level used, relative to the channel peak power, to compute the bandwidth.
- **Input wavelength offset (nm)**: indicates the offset value applied on the input wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.
- **Input power offset (dB)**: indicates the offset value applied on the input power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. The power offset appears in the graph, on the left, above the ↕ arrow.

To edit the power offset value in percentage, press **Edit %** button.

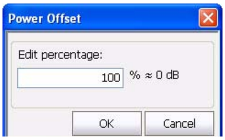


The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

- **Output wavelength offset (nm)**: indicates the offset value applied on the output wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the arrow.

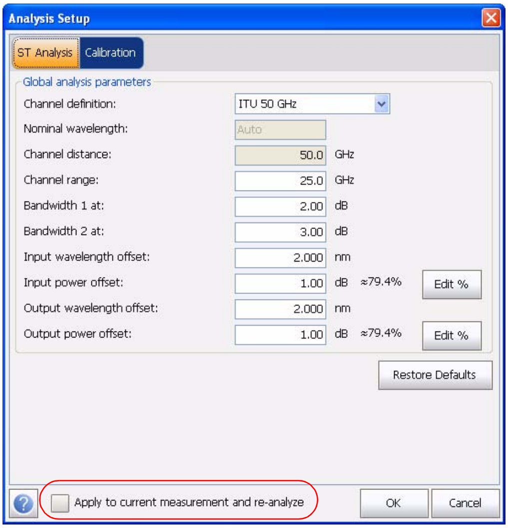
- **Output power offset (dB)**: indicates the offset value applied on the output power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed use. The power offset appears in the graph, on the left, above the arrow.

  To edit the power offset value in percentage, press **Edit %** button.



The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

4. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option and press **OK**.



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Setting Up Acquisition Parameters

Before performing your test, you must set the acquisition type and parameters.

There are three types of acquisitions in Spectral Transmittance mode: Single, Averaging and Real-Time.

➤ **Single:** Spectral measurement is performed once. The results are displayed according to this measurement. In single mode, noise measurement can be performed In-Band if the option is available.

➤ **Averaging:** Spectral measurements are performed based on the number of scans that you have entered for this parameter. The trace will be displayed after each acquisition and averaged with the previous traces.

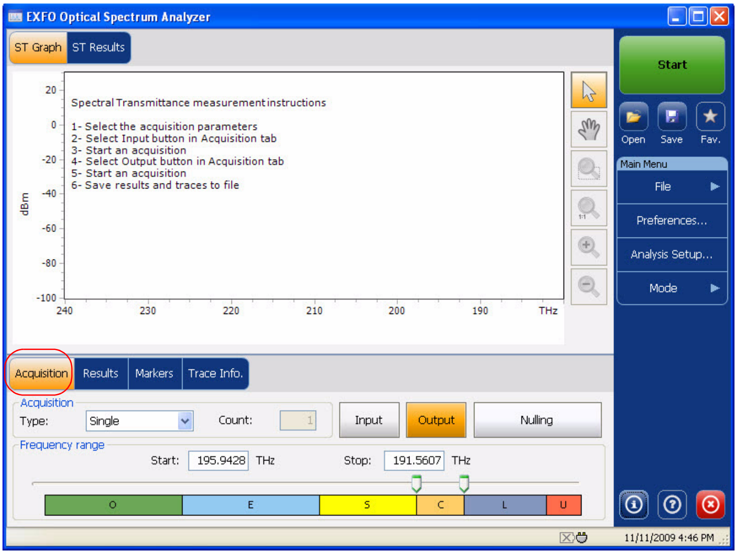
➤ **Real-Time**: In Real-Time acquisition, spectral measurements are performed continuously until you press **Stop**. No averaging is done for spectral measurements. The graph and results are refreshed after each acquisition.

Before performing measurements on an optical spectrum, you must select the wavelength/frequency range to use. You can perform the scan on the full range, on spectral bands, or select a custom range.

Note: *The shorter the wavelength or frequency range, the faster the acquisition.*

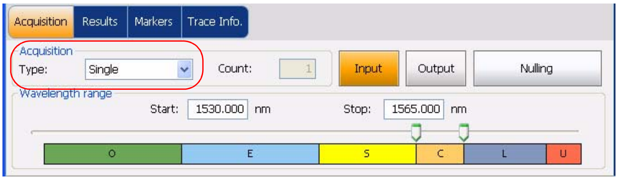
To set parameters in the Acquisition tab:

1. From the main window, select the **Acquisition** tab.

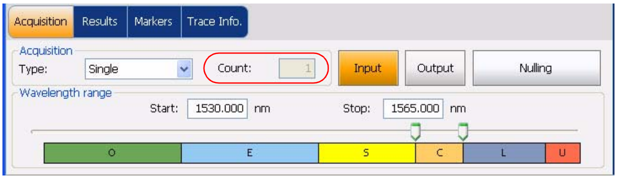


2. Select the acquisition type.

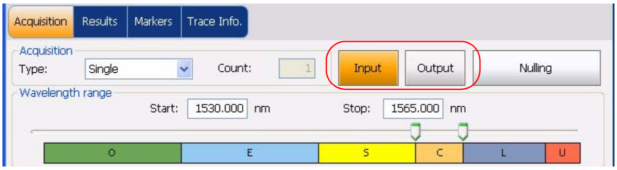
➤ If you are performing a **Single** or **Real-Time** acquisition, you cannot modify the number of scans count value.



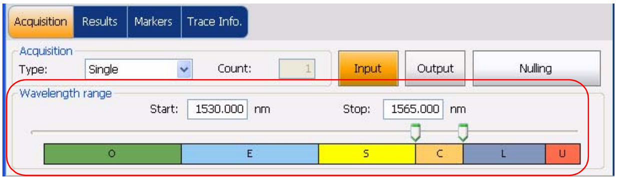
➤ If you are performing an **Averaging** type acquisition, enter the number of scans the unit will perform.



3. Press **Input** or **Output** to specify which position to use to store the next acquisition.



4. Select the wavelength range for your acquisition.



You can select the wavelength range by entering the start and stop values or by selecting a range on the double slider.

To select the wavelength range using the double slider, move the left and right handles on the double slider or simply click on any band.

The wavelength range covered within these bands of the spectra are listed below.

- O band (original): 1259 to 1361 nm
- E band (extended): 1359 to 1461 nm
- S band (short wavelengths): 1459 to 1531 nm
- C band (conventional “erbium window”): 1529 to 1566 nm
- L band (long wavelengths): 1564 to 1626 nm
- U band (ultralong wavelengths): 1624 to 1650 nm.

9 Setting Up the Instrument in EDFA Mode

Before performing a spectral analysis in the EDFA mode, you must set up your OSA module and the test application with the appropriate parameters, as explained in this chapter.

Select the EDFA test mode as explained in *Selecting a Test Mode* on page 18 before setting up the EDFA test parameters.

➤ The *preferences* are the result displayed in the graph and tables, as well as the job information and related comments saved with each file (See *Defining Preferences* on page 210 for more details).

➤ The *analysis parameters* include the channel list details, and allows you to configure global analysis parameters (See *Setting Up EDFA Analysis Parameters* on page 230 for more details).

➤ The *acquisition parameters* include the type of measurement you want to perform and the wavelength range (See *Setting Up Acquisition Parameters* on page 265 for more details).

You can set up your unit in different manners, depending on your testing needs. The possible ways for EDFA mode are preferred and efficient.

➤ The preferred way is to use the complete analysis setup parameters and complete the information in all tables, as explained in *Setting Up EDFA Analysis Parameters* on page 230. This setup will be used for the next acquisition.

➤ The most efficient way to setup the instrument is to use one of the favorites configurations, uploading a pre-customized acquisition and analysis setup configuration. The operator in the field only has to press the star button, select the appropriate configuration and press **Start**. As an example, a pre-customized configuration could be: “32 channels DWDM 50GHz”; “Toronto-Montreal CWDM” or “Vendor ABC DWDM ROADM 40Gb”. This is explained in *Managing Favorites* on page 250.

Defining Preferences

The preferences window allows you to set general information and comments on trace, set display parameters and customize the EDFA results table. This information is saved with all the traces.

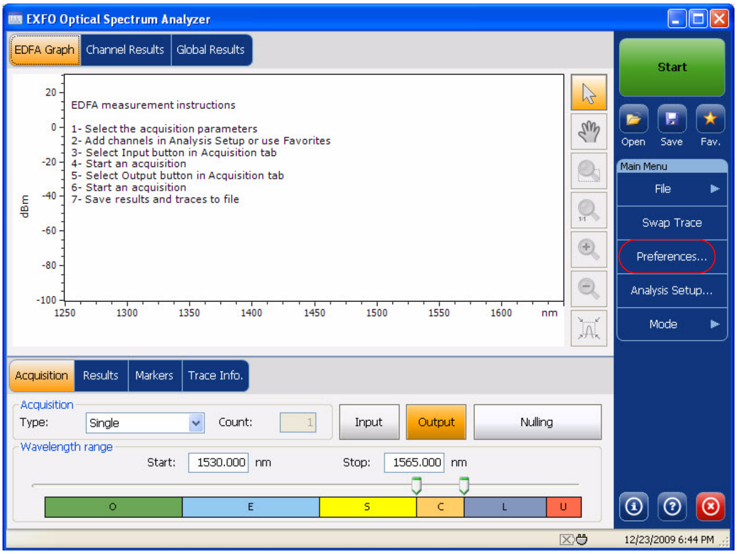
Defining Trace Information

The trace information relates to the description of the job to be done, cable and job IDs, and any relevant information about what is being tested.

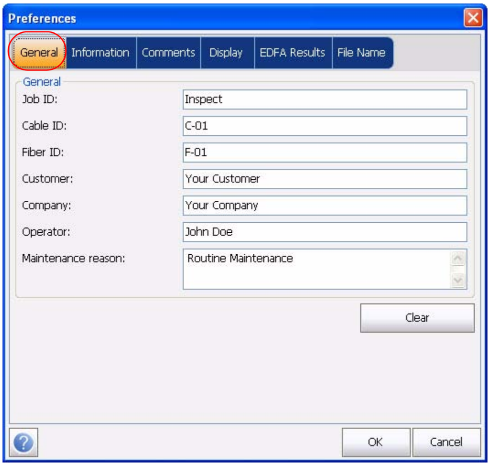
The Link ID is used by the application to propose a file name when you want to save the current acquisition. The link parameters are prefix and suffix values (file names) for the link IDs.

To enter general information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **General** tab.



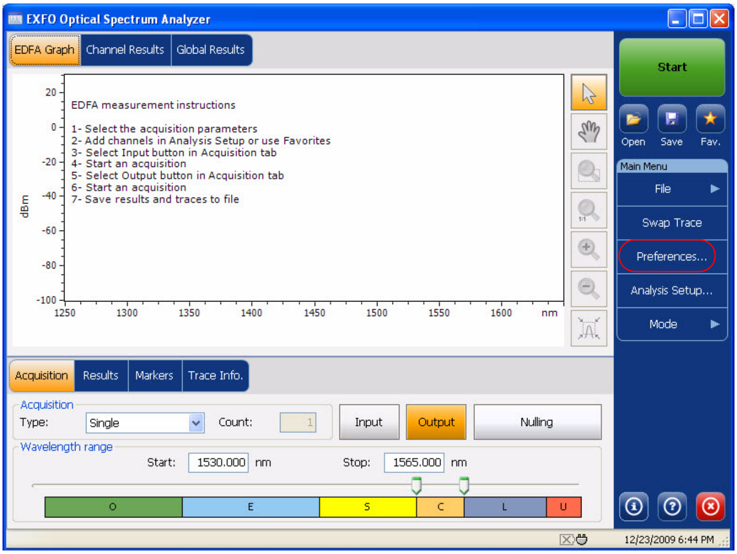
3. Define the general parameters as needed.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

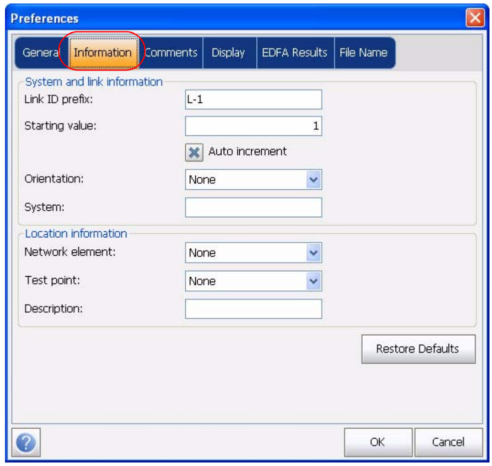
Press **Clear** to clear all the changes made in the **General** tab.

To enter link and location information:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Information** tab.



3. Under **System and link information**, define the following parameters as needed:

➤ **Link ID prefix**: Sets the prefix value for the link ID. You can enter any alphanumeric value.

➤ **Starting value**: Sets the suffix increment starting value for the link ID.

This value is incremented each time a new file is saved provided the **Auto Increment** option is selected.

IMPORTANT

When the **Auto Increment** option is not selected, while saving the trace file, you have to manually change the file name, else the application will overwrite the previously saved files every time you save a new trace.

➤ **Orientation**: Sets the orientation of the link.

➤ **System**: Sets the system information.

4. Under **Location Information**, define the following parameters as needed:

➤ **Network element**: Sets the type of network element.

➤ **Test point**: Sets the test point.

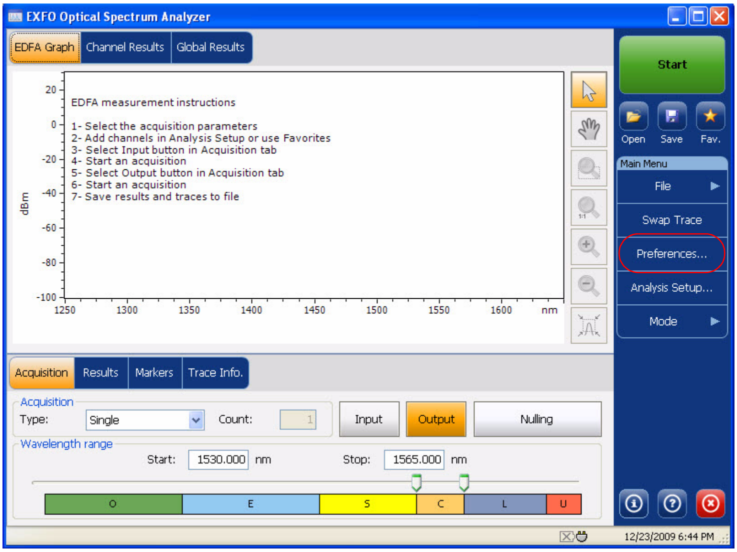
➤ **Description**: Enter the description of location if required.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

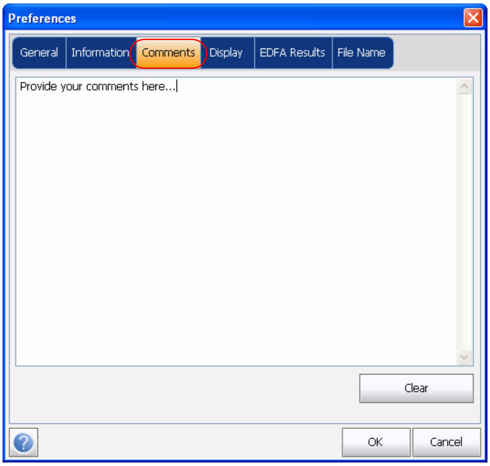
Press **Restore Defaults** to remove all the changes and apply the default values.

To enter comments:

1. From the **Main Menu**, press **Preferences**.



2. Select the **Comments** tab.



3. Enter your comments for the current trace.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

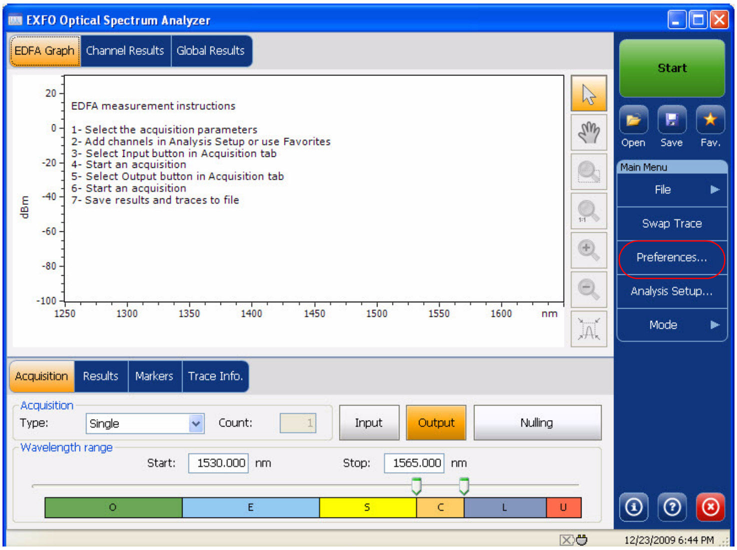
Press **Clear** to clear all the changes made in the **Comments** tab.

Defining Display Parameters

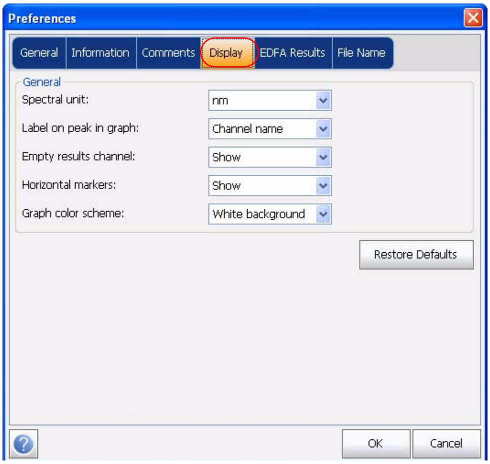
The application allows you to set display settings for the acquisition trace. You can set the spectral unit for the trace and the results table. You can also select the label that should appear on the peaks of the trace.

To define display parameters:

1. From the **Main Menu**, press **Preferences**.

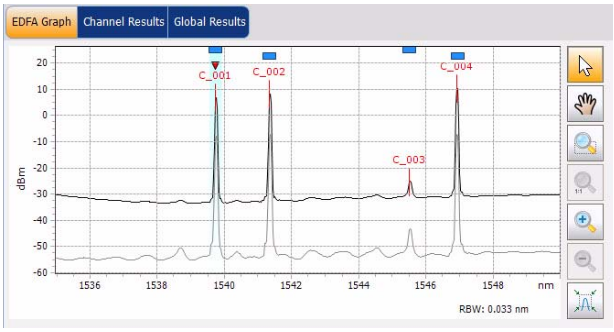


2. Select the **Display** tab.

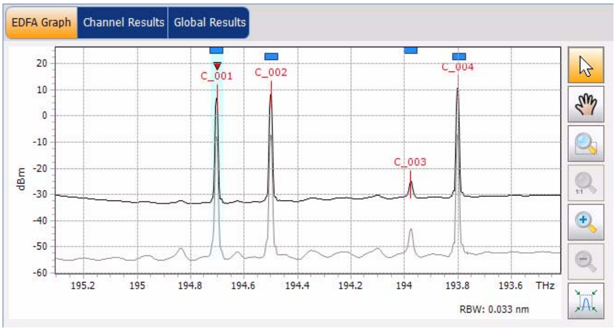


3. Select the spectral unit you want to work with.

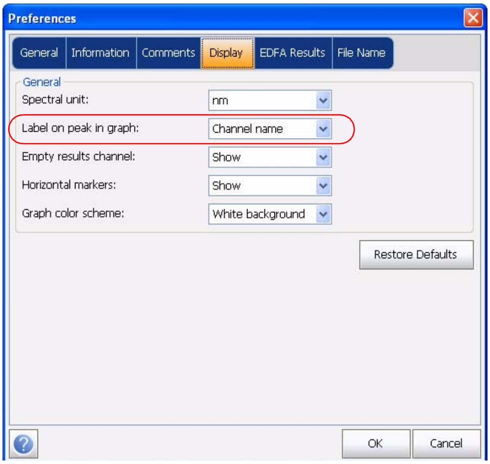
With the nanometer (nm) spectral unit, the trace will appear as shown below:



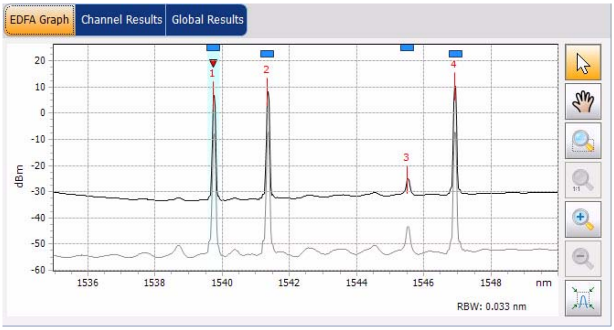
With the terahertz (THz) spectral unit, the trace will appear as shown below:



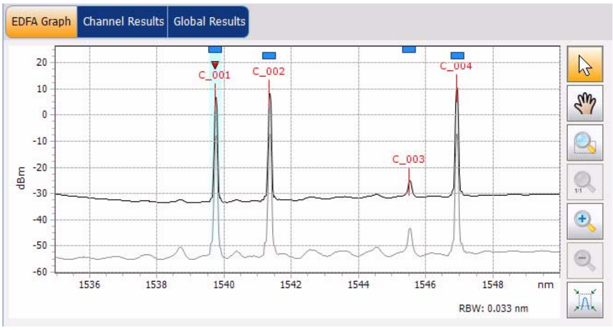
4. Select the label that will appear on the peaks in the graph.



Note: *The channel name and channel number cannot be shown at the same time. If a number is displayed on the peak, it means that the channel name for that peak is not defined. If a channel name is defined for the peak, it will be displayed at the top of the peak.*

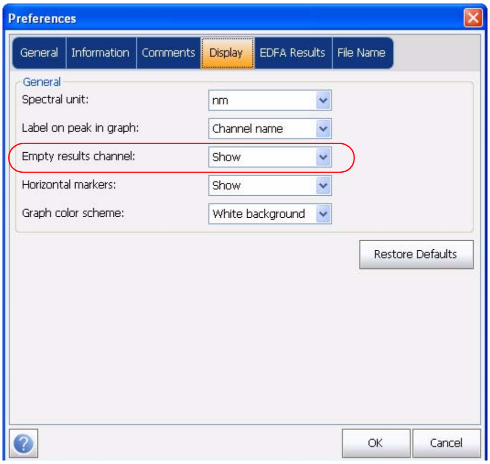


Default channel names

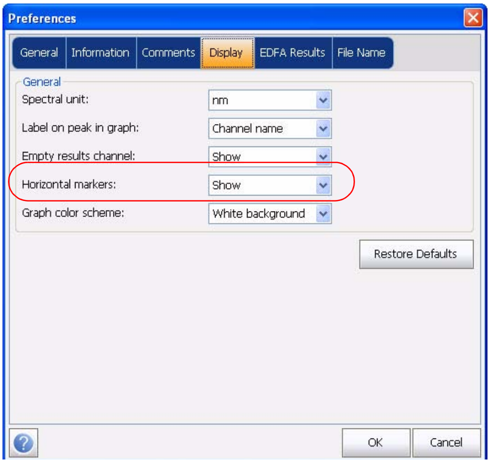


Defined channel names

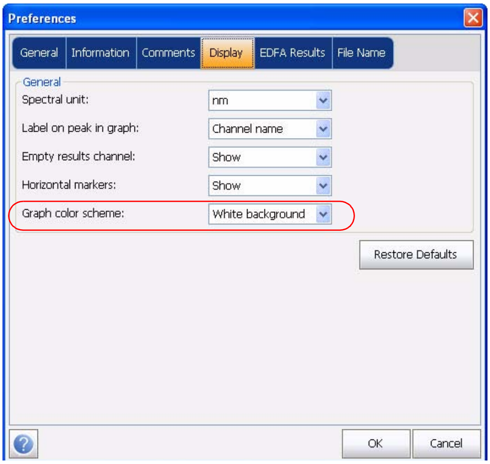
5. Select **Show** to display the empty channels from the channel list in the **Results** tab. If **Hide** is selected, it will not display the empty channels in the **Results** tab.



6. Select **Show** to enable horizontal markers in the **Markers** tab. If **Hide** is selected, horizontal markers will be disabled.



7. Select the background color scheme for Graph as desired.



8. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

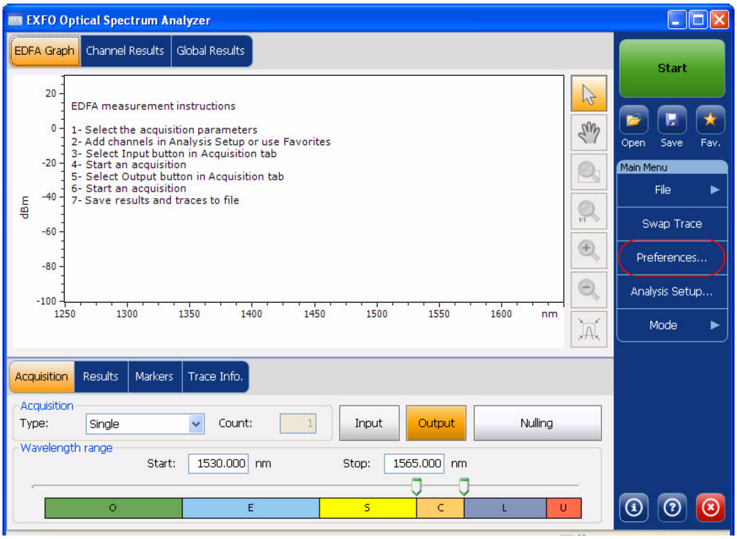
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing EDFA Results Table

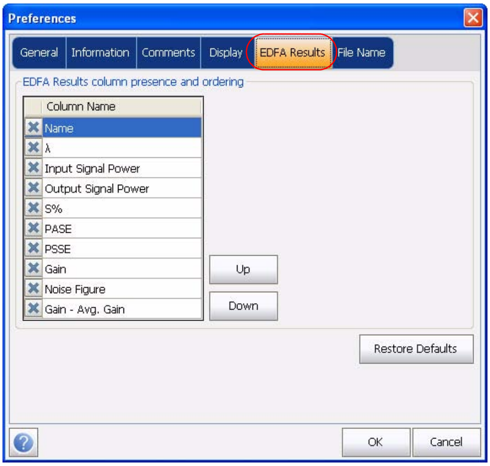
It is possible to select which results you would like to be displayed in the **Results** tab of your EDFA tests.

To customize the results table:

1. From the **Main Menu**, press **Preferences**.



2. Select the **EDFA Results** tab.



3. Select which parameters you want to display in the **Results** tab from the list of available choices:

- **Name**: indicates the name of channel.
- **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.
- **Input Signal Power**: indicates the signal power for the selected channel (excludes noise).
- **Output Signal Power**: indicates the signal power for the selected channel (excludes noise).
- **S %**: indicates the current output power according to the measured output power (Output Signal Power / [Output Signal Power + PASE]).
- **PASE**: indicates the power of the spontaneous emission amplified by EDFA.
- **PSSE**: indicates the power of the spontaneous emission of the source.

- **Gain**: indicates the gain (Output Signal Power - Input Signal Power) for the selected channel.
- **Noise Figure**: indicates the EDFA's noise figure measured for the selected channel.
- **Gain - Avg. Gain**: indicates the selected channel gain minus the average of all channel gains.

4. Press **Up** or **Down** to change the order in which the columns will appear in the **Results** tab.
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

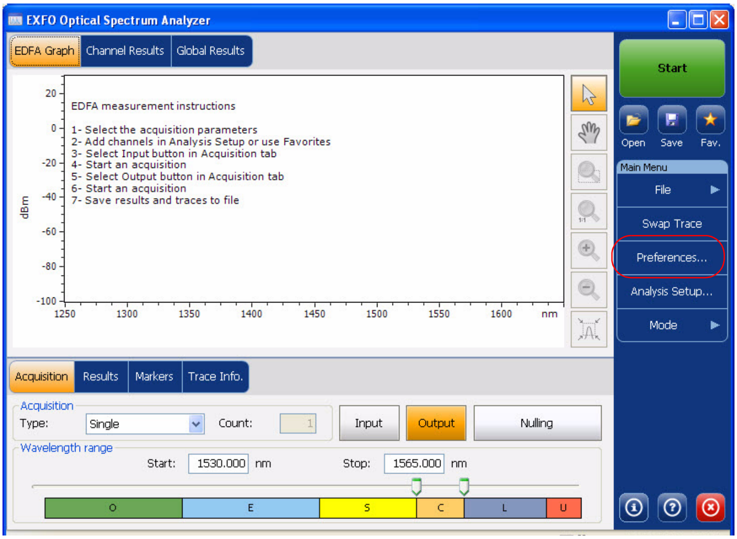
Press **Restore Defaults** to remove all the changes and apply the default values.

Customizing EDFA File Name

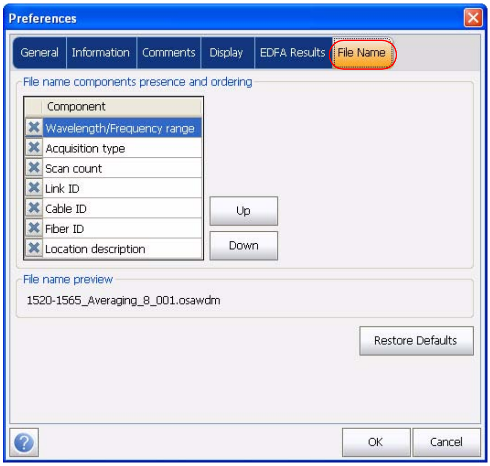
The application shall provide a way to define the name of the next file to be saved. Defining a file autonaming format will allow you to quickly and automatically name traces in a sequential order. The customized name appears when the file is saved using the **Save As** option. It is possible to select which fields you want to include in the file name and the order in which it should be displayed.

To customize the file name:

1. From the **Main Menu**, press **Preferences**.



2. Select the **File Name** tab.

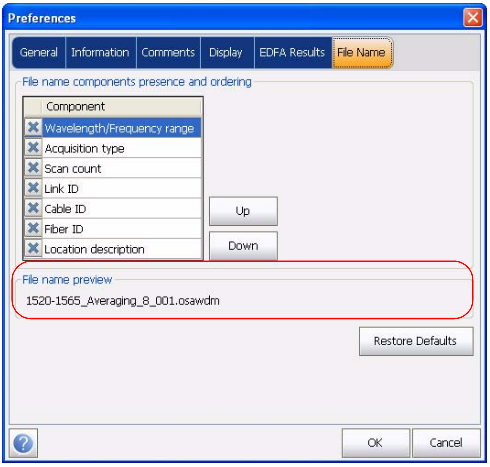


3. Select which parameters you want to include in the file name from the list of available choices:

- **Wavelength/frequency range**: indicates the current wavelength/frequency acquisition range.
- **Acquisition type**: indicates the current acquisition type selected.
- **Scan count**: indicates the current number of scans in the acquisition tab.
- **Link ID**: indicates the prefix value for the link ID configured in the **Preferences-Information** tab.
- **Cable ID**: indicates the prefix value for the cable ID configured in the **Preferences-General** tab.
- **Fiber ID**: indicates the prefix value for the fiber ID configured in the **Preferences-General** tab.
- **Location description**: indicates the location description provided in the **Preferences-Information** tab.

4. Press **Up** or **Down** to change the order in which the field values will appear in the file name.

   Based on the fields selected, a preview of the file name shall be displayed under **File name preview**. The field values are separated with an underscore (_).



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default settings.

Setting Up EDFA Analysis Parameters

This section presents the various analysis settings for the application, particularly the channel list and settings. These settings are applied on subsequent acquisitions. You can set the channel list, channel parameters, manage favorite configurations and perform user calibration.

Note: *The analysis setup parameters will be applied to the global results and channel results, upon the next acquisition.*

Defining General Settings

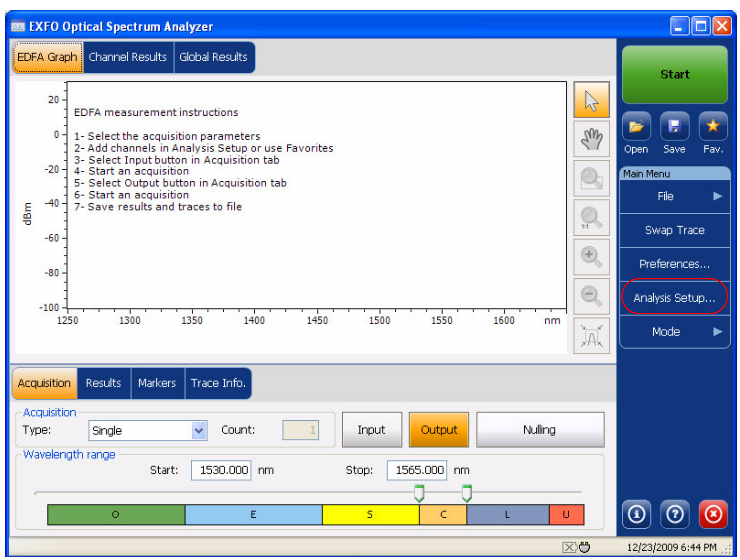
The general analysis parameters for EDFA acquisitions affect the calculation of the results. These calculations take place after an acquisition. If these settings are modified, they will be applied to the next acquisition.

IMPORTANT

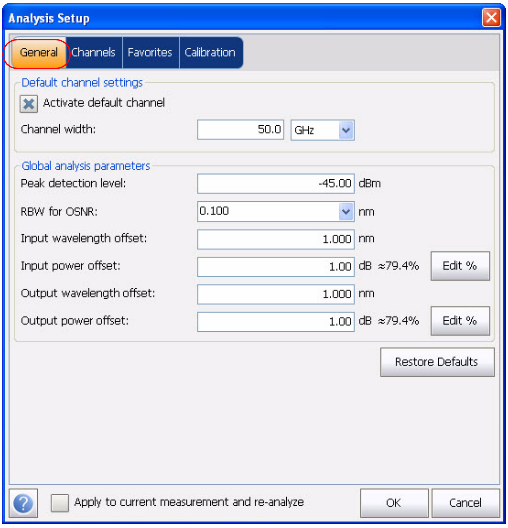
In the General tab, you can set the default channel parameters. Any channel found during an acquisition that is not defined in the channel list will be analyzed according to the default channel settings.

To define general settings:

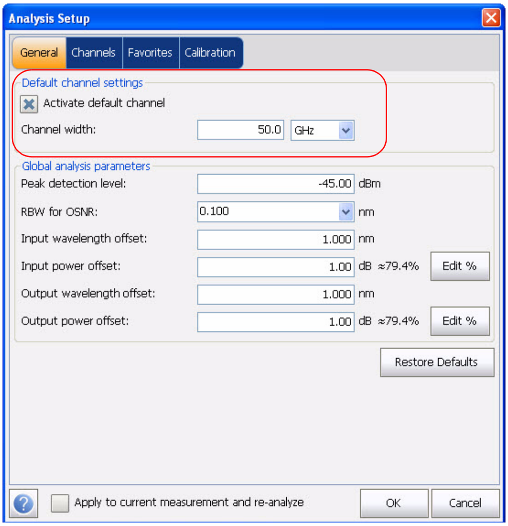
1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **General** tab.



3. Under **Default channel settings**, define the following parameters as needed:

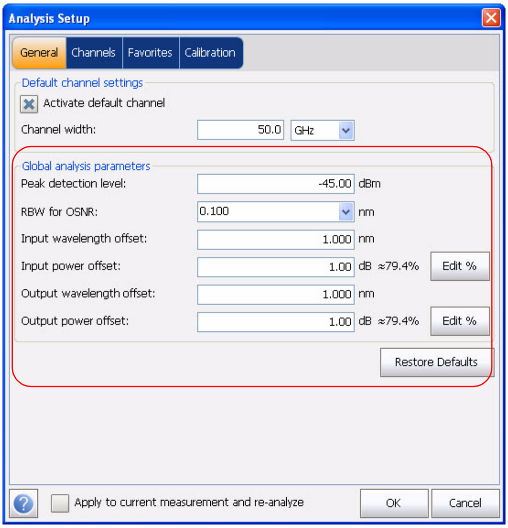


➤ Clear the **Activate default channel** selection, to use the currently defined channel for analysis. This reduces the analysis time by eliminating the peak detection over the complete spectral range. The peaks outside the defined channel will not be detected or analyzed.

➤ **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.

For default channels, the channel width that sets the limits of the channel, should be the same as the channel distance or smaller (channel distance is defined while creating a channel list). If the channel width is not compatible with the channel spacing, either a single peak may be found for two distinct channels and two analysis would be performed and displayed for that peak, or, it is possible that two peaks may be found within the same channel and be considered as one multi-peak signal. With this result, you can use markers to find the spacing between adjacent channels or to find the channel width.

4. Under **Global analysis parameters**, define the following parameters as needed:



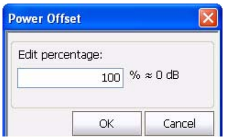
➤ **Peak detection level (dBm)**: indicates the minimum power level from where the peak can be considered as a signal.

➤ **RBW for OSNR (nm)**: indicates the resolution bandwidth for the selected OSNR value. The resolution bandwidth of an OSA determines its ability to deal with close optical channel spacing. It is measured as the width of the response curve at half peak power (i.e., 3 dB down) of the instrument to a monochromatic test signal. The instrument's RBW value is written below the graph on the right (below wavelength offset).

➤ **Input wavelength offset (nm)**: indicates the offset value applied on the input wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.

➤ **Input power offset (dB)**: indicates the offset value applied on the input power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the ↕ arrow.

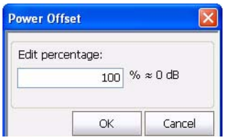
To edit the power offset value in percentage, press **Edit %** button.



The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

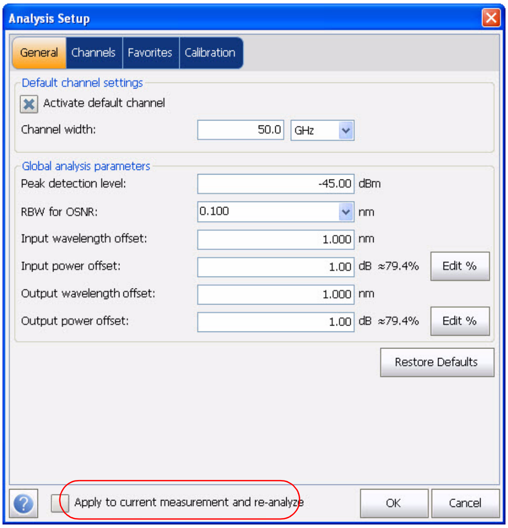
- **Output wavelength offset (nm)**: indicates the offset value applied on the output wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the arrow.
- **Output power offset (dB)**: indicates the offset value applied on the output power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the arrow.

  To edit the power offset value in percentage, press **Edit %** button.



The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

5. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option and press **OK**.



6. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Managing Channels

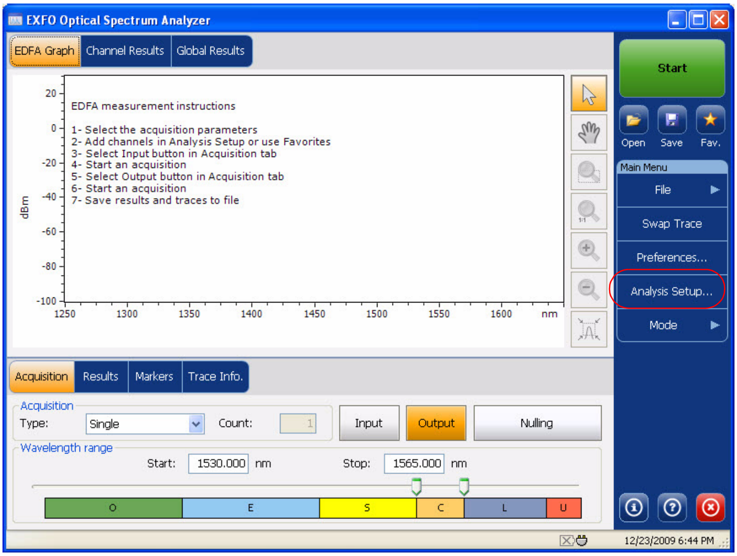
Testing DWDM systems involves characterizing multiple signals in a link. The application allows you to define channels using a channel editor or quickly generate them from the current data. You can also rapidly create a list of equally spaced channels. Once a channel list is created, you can modify it as needed. You can edit the analysis parameters for one channel or multiple channels.

Adding Channels

While creating the channel list, some channels may overlap. When the channel widths are specified in nm, two channels are considered to be overlapping when more than 0.001 nm of wavelength range is common between the two channels.

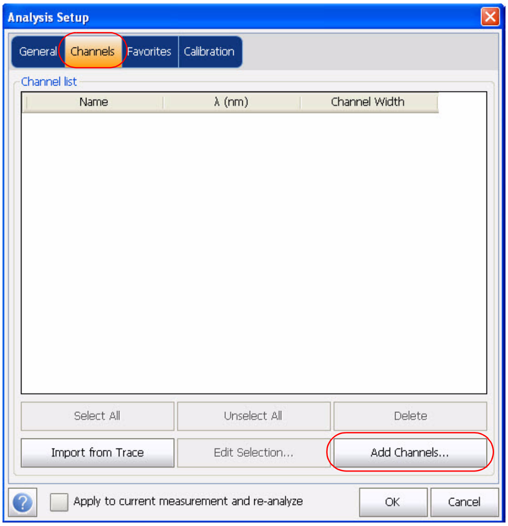
To add a channel list:

1. From the **Main Menu**, press **Analysis Setup**.

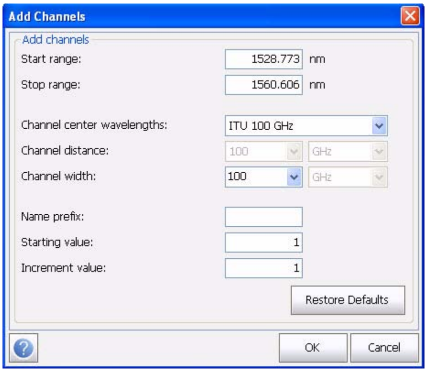


2. Select the **Channels** tab.

3. By default, the channel list is empty. Press **Add Channels**.



4. Enter values in the boxes as explained below:



- **Start range (nm or THz)**: indicates the starting range of the channel list.
- **Stop range (nm or Thz)**: indicates the ending range of the channel list.
- **Channel center wavelength/frequency:** indicates the spectral center-of-mass for the peak in that channel.

Note: *When using the custom channel center wavelength option, the first channel will be centered at the Start Range, and the list will be created using channel distance and channel width.*

- **Channel distance (nm or GHz)**: indicates distance between the channels. The value of channel distance will be set depending on the selection made for the channel center wavelength option. The channel distance box will be enabled only when the channel center wavelength option is set to custom.
- **Channel width (nm or GHz)**: indicates the limit inside which the power values will be considered in the channel. Integrated power is calculated on channel width.
- **Name prefix:** Adds prefix to the channel names.

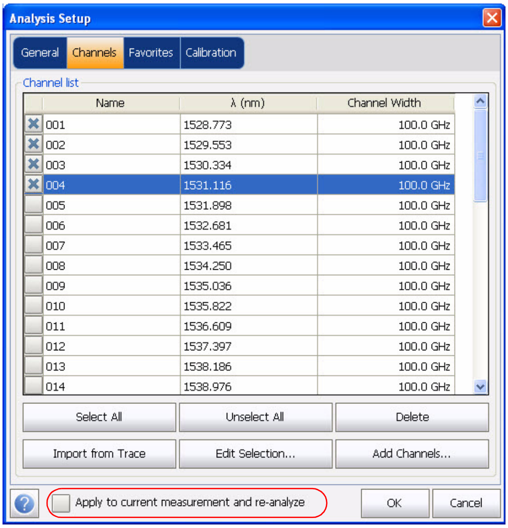
- **Starting Value:** Sets the increment starting value for the channel name in the channel list.
- **Increment value:** Sets the increment value for the channel name in the channel list.

5. Press **OK** to return to the **Channels** window, which now lists the added channels.

Note: *When new channels are added, the Use Default thresholds will be applied to the channel parameters.*

Note: *A warning message will be displayed if any channels are overlapping, but the analysis can still be performed on overlapping channels. If any duplicate channels are added, a confirmation message will be displayed to overwrite the existing channels with the duplicate channels.*

6. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option.

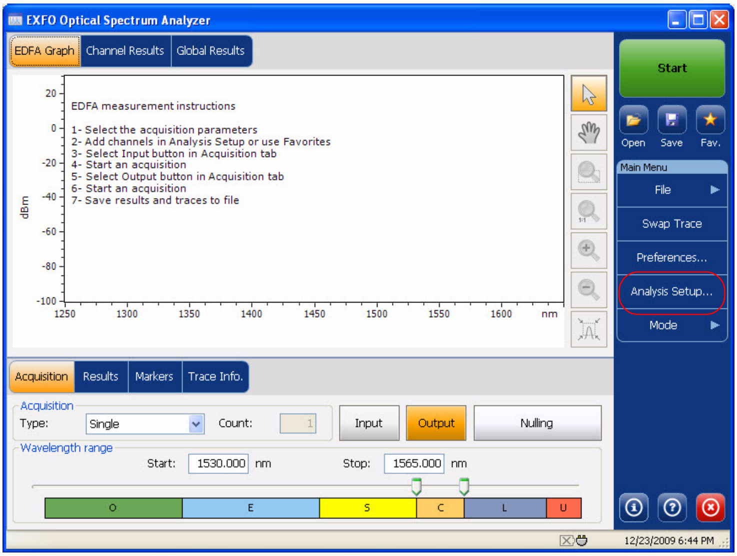


7. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

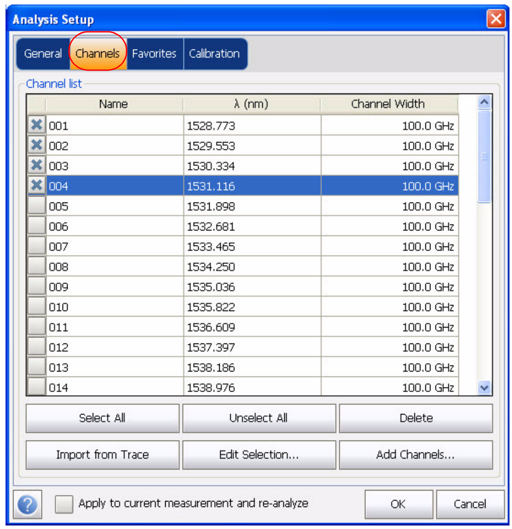
Note: *The application displays a message if more than 1000 channels are added. You can exit the* ***Analysis Setup*** *window only after deleting the extra channels from the channel list. You can delete the channels manually as required.*

To edit the parameters of a specific channel:

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.

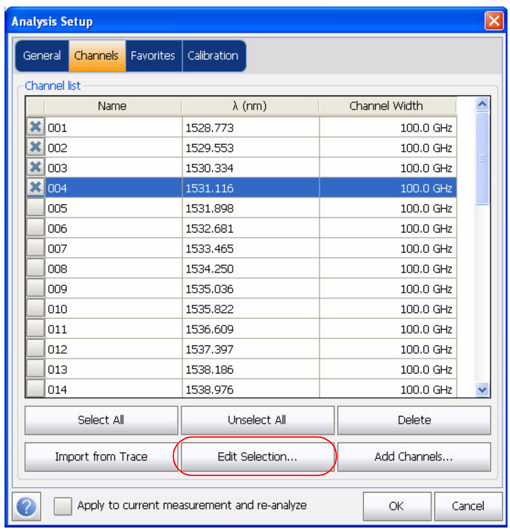


3. Select the channel or channels to be modified in the channel list.

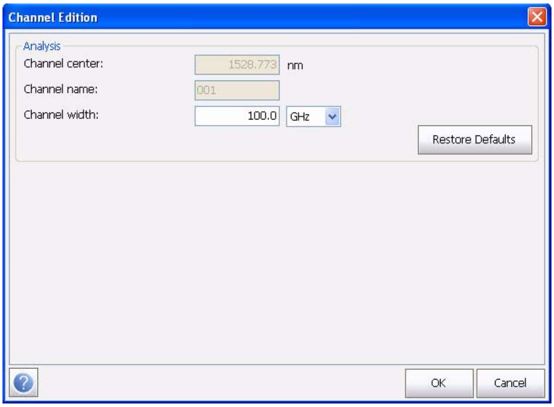
Selected channels have a  in the first column of the channels table.

If you want the changes to be applied to all of your channels, press **Select All**. Channels can be selected one by one or all together. You can press **Unselect All** to clear all channel selections. To delete the selected channels, press **Delete**.

4. Press **Edit Selection**.

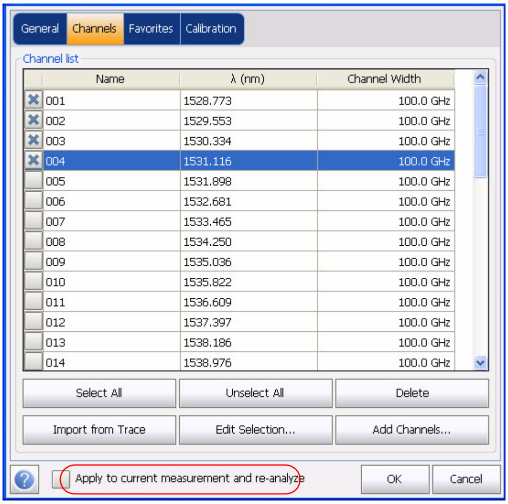


5. Modify the settings as needed. For more information about the settings, see Adding Channels *on page 238*. If you leave a box empty, it will remain as it was before your changes. Modify appropriate settings.



6. Press **OK** to return to the **Channels** tab, which now contains the modified settings.

7. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option in the **Channels** window and press **OK**.

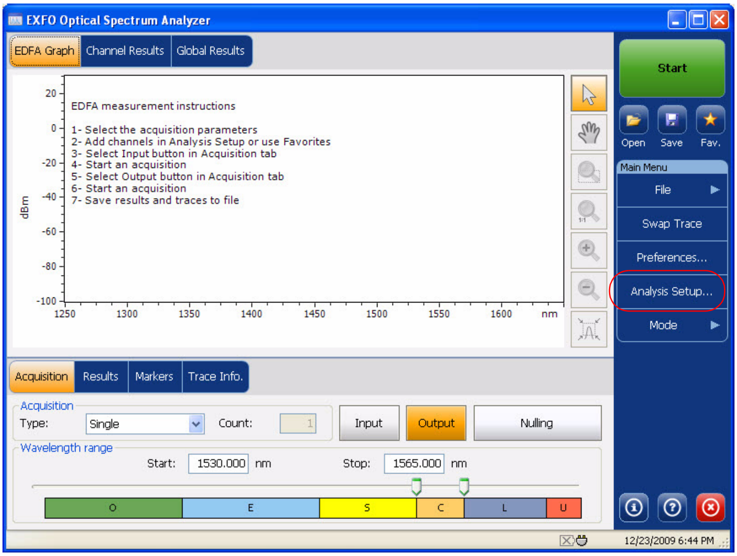


8. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

To add current peaks:

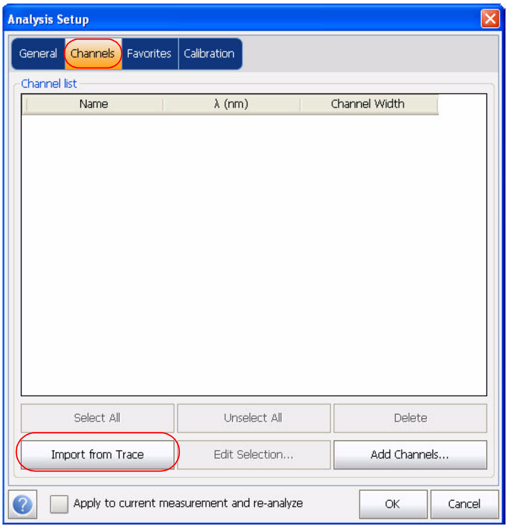
Note: *You can add current peaks to the channel list only if an acquisition has already been performed.*

1. From the **Main Menu**, press **Analysis Setup**.



2. Select the **Channels** tab.

3. Press **Import from Trace**. All peaks from the current trace will be added to the channel list.

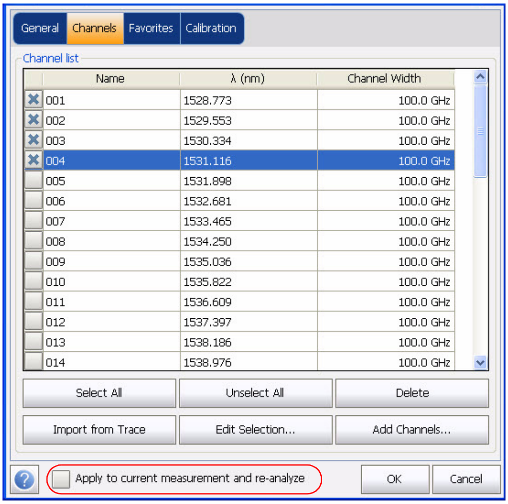


A warning message will be displayed if any channels are overlapping. Press **OK** to close the warning window.

Note: *Changes can be applied to any channel at any time.*

Note: *If some channels were already present in the channel list, the new channels created with the Add Current Peaks button will be added to the list.*

4. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option and press **OK**.



5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Managing Favorites

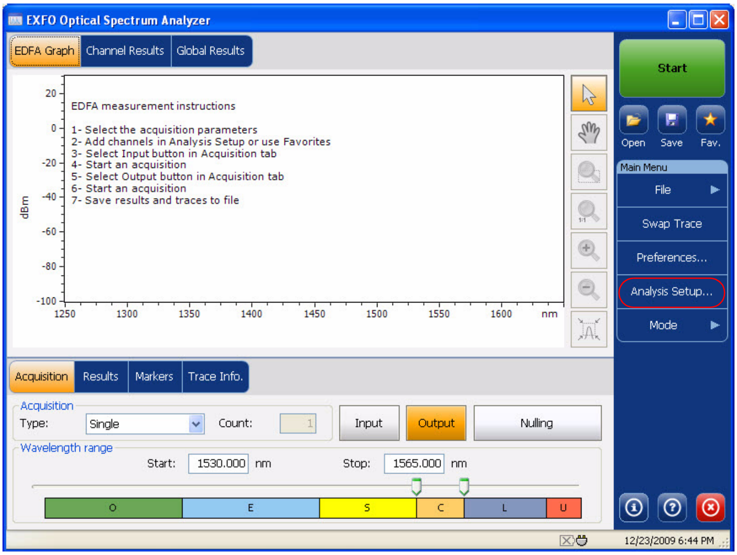
Favorites are configurations files that contain all of the parameters from the **Analysis Setup** tab and **Acquisition** tab. When you often use the same settings, you can save them as a favorite, then recall them for future acquisitions.

To apply a test configuration to the current acquisition:

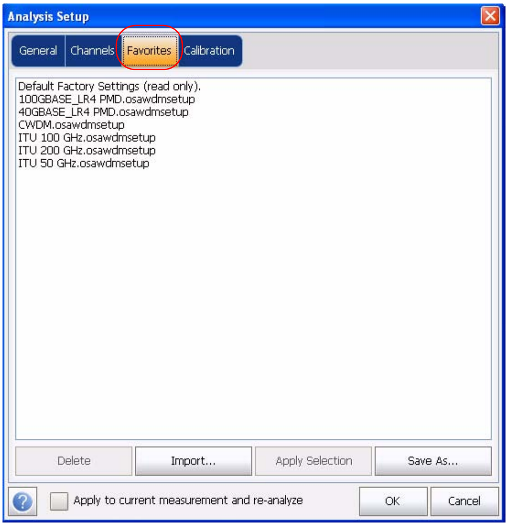
1. From the **Main Menu**, press **Analysis Setup**.

   OR

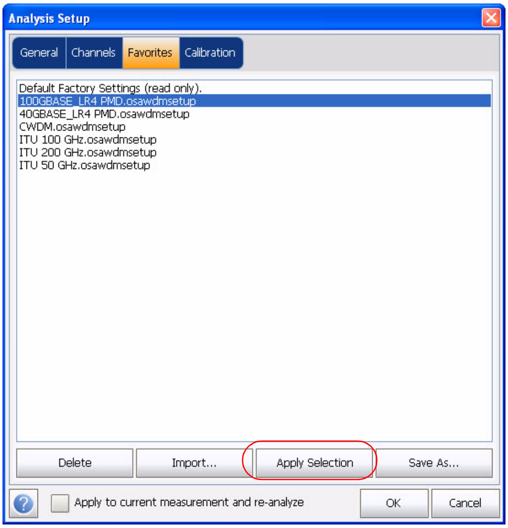
   From the main window, press .



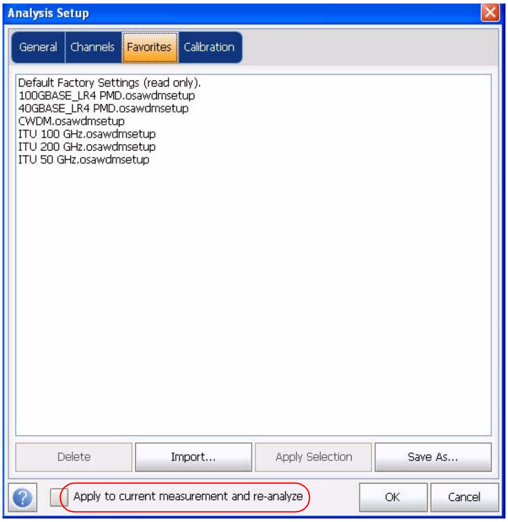
2. Select the **Favorites** tab.



3. To apply the settings from a favorite file to the current analysis setup, select a file from the favorites list and press **Apply Selection**. The apply selection button will be enabled only when a file is selected from the favorites list. When the **Apply Selection** button is pressed, the content of the file will be loaded in the other tabs of this page. To make these parameters effective for the next acquisition you need to press the **OK** button.



4. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option and press **OK**.



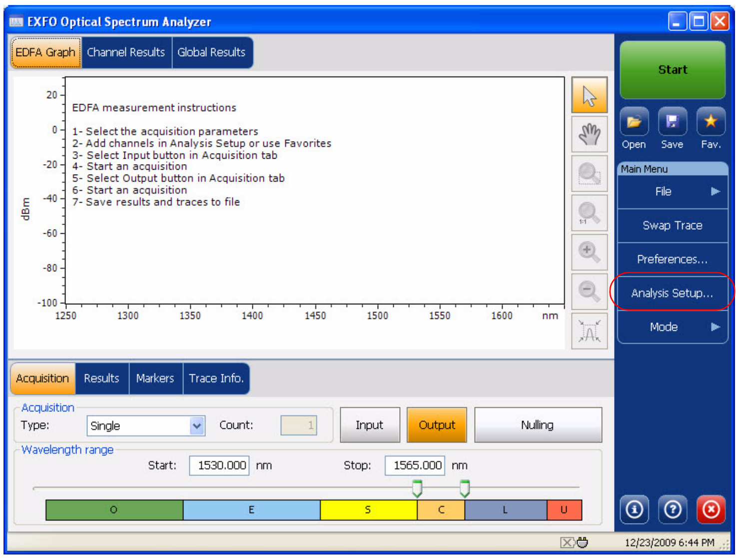
5. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To save a test configuration:

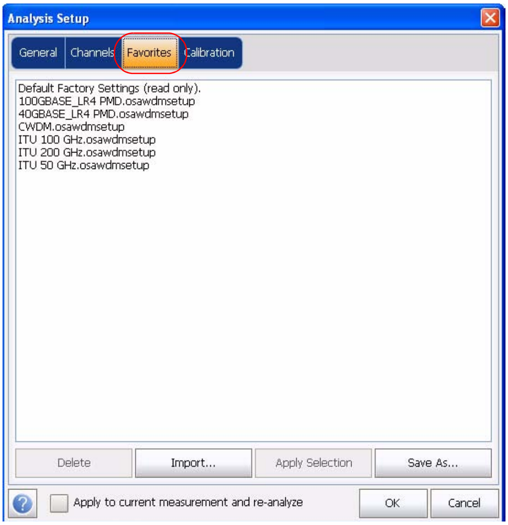
1. From the **Main Menu**, press **Analysis Setup**.

   OR

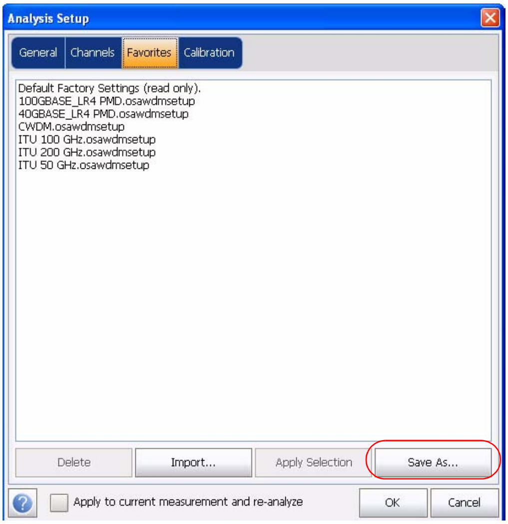
   From the main window, press .



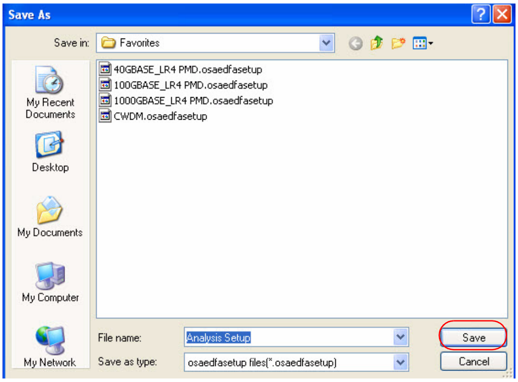
2. Select the **Favorites** tab.



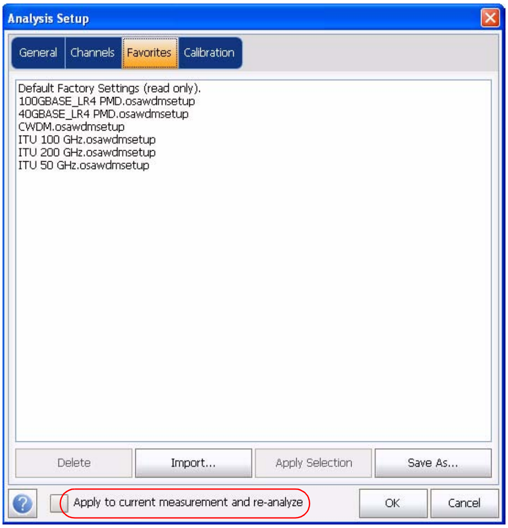
3. To save an analysis setup to a file, press **Save As**. The default directory where the file will be saved is the favorites folder. You should use this folder unless you save the file on a USB stick.



4. In the **Save As** window, enter a file name and press **Save**. The file will be added to the favorites list in the **Analysis setup – Favorites** window.



5. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option and press **OK**.



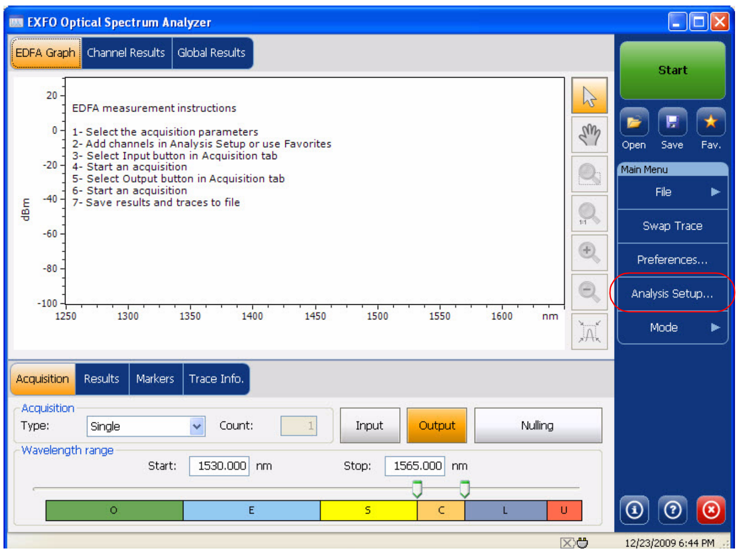
6. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To import a test setup:

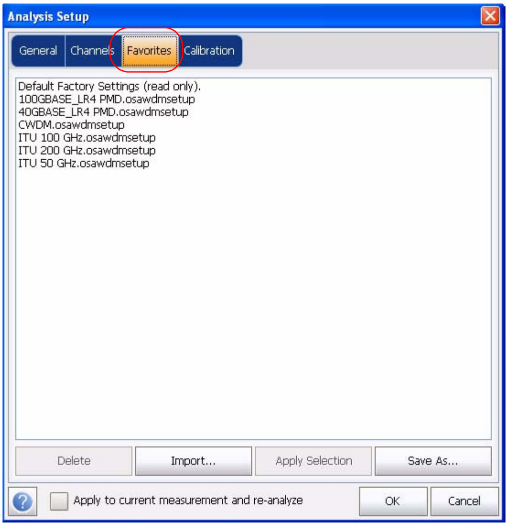
1. From the **Main Menu**, press **Analysis Setup**.

OR

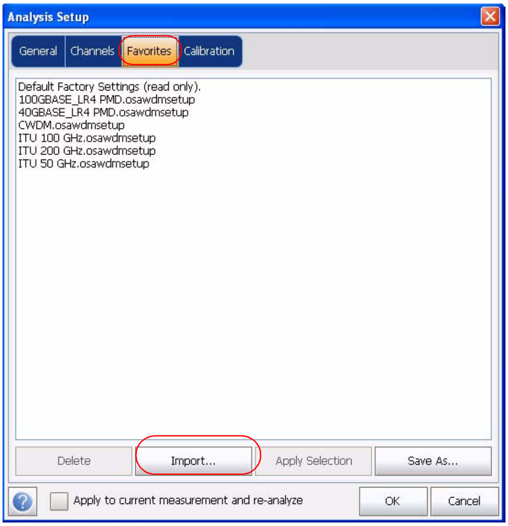
From the main window, press .



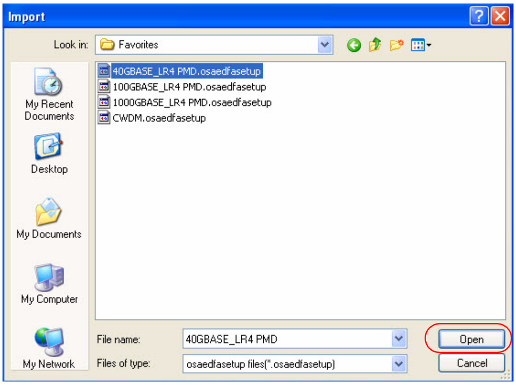
2. Select the **Favorites** tab.



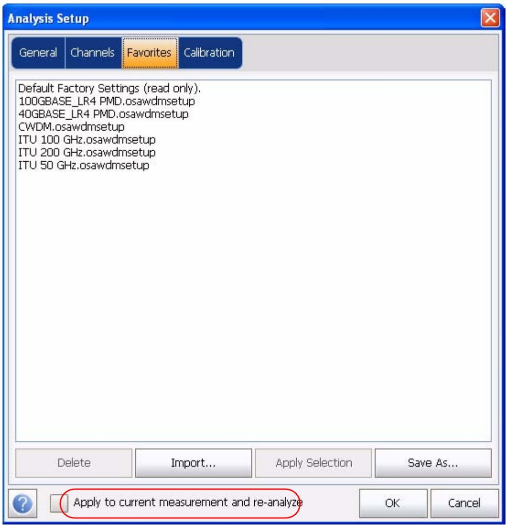
3. Press **Import** to import an analysis setup from a file.



4. From the Import window, select the file you want to import and press **Open**. The file will be added to the favorites list in the **Analysis setup – Favorites** window.



5. If you want to apply the above settings to the active trace, select the **Apply to current measurement and re-analyze** option and press **OK**.



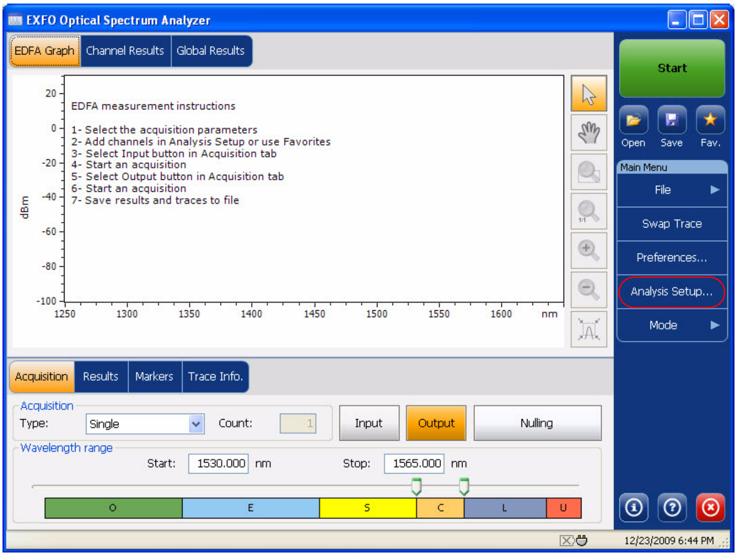
6. Press **OK** to load the configuration and close the window, or press **Cancel** to exit without saving.

To delete a test configuration:

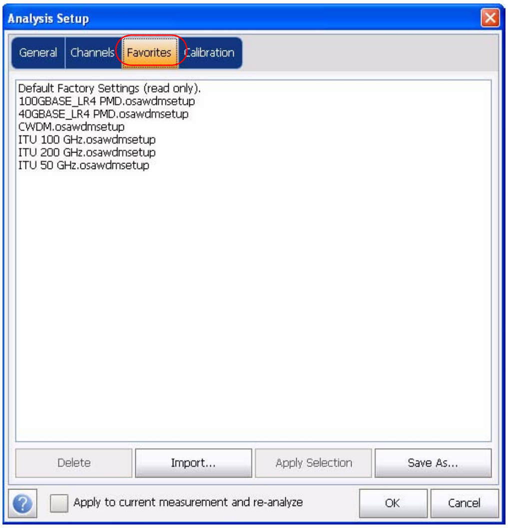
1. From the **Main Menu**, press **Analysis Setup**.

OR

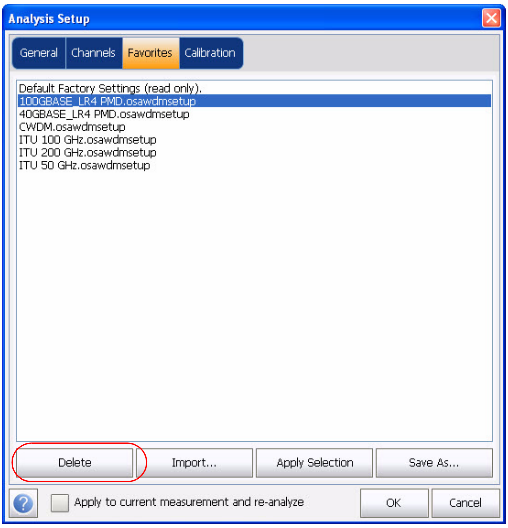
From the main window, press  .



2. Select the **Favorites** tab.



3. To delete a configuration file from the favorites list, select a file from the favorites list and press **Delete**. Press **Yes** to delete the file, else press **No**.



Setting Up Acquisition Parameters

Before performing your test, you must set the acquisition type and parameters.

There are three types of acquisitions in EDFA mode: Single, Averaging and Real-Time.

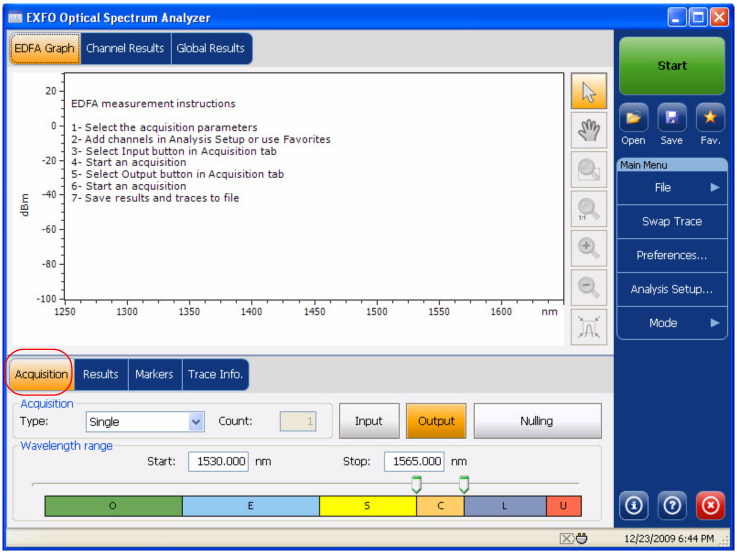
- **Single:** Spectral measurement is performed once. The results are displayed according to this measurement. In single mode, noise measurement can be performed In-Band if the option is available.
- **Averaging:** Spectral measurements are performed based on the number of scans that you have entered for this parameter. The trace will be displayed after each acquisition and averaged with the previous traces.
- **Real-Time**: In Real-Time acquisition, spectral measurements are performed continuously until you press **Stop**. No averaging is done for spectral measurements. The graph and results are refreshed after each acquisition.

Before performing measurements on an optical spectrum, you must select the wavelength/frequency range to use. You can perform the scan on the full range, on spectral bands, or select a custom range.

Note: *The shorter the wavelength or frequency range, the faster the acquisition.*

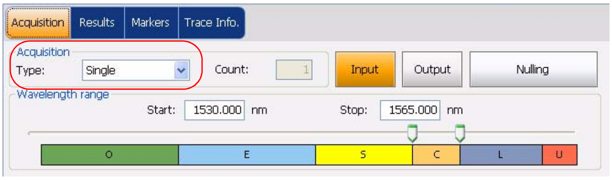
To set parameters in the Acquisition tab:

1. From the main window, select the **Acquisition** tab.

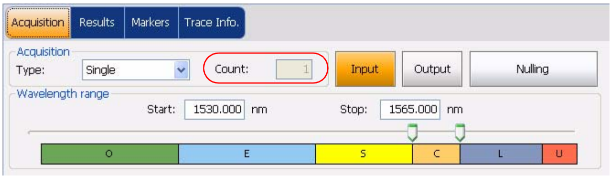


2. Select the acquisition type.

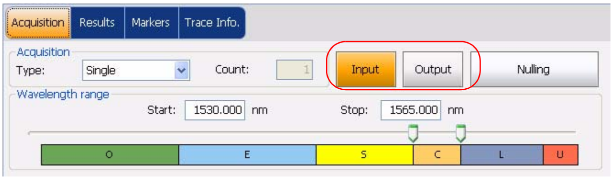
➤ If you are performing a **Single** or **Real-Time** acquisition, you cannot modify the number of scans count value.



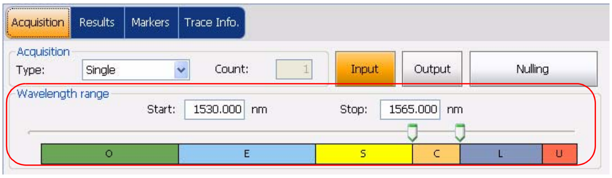
➤ If you are performing an **Averaging** type acquisition, enter the number of scans the unit will perform.



3. Press **Input** or **Output** to specify which position to use to store the next acquisition.



4. Select the wavelength range for your acquisition.



You can select the wavelength range by entering the start and stop values or by selecting a range on the double slider.

To select the wavelength range using the double slider, move the left and right handles on the double slider or simply click on any band.

The wavelength range covered within these bands of the spectra are listed below.

- O band (original): 1259 to 1361 nm
- E band (extended): 1359 to 1461 nm
- S band (short wavelengths): 1459 to 1531 nm
- C band (conventional “erbium window”): 1529 to 1566 nm
- L band (long wavelengths): 1564 to 1626 nm
- U band (ultralong wavelengths): 1624 to 1650 nm.

10 Testing DWDM Systems

Optical spectrum analysis is the measurement of optical power as a function of wavelength or frequency and optical signal to noise ratio (OSNR). Applications include commissioning of a new DWDM or CWDM link, as well as maintenance and troubleshooting of existing networks.

IMPORTANT

For optimal test results, you must allow a minimum warm up period of two hours for your Optical Spectrum Analyzer before starting your tests.

Using the Discover Feature

The Discover feature allows you to start a procedure to automatically build an analysis setup (scan range, channel list, analysis parameters, etc.) based on the signal being detected on the input port of the OSA module. It allows you to start an automatic setup scan.

Note: *Discover features is available on for the WDM and Drift test modes only.*

An automatic scan is a full range single scan that locates the various peaks from the incoming signal. Prior to this scan, the application will clear the data used for processing, empty the channel list, set the analysis parameters to default values (equivalent to pressing the default button in every setup screen).

If no channel is detected with this first scan, then the graph shows the full range scan and ends the analysis. The wavelength range will be set to 1250 nm to 1650 nm and the channel list will be empty in the **Channels** tab on the **Analysis Setup** window.

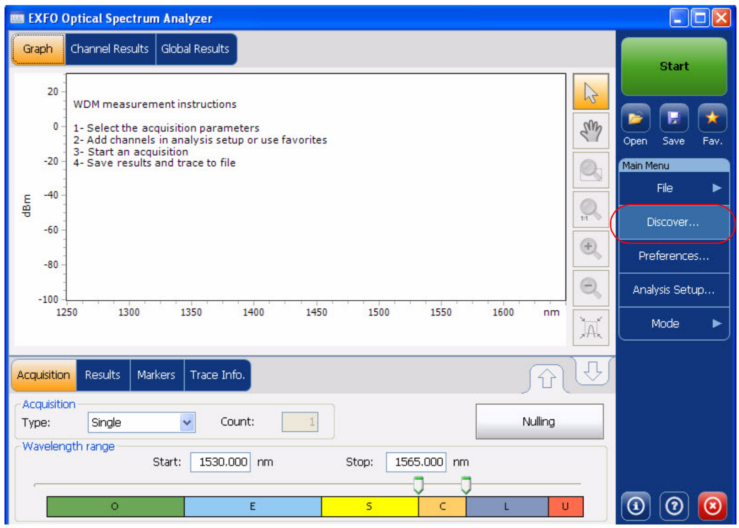
If at least one channel is detected, then the wavelength range is set to 5 nm before the first detected signal peak and 5 nm after the last detected signal peak (and within the wavelength full range, 1250 nm to 1650 nm). All detected channels are added to the channel list and the default settings are used for all channel parameters. The channel center wavelength is aligned with an ITU grid (200, 100, 50 or 25 GHz for DWDM). The channel width is determined using the overlap criteria, if two channels overlap by more than 0.001 nm or 0.001 GHz, then their width is reduce to the lower width. If the width of two channels is at 25 GHz and they still overlap, then the width is not reduced and application considers it as a multi-peak signal (like recent modulation formats for 10 Gb/s or 40 Gb/s) and sets the width of the channel to 50 GHz and calculates the results based on this information.

A new single acquisition starts using the modified wavelength range. The in band acquisition option is always inactive and the analysis is performed on each channel. The application displays the results and graph for the detected channels and the newly discovered analysis parameters are applied automatically to the analysis setup without displaying any confirmation message.

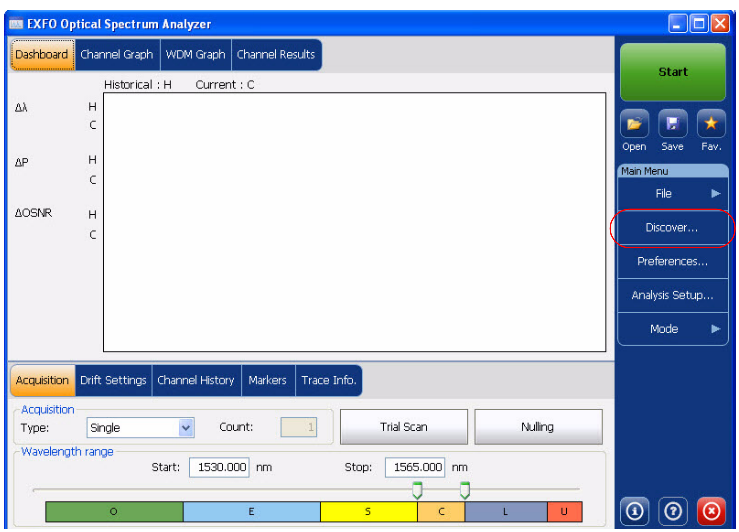
One of the limitations of using the Discover feature is that the channels are discovered based on the ITU-Grid. All detected peaks will be aligned with an ITU-channel and the channel width and channel distance are computed and fit in one of the ITU grids (25, 50, 100 or 200 GHz). If your channel is not based on the ITU grid, the results may not be correct. In this case, you can use the default channel definition or create a new channel list.

To start an automatic setup scan:

From the **Main Menu**, press **Discover**.



WDM Mode



Drift Mode

Note: *If you already have an active trace on screen that was modified, you will be prompted to save it.*

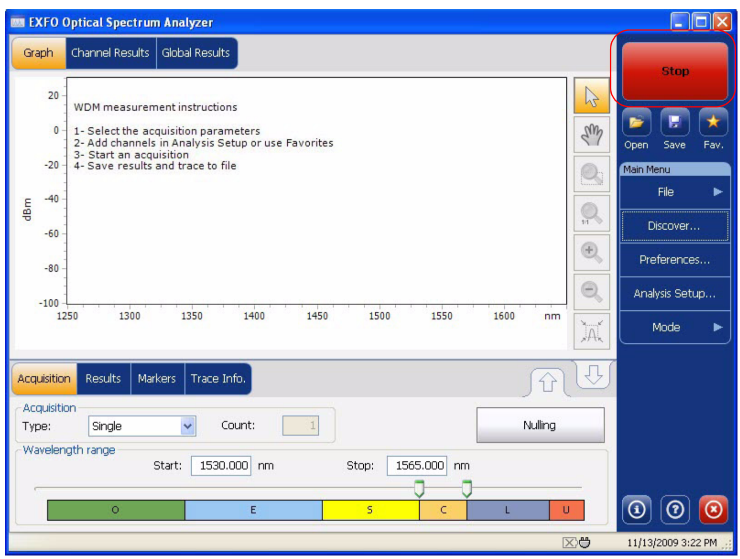
A scan starts automatically.

When the automatic scan is complete, you can start using these newly detected parameters. Simply press Start to perform another measurement with the newly found settings.

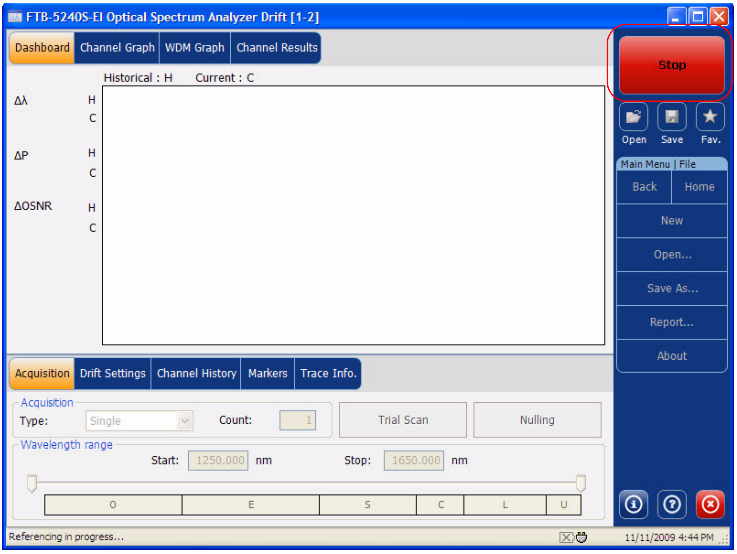
Starting a Measurement

Note: *Before starting a measurement you must select and configure a test mode. You will find the instructions to select a test mode in* Selecting a Test Mode *on page 18. For instructions on configuring various test modes refer* Setting Up the Instrument in WDM Mode *on page 37,* Setting Up the Instrument in Drift Mode *on page 103,* Setting Up the Instrument in DFB Mode *on page 165,* Setting Up the Instrument in Spectral Transmittance Mode *on page 183 and* Setting Up the Instrument in EDFA Mode *on page 209.*

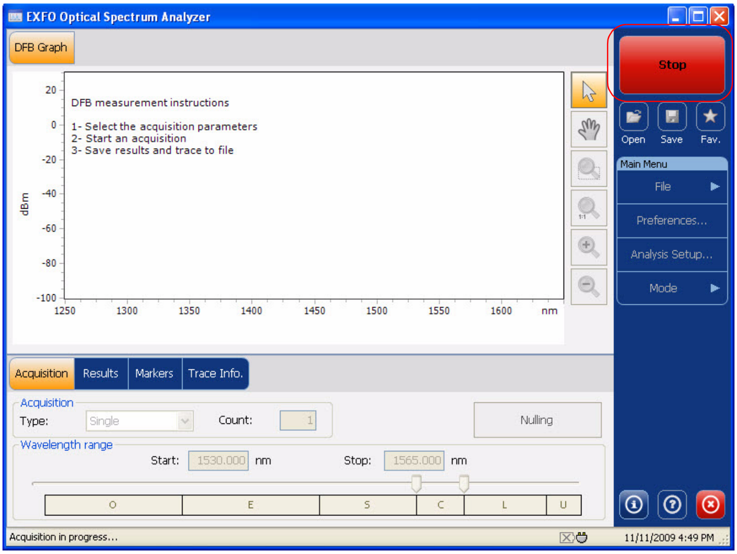
From the main window, Press **Start**.



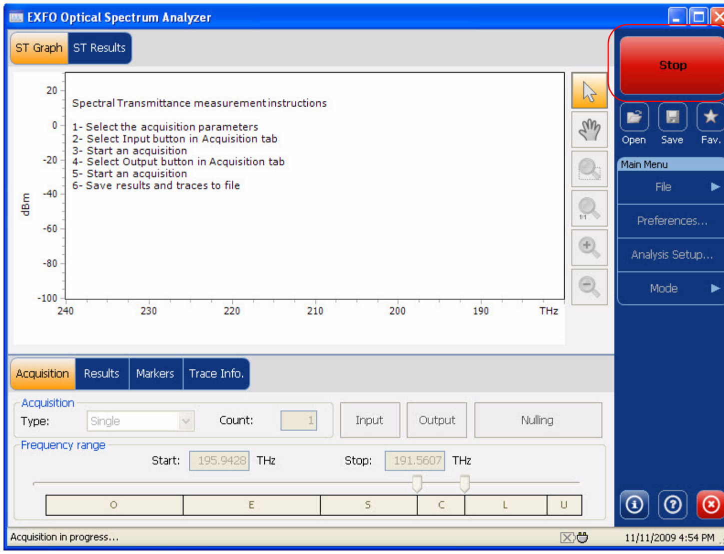
WDM Mode



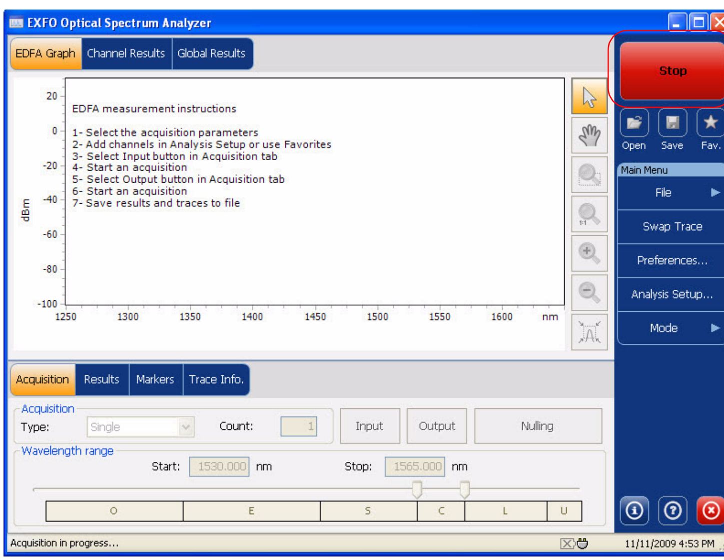
Drift Mode



DFB Mode



Spectral Transmittance Mode



EDFA Mode

When an acquisition is undergoing in the module, the **Start** button changes to **Stop**. Pressing the **Stop** button will end the acquisition.

You are notified that the acquisition is in progress in the status bar.

When the acquisition is complete, the active trace contains the graph, the result data, the acquisition and analysis setup, the trace information and all pass-fail status if activated.

The appropriate sequence to perform a measurement is:

➤ Set-up the acquisition and analysis parameters (see *Setting Up the Instrument in WDM Mode* on page 37, *Setting Up the Instrument in Drift Mode* on page 103, *Setting Up the Instrument in DFB Mode* on page 165, *Setting Up the Instrument in Spectral Transmittance Mode* on page 183 or *Setting Up the Instrument in EDFA Mode* on page 209 for more details) including channel list or use Discover feature (see *Using the Discover Feature* on page 269 for more details).

➤ Start a measurement (see *Starting a Measurement* on page 273 for more details).

➤ Manage results (see *Managing Results* on page 277 for more details).

11 Managing Results

Your Optical Spectrum Analyzer allows you to manage results in the WDM, Drift, DFB, Spectral Transmittance and EDFA modes. You can view the trace details, channel results and global results for all measured channels. The application also allows you to edit setup parameters and re-analyze the results for WDM, EDFA and Spectral Transmittance modes.

You can use zoom options on the trace, configure markers to view the power values for specific wavelengths, and view trace information.

You can also manage trace files (save, open) and generate reports for all test modes.

Note: *When a power result is flagged using a star character (*), it means the detector is saturated. When the optical power on the detector is too high, the detector gets saturated and the returned value is probably incorrect.*

Note: *When an OSNR or a noise result is flagged using a question mark character (?), it means the polarization discrimination is insufficient to make a valid OSNR calculation. This indication can only be produced when making an In-Band measurement. This indication can arise in the following situations:*

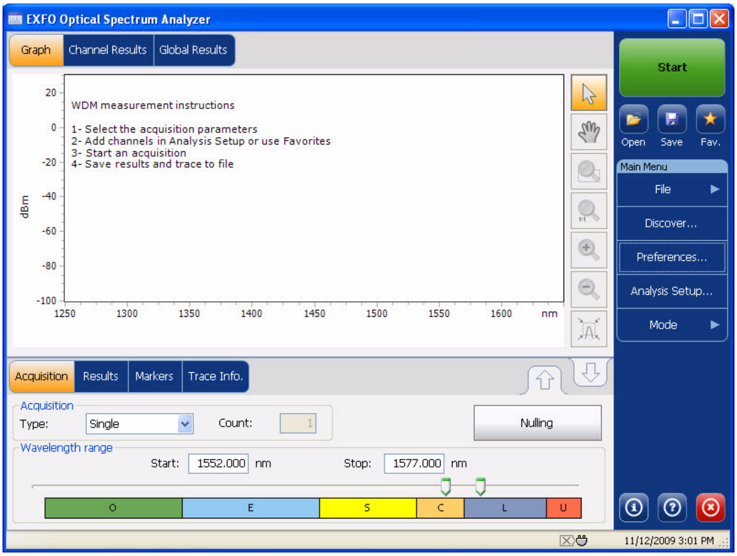
- In-Band averaging was made using a very few scans (1 or 2 for example). Usually produced when the operator presses the **Stop** button before the end of the In-Band acquisition.
- The data in the measured channel is subjected to fast polarization scrambling.
- The channel is a POLMUX channel.
- One channel might have a misfit with the current state of the polarization scrambler inside the OSA module. Can be corrected by moving the fiber at the entrance of the module.
- The polarization scrambler inside the OSA module could be defective. If you suspect this option please contact EXFO technical support for a more complete diagnosis.

Managing WDM Test Results

The application allows you to view and manage your WDM test results. You can view the graph of your acquisition, results for a single channel, global results and information about the trace.

Graph Tab

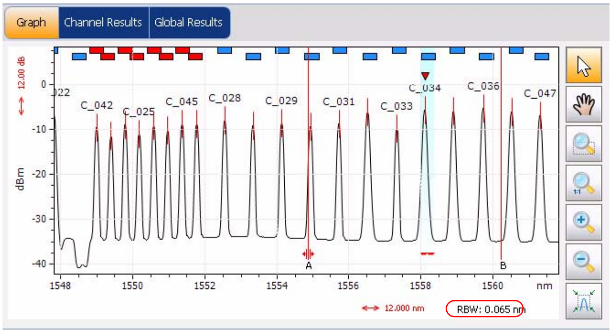
The **Graph** tab allows you to view the spectrum of the active trace. This graph represents the optical power against wavelength. When there is no acquisition, the graph zone shows a procedure explaining how to take measurements with the OSA application.



When the acquisition is taken (see *Starting a Measurement* on page 273 for details on how to perform a test), the trace will be displayed in the **Graph** tab with information along the following axis values:

➤ **X axis**: wavelength in nm or frequency in THz

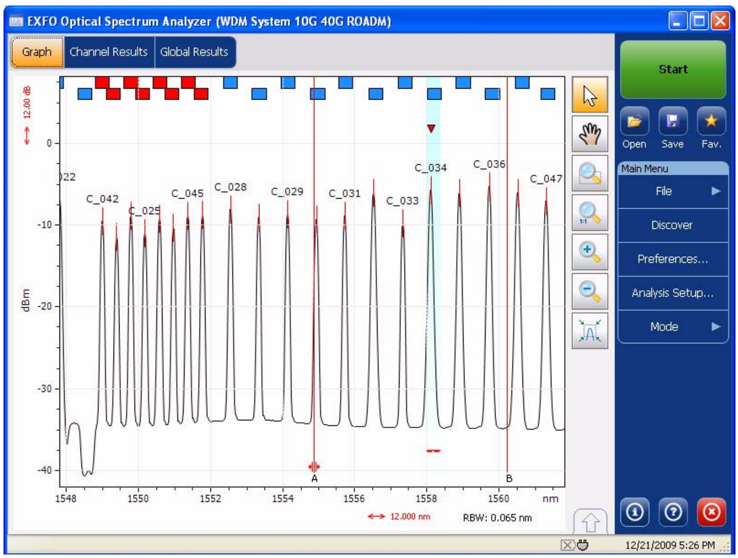
➤ **Y axis**: optical power expressed in dBm, as measured in the optical resolution bandwidth (RBW) of the OSA. This reference RBW is shown at the bottom right of the graph.



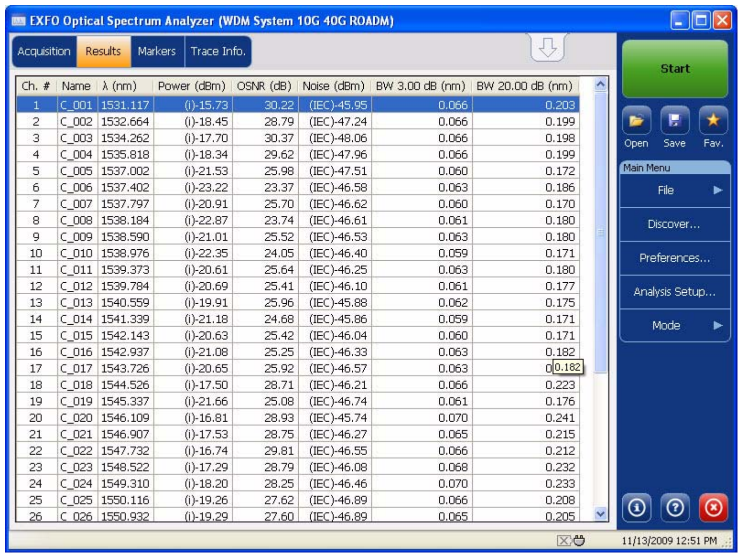
If the current trace was previously saved, the graph will display the file name of the current trace in the title bar.

Your application allows you to toggle the view of your WDM main window. You can change the view of the upper and lower tabs from the normal view to 100 % upper tabs or 100 % lower tabs view.

For 100 % upper tabs view, press .



For 100 % lower tabs view, press .

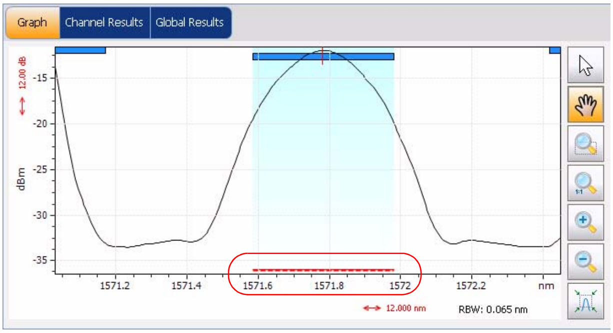


Ch. #	Name	λ (nm)	Power (dBm)	OSNR (dB)	Noise (dBm)	BW 3.00 dB (nm)	BW 20.00 dB (nm)
1	C_001	1531.117	(i)-15.73	30.22	(IEC)-45.95	0.066	0.203
2	C_002	1532.664	(i)-18.45	28.79	(IEC)-47.24	0.066	0.199
3	C_003	1534.262	(i)-17.70	30.37	(IEC)-48.06	0.066	0.198
4	C_004	1535.818	(i)-18.34	29.62	(IEC)-47.96	0.066	0.199
5	C_005	1537.002	(i)-21.53	25.98	(IEC)-47.51	0.060	0.172
6	C_006	1537.402	(i)-23.22	23.37	(IEC)-46.58	0.063	0.186
7	C_007	1537.797	(i)-20.91	25.70	(IEC)-46.62	0.060	0.170
8	C_008	1538.184	(i)-22.87	23.74	(IEC)-46.61	0.061	0.180
9	C_009	1538.590	(i)-21.01	25.52	(IEC)-46.53	0.063	0.180
10	C_010	1538.976	(i)-22.35	24.05	(IEC)-46.40	0.059	0.171
11	C_011	1539.373	(i)-20.61	25.64	(IEC)-46.25	0.063	0.180
12	C_012	1539.784	(i)-20.69	25.41	(IEC)-46.10	0.061	0.177
13	C_013	1540.559	(i)-19.91	25.96	(IEC)-45.88	0.062	0.175
14	C_014	1541.339	(i)-21.18	24.68	(IEC)-45.86	0.059	0.171
15	C_015	1542.143	(i)-20.63	25.42	(IEC)-46.04	0.060	0.171
16	C_016	1542.937	(i)-21.08	25.25	(IEC)-46.33	0.063	0.182
17	C_017	1543.726	(i)-20.65	25.92	(IEC)-46.57	0.063	0 0.182
18	C_018	1544.526	(i)-17.50	28.71	(IEC)-46.21	0.066	0.223
19	C_019	1545.337	(i)-21.66	25.08	(IEC)-46.74	0.061	0.176
20	C_020	1546.109	(i)-16.81	28.93	(IEC)-45.74	0.070	0.241
21	C_021	1546.907	(i)-17.53	28.75	(IEC)-46.27	0.065	0.215
22	C_022	1547.732	(i)-16.74	29.81	(IEC)-46.55	0.066	0.212
23	C_023	1548.522	(i)-17.29	28.79	(IEC)-46.08	0.068	0.232
24	C_024	1549.310	(i)-18.20	28.25	(IEC)-46.46	0.070	0.233
25	C_025	1550.116	(i)-19.26	27.62	(IEC)-46.89	0.066	0.208
26	C_026	1550.932	(i)-19.29	27.60	(IEC)-46.89	0.065	0.205

The graph will display peak indicators for all the channels found by the application. A blue horizontal bar ( ) will be displayed on the top of the channels, with a red horizontal line over the peaks to indicate the peak position.

The graph will also display the noise level for a channel with a doted line under the selected peak. The width of the noise level indicator shall be set according to the current **Noise for OSNR** setting.

- If **Noise for OSNR** is IEC, then the width of the noise level indicator will be equal to the full width of the channel.
- If **Noise for OSNR** is InB, then the width of the noise level indicator will be 50 GHz or the full width of the channel, whichever is the narrowest.
- If **Noise for OSNR** is InB nf, then the width of the noise level indicator will be 25 GHz or the full width of the channel, whichever is the narrowest.



You can also perform the following on your graph:

- Move markers on the graph.
- Use zoom functions to zoom on specific areas or peaks of the graph.

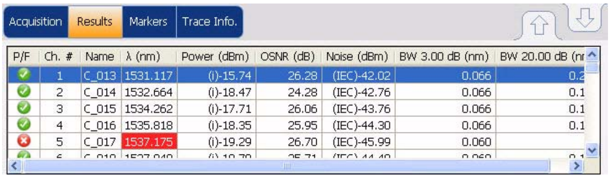
Note: *For more information on these, see* Markers Tab *on page 285 and* Using Zoom Controls *on page 396.*

Results Tab

The results table shows channel results for the active trace. The results for only the channels within the scan range will be displayed. The pass/fail verdict for thresholds are also displayed in the results table. If the verdict is fail for any parameter, its value will appear in red.

To view results:

From the main window, select the **Results** tab.



The following information related to the channels is displayed in the **Results** table:

➤ **P/F**: indicates Pass/Fail status of the channel.

: indicates Pass status.

: indicates Fail status, meaning at least one parameter failed.

➤ **Ch#**: indicates the channel number. Channel 1 corresponds to the first channel found.

➤ **Name**: indicates the channel name as defined in the channel list.

➤ **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.

➤ **Power (dBm)**: indicates the signal power for the selected channel (excludes noise) (according to current calculation method, in dBm).

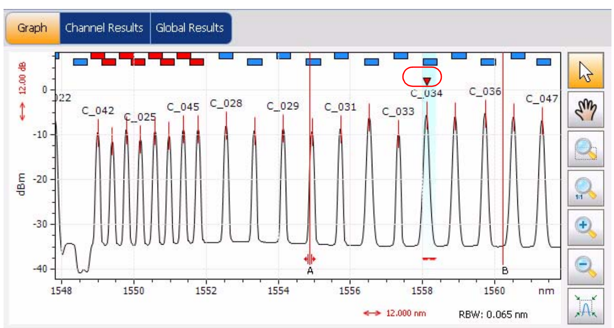
➤ **OSNR (dB)**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).

➤ **Noise (dBm)**: indicates the level of the noise for the selected channel (according to the current calculation method, in dBm).

➤ **BW 3.00 dB (nm)**: indicates bandwidth measured by taking the width of a signal at 50 % linear power of the peak or -3 dB from the peak.

➤ **BW x dB (nm)**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.

A small red marker (▼) will point down at the peak in the **Graph** tab when you select a row in the **Results** tab. The red marker will move accordingly to indicate the corresponding peak on the graph, with a focus on the selected channel.



Markers Tab

You can use markers to perform manual measurements and verification directly on the trace. There are two vertical and two horizontal markers indicating the wavelength and power positions respectively, which you can also move directly on the trace using the zoom select option. You can measure actual power and wavelength values of any point on the trace.

Note: *These markers are visible in the **Graph** tab only when the markers tab is selected in the main window.*

Note: *Horizontal markers will be displayed only if the markers are activated in the **Preferences** tab.*

When the vertical markers are set, you will find two markers identified by the letters A and B in the **Graph** tab. Horizontal markers are identified by letters C and D. These markers are represented both in the graph and the markers tab to customize your markers.

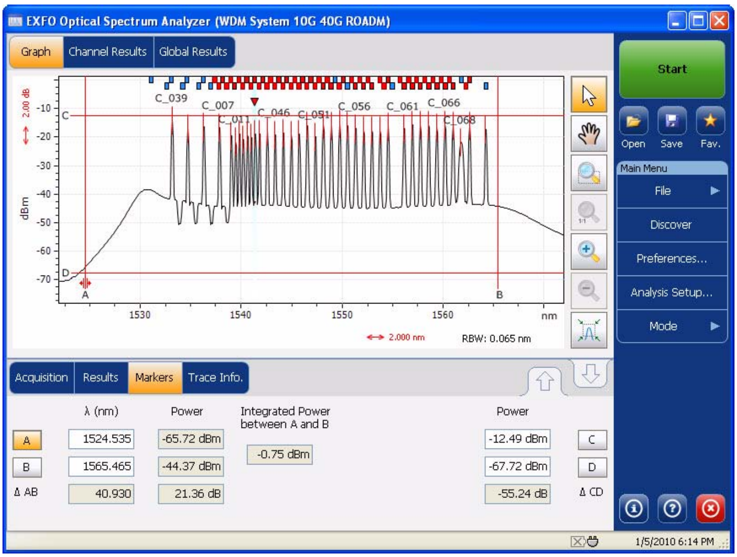
The application allows you to fix the distance between the vertical markers. When this feature is activated, while moving any one marker, both markers shall move at the same rate and distance.

The markers A and B in the **Markers** tab act as toggle buttons to enable selection. When a marker is activated, the colour of the button changes to orange and the selected marker displays a double arrow at the base of the marker in the **Graph** tab, which means that the marker can be moved.

At this point, if you press the other vertical marker in the **Graph** tab, the selection of the toggle switches to this marker. However, if you select the other marker button from the **Markers** tab, both the markers will be selected and the distance between both will be locked. When both the vertical markers are locked for fixed movement, pressing any horizontal marker from the **Markers** tab or the **Graph** tab will remove the selection of both the vertical markers.

To define a marker:

1. From the main window, press the **Markers** tab.
2. Set the marker by entering precise values in the **A**, **B**, **C**, and **D** boxes.



Vertical markers A and B appears on the graph and the following values related to the vertical markers will be displayed in the **Markers** tab.

- power value corresponding to the wavelength position of both the markers.
- wavelength difference between the markers (A-B)
- power difference in dB between the markers
- integrated power between markers in dBm.

Horizontal markers C and D appears on the graph and the power difference between the markers (C-D) related to the horizontal markers will be displayed in the **Markers** tab.

You can also move the markers directly on the **Graph** tab. Drag the marker to the desired area in the display. You will notice that the corresponding box in the **Markers** tab changes according to the marker's position. If you want to set a precise value for the marker, simply enter it in the box.

Note: *After using the zoom tools in the **Graph** tab, you can only move the markers again on the graph after deactivating the zoom. Pressing the arrow in the zoom tools section will deactivate the zoom function.*

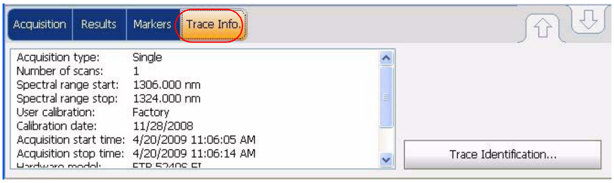
Note: *Markers A and B cannot cross. Displacing a marker over the second one will cause both markers to move together.*

Trace Info. Tab

After acquiring a trace, you might want to view details about the acquisition. The **Trace Info.** tab shows information related to acquisition parameters and conditions. You can also edit information about the tested fiber and job or add comments. This information is saved along with the trace.

To view trace information parameters:

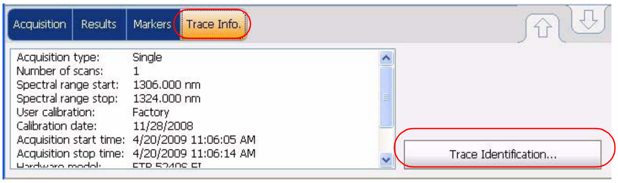
From the main window, select the **Trace Info.** tab.



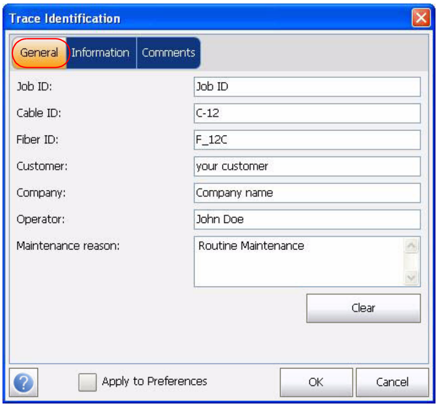
To edit general information:

1. From the main window, select the **Trace Info.** tab.

2. Press **Trace Identification**.



3. Select the **General** tab.



4. Edit the general information as required.

Note: *If you want to apply the above settings to the **Preferences** tab, select the **Apply to Preferences** option and press **OK**.*

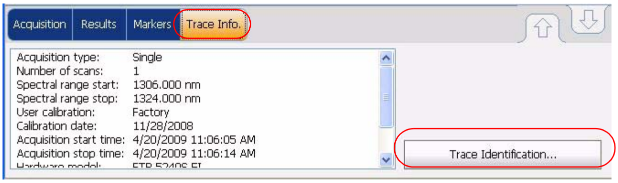
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Clear** to clear all the changes made in the **General** tab.

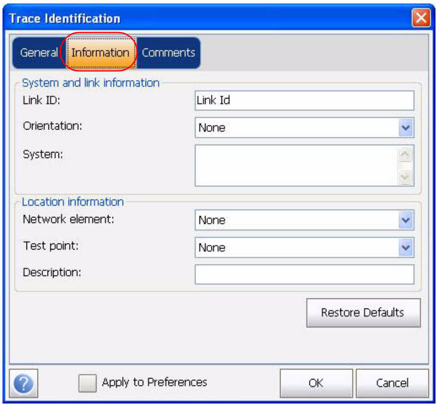
To edit trace information:

1. From the main window, select the **Trace Info.** tab.

2. Press **Trace Identification**.



3. Select the **Information** tab.



4. Edit the information as required.

Note: *If you want to apply the above settings to the **Preferences** tab, select the **Apply to Preferences** option and press **OK**.*

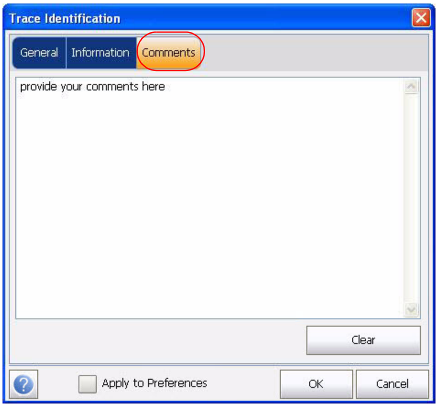
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

To edit comments:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.

3. Select the **Comments** tab.



4. Edit comments in the **Comments** window for the current trace.

Note: *If you want to apply the above settings to the preferences tab, select the **Apply to Preferences** option and press **OK**.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

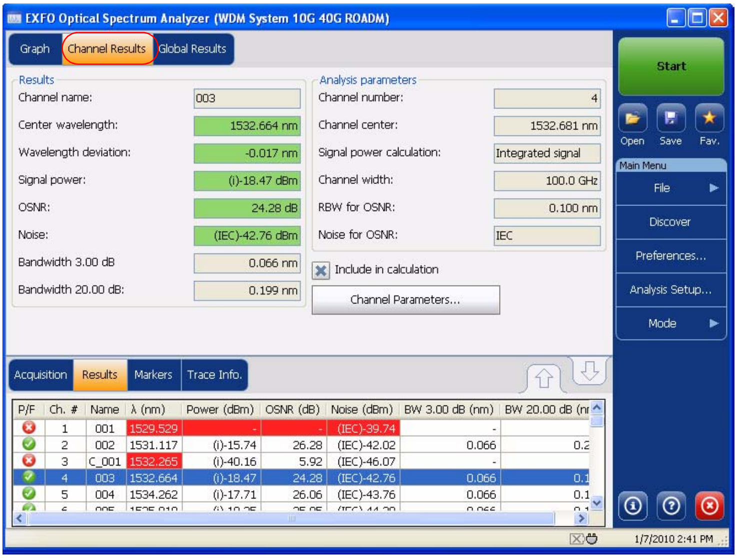
Press **Clear** to clear all the changes made in the **Comments** tab.

Channel Results Tab

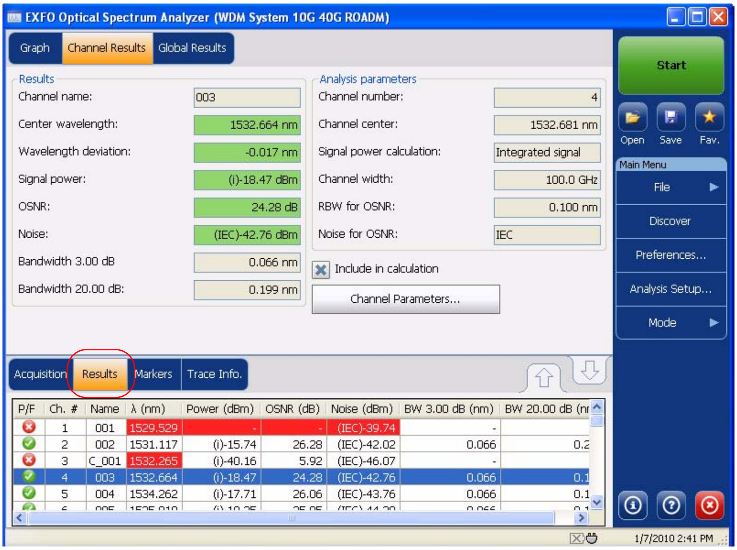
When you select a row from the **Results** tab, the **Channel Results** tab will show complete information about the parameters measured for the selected channel. The pass/fail verdict for thresholds are also displayed in the **Channel Results** tab. If the verdict is fail for any parameter, its value will appear in red. If the verdict is pass, its value will appear in green.

To view channel results:

1. From the main window, select **Channel Results** tab.



2. Select a row from the **Results** tab to view the channel results for the selected channel.



➤ **Channel name**: indicates the name of the channel as defined in the channel list.

➤ **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.

➤ **Wavelength/Frequency deviation**: indicates the difference between the channel center wavelength/frequency and the measured signal center wavelength/frequency.

➤ **Signal power**: indicates the signal power for the selected channel (excludes noise).

➤ **OSNR**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).

➤ **Noise**: indicates the level of the noise for the selected channel.

- **Bandwidth 3.00 dB**: indicates bandwidth measured by taking the width of a signal at 50 % linear power of the peak or -3 dB from the peak.
- **Bandwidth x dB**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.
- **Channel number**: indicates the number of the channel. Channel 1 corresponds to the first channel found.
- **Channel center**: indicates the channel center for the selected channel.
- **Signal power calculation**: indicates which calculation method to apply for signal power value.

  Integrated signal power: The integrated signal power represents the sum of the power values included between the channel limits of this channel, minus the estimated noise contribution between the same boundaries. In some cases, for instance CATV signals, signals with high-frequency modulation, or signals with an inherent line width similar or larger than the OSA's resolution bandwidth, this calculation becomes a better estimation of the true signal power.

  Peak signal power: The peak signal power represents the maximum power value inside the channel, but it differs a little from the peak measurement on the spectrum due to the fact that the estimated noise is subtracted to get the peak signal power.

  Total channel power: The total channel power represents the sum of the power values included inside the channel width (including noise).
- **Channel width**: indicates the limit within which the power values will be considered in the channel.
- **RBW for OSNR**: indicates the resolution bandwidth for the selected OSNR value.

- **Noise for OSNR**: indicates which calculation method to use for OSNR value.

  Fixed range IEC based (IEC): The IEC method uses the interpolation of noise measured on both sides of the signal to estimate the noise level. The position at which the noise is estimated from the center wavelength is given by the OSNR distance.

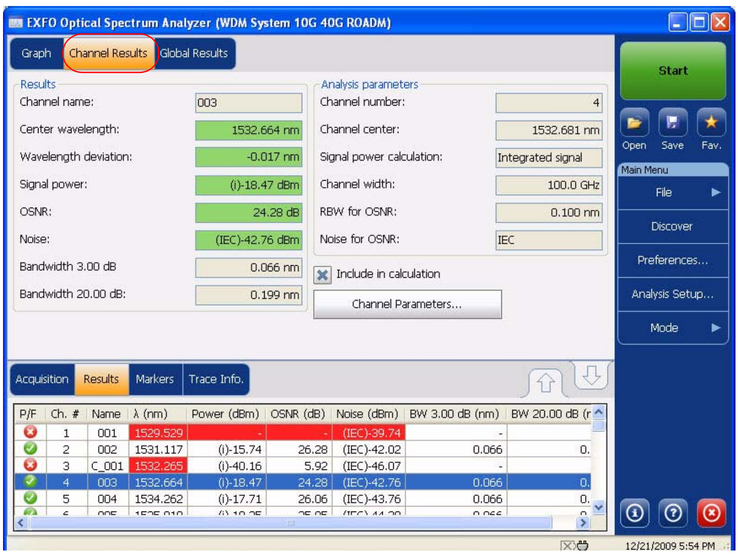
  In-Band (InB): The In-Band method uses a series of scans having different polarization states to calculate the noise level in band.

  In-Band narrow filter (InB nf): The In-Band narrow filter method uses the In-Band method with additional processing to provide an accurate OSNR value for the narrow carved noise. This is because with narrow filters, the noise level under the peak is not uniform and the OSNR value depends on the processing width selected.

You can include selected channels results in the global results. These channels will be integrated in the average signal power, average OSNR, flatness, etc. in the global results and will be enabled in the **Results** tab.

To include channels in the global measurement results:

1. From the main window, select **Channel Results** tab.



2. Select a row from the **Results** tab to view the channel results for the selected channel.

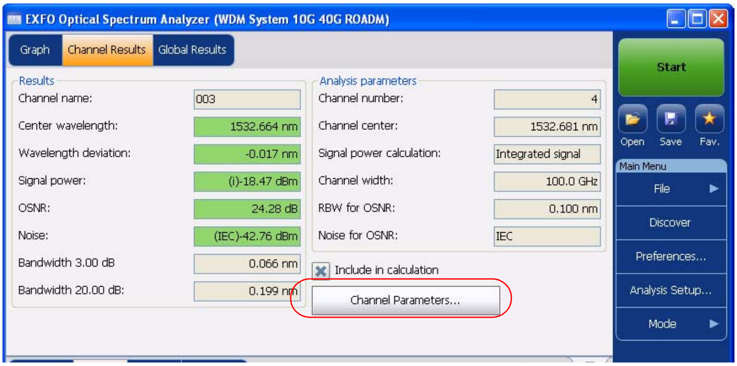
P/F	Ch. #	Name	λ (nm)	Power (dBm)	OSNR (dB)	Noise (dBm)	BW 3.00 dB (nm)	BW 20.00 dB (r
	1	001	1529.529	-	-	(IEC)-39.74	-	
	2	002	1531.117	(i)-15.74	26.28	(IEC)-42.02	0.066	0.
	3	C_001	1532.265	(i)-40.16	5.92	(IEC)-46.07	-	
	4	003	1532.664	(i)-18.47	24.28	(IEC)-42.76	0.066	0.
	5	004	1534.262	(i)-17.71	26.06	(IEC)-43.76	0.066	0.

3. Select the **Include in calculation** option.

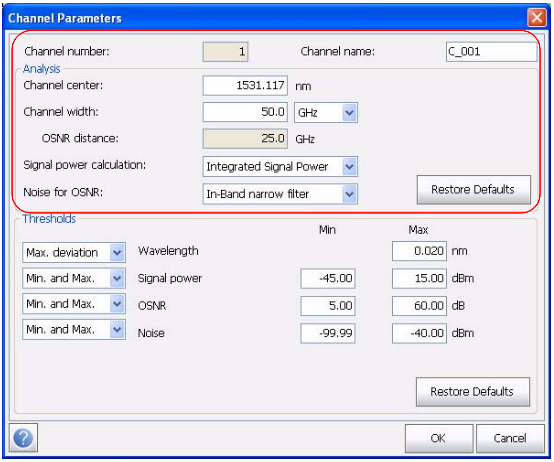
To re-analyze channel results:

1. From the main window, select the **Channel Results** tab.

2. Press **Channel Parameters**.



3. Edit the values as explained below:



- **Channel number**: indicates the number of the channel. This field value cannot be edited.
- **Channel name**: indicates the name of the channel.
- **Channel Center (nm or THz)**: indicates the central wavelength/frequency for the channel.
- **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.
- **OSNR distance (GHz or nm)**: OSNR distance for IEC interpolation method is automatically set at the channel edge, that is, at the half of the channel width from the center wavelength.

- **Signal power calculation**: indicates which calculation method to apply for signal power value.

  Integrated signal power: The integrated signal power represents the sum of the power values included between the channel limits of this channel, minus the estimated noise contribution between the same boundaries. In some cases, for instance CATV signals, signals with high-frequency modulation, or signals with an inherent line width similar or larger than the OSA's resolution bandwidth, this calculation becomes a better estimation of the true signal power.

  Peak signal power: The peak signal power represents the maximum power value inside the channel, but it differs a little from the peak measurement on the spectrum due to the fact that the estimated noise is subtracted to get the peak signal power.

  Total channel power: The total channel power represents the sum of the power values included inside the channel width (including noise).

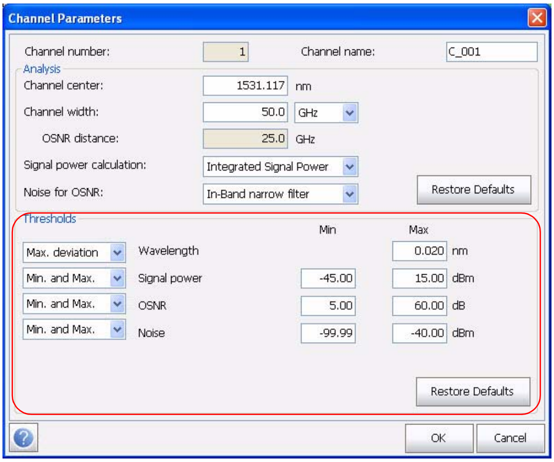
- **Noise for OSNR**: indicates which calculation method to use for OSNR value.

  Fixed range IEC based (IEC): The IEC method uses the interpolation of noise measured on both sides of the signal to estimate the noise level. The position at which the noise is estimated from the center wavelength is given by the OSNR distance.

  In-Band (InB): The In-Band method uses a series of scans having different polarization states to calculate the noise level in band.

  In-Band narrow filter (InB nf): The In-Band narrow filter method uses the In-Band method with additional processing to provide an accurate OSNR value for the narrow carved noise. This is because with narrow filters, the noise level under the peak is not uniform and the OSNR value depends on the processing width selected.

4. Edit the thresholds as explained below:



➤ **Wavelength/Frequency (nm/GHz)**: indicates the channel's central wavelength/frequency.

➤ **Signal power (dBm)**: indicates the signal power for the selected channel (excludes noise).

➤ **OSNR (dB)**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).

➤ **Noise (dBm)**: indicates the level of the noise for the selected channel.

Note: *If a channel is already included in the channel list, the application will ask if you want to replace the threshold and analysis parameters for that channel. Select **Yes**, **No** or **Cancel** as required. If the channel is not present in the channel list, it will be added provided the number of channels does not exceed the maximum allowed count of 1000 channels in the channel list.*

Note: *The application will re-analyze the modified channel only with the new channel parameters.*

5. Press **OK** to save the changes for the active trace, re-analyze and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Global Results Tab

The application allows you to view the global results of the current measurement. You can also edit global parameters to reanalyze the active trace. The pass/fail verdict for thresholds are displayed in the **Global Results** tab. If the verdict is fail for any parameter, its value will appear in red. If the verdict is pass, its value will appear in green.

To view global results:

From the main window, select the **Global Results** tab.

Results for the following parameters for all the channels will be displayed:

- **Average signal power**: indicates the sum of the signal powers of all the peaks detected in the current acquisition, divided by the total number of peaks.
- **Signal power flatness**: indicates the difference between the maximum and minimum signal power values of the detected peaks, in dB.
- **Average OSNR**: indicates the sum of the entire OSNR of the peaks detected in the current acquisition, divided by the total number of peaks.
- **OSNR flatness**: indicates the difference between the maximum and minimum OSNR values of the detected peaks, in dB.

- **Total power in scan range**: indicates the total power for the selected scan range.
- **Channel count**: indicates the number of channels detected after an acquisition for the selected scan range.
- **Empty channel count**: indicates the number of empty channels from the channel list.
- **Peak detection level**: indicates the minimum power level from where the peak can be considered as a signal.
- **RBW for OSNR**: indicates the resolution bandwidth for the selected OSNR value.
- **Wavelength offset**: indicates the offset value applied on the wavelength.
- **Power offset**: indicates the offset value applied on the power.

In addition, you can also view the global pass/fail status, provided the thresholds are activated in the **Global Result Thresholds** tab in the **Analysis Setup** window. If the thresholds are disabled, the **Global pass/fail status** pane will display **Not Active**. If the thresholds are activated, the **Global pass/fail status** pane will display either a **Pass** or **Fail** based on the global results.

Note: *To edit global parameters, see* Changing Active Trace Analysis Parameters and Re-analyzing *on page 306.*

Changing Active Trace Analysis Parameters and Re-analyzing

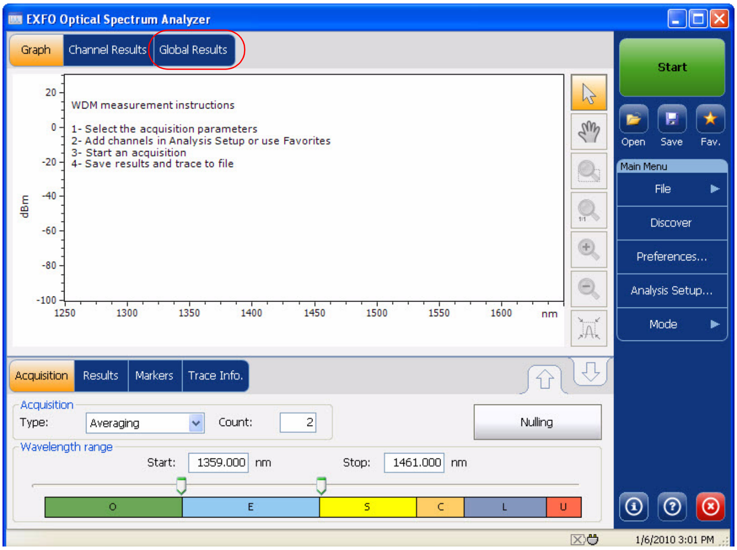
The global parameters modifies the active trace. These changes pertain to the analysis, thresholds and the channel parameters. After changing the parameters, you can analyze the trace again to view the results.

Changing Global Analysis Parameters

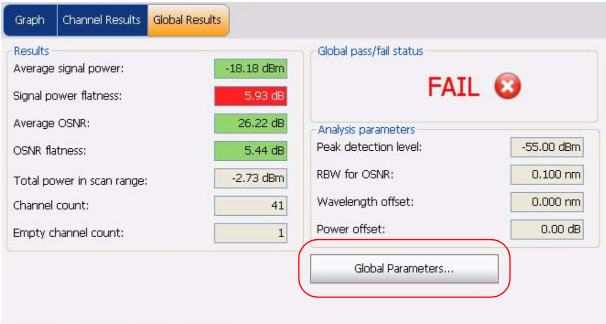
Note: *The modifications made to the global analysis parameters will be applied to the active trace only.*

To change analysis parameters:

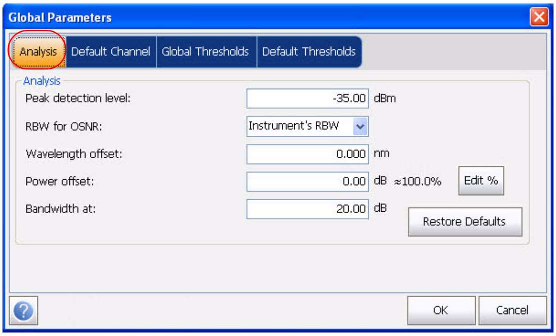
1. From the main window, select the **Global Results** tab.



2. Press **Global Parameters**.



3. Select the **Analysis** tab.



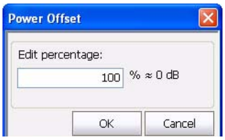
4. Enter values in the boxes as explained below:

➤ **Peak detection level (dBm)**: indicates the minimum power level from where the peak can be considered as a signal.

➤ **RBW for OSNR (nm)**: indicates the resolution bandwidth for the selected OSNR value. The resolution bandwidth of an OSA determines its ability to deal with close optical channel spacing. It is measured as the width of the response curve at half peak power (i.e., 3 dB down) of the instrument to a monochromatic test signal. The instrument's RBW value is written below the graph on the right (below wavelength offset).

- **Wavelength offset (nm)**: indicates the offset value applied on the wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.
- **Power offset (dB)**: indicates the offset value applied on the power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the ↕ arrow.

  To edit the power offset value in percentage, press **Edit %** button.

  The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.
- **Bandwidth at (dB)**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.

5. Press **OK** to save the changes, re-analyze the trace and close the window, or press **Cancel** to exit without saving.

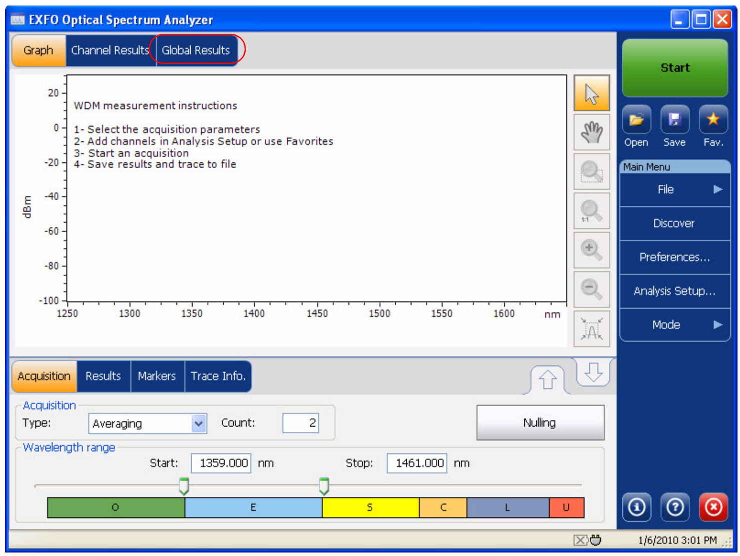
Press **Restore Defaults** to remove all the changes and apply the default values.

Changing Default Channel Parameters

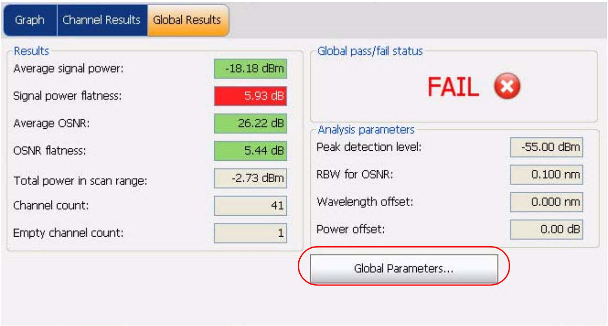
Default channel settings will define how any channel found outside the channel list will be analyzed.

To change default channel parameters:

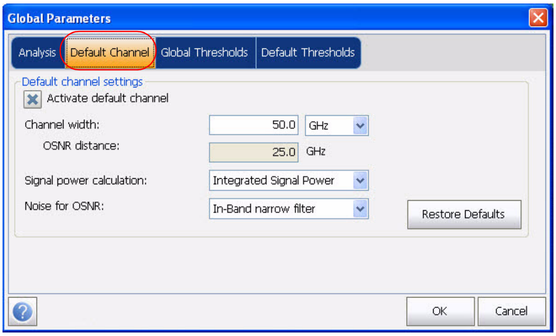
1. From the main window, select the **Global Results** tab.



2. Press **Global Parameters**.



3. Select the **Default Channel** tab.



4. Enter values in the boxes as explained below:

- Clear the **Activate default channel** selection, to use the currently defined channel for analysis. This reduces the analysis time by eliminating the peak detection over the complete spectral range. The peaks outside the defined channel will not be detected or analyzed.

- **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.

  For default channels, the channel width that sets the limits of the channel, should be the same as the channel distance or smaller (channel distance is defined while creating a channel list). If the channel width is not compatible with the channel spacing, either a single peak may be found for two distinct channels and two analysis would be performed and displayed for that peak, or, it is possible that two peaks may be found within the same channel and be considered as one multi-peak signal. With this result, you can use markers to find the spacing between adjacent channels or to find the channel width.

- **OSNR distance (GHz or nm)**: OSNR distance is automatically set at the channel edge, that is, at half of the channel width from the center wavelength.

- **Signal power calculation**: indicates which calculation method to apply for signal power value.

  Integrated signal power: The integrated signal power represents the sum of the power values included between the channel limits of this channel, minus the estimated noise contribution between the same boundaries. In some cases, for instance CATV signals, signals with high-frequency modulation, or signals with an inherent line width similar or larger than the OSA's resolution bandwidth, this calculation becomes a better estimation of the true signal power.

  Peak signal power: The peak signal power represents the maximum power value inside the channel, but it differs a little from the peak measurement on the spectrum due to the fact that the estimated noise is subtracted to get the peak signal power.

  Total channel power: The total channel power represents the sum of the power values included inside the channel width (including noise).

- **Noise for OSNR**: indicates which calculation method to use for OSNR value.

  Fixed range IEC based (IEC): The IEC method uses the interpolation of noise measured on both sides of the signal to estimate the noise level. The position at which the noise is estimated from the center wavelength is given by the OSNR distance.

  In-Band (InB): The In-Band method uses a series of scans having different polarization states to calculate the noise level under the peak (In-Band).

  In-Band narrow filter (InB nf): The In-Band narrow filter method uses additional processing to provide an accurate OSNR value for the narrow carved noise. This is because with narrow filters, the noise level under the peak is not uniform and the OSNR value depends on the processing width selected.

5. Press **OK** to save the changes, re-analyze data and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Changing Global Thresholds Parameters

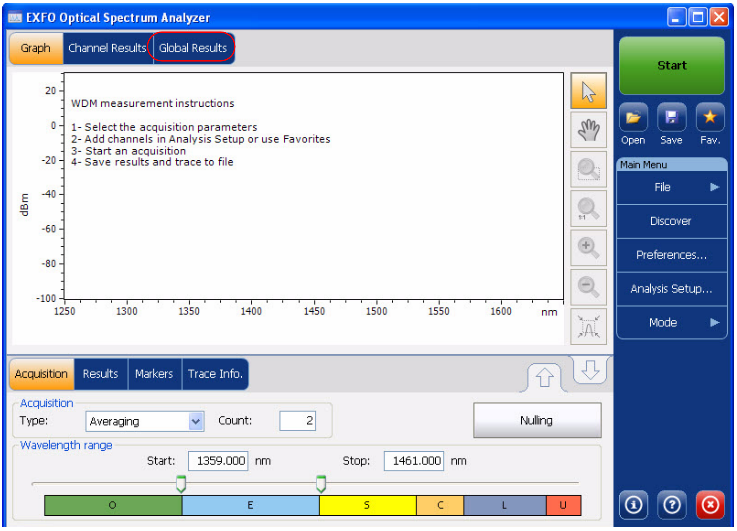
The application allows you to activate and deactivate the threshold functionality with a single control. When thresholds are globally enabled, the results are displayed with the Pass/Fail status based on various settings (global results, channel results). In addition, a global pass/fail status is also displayed in the **Global Results** tab (See *Global Results Tab* on page 304).

When thresholds are globally disabled, results are displayed without a Pass/Fail status and the **Global pass/fail status** will not be active in the **Global Results** tab. The **P/F** column under the results table will not be displayed.

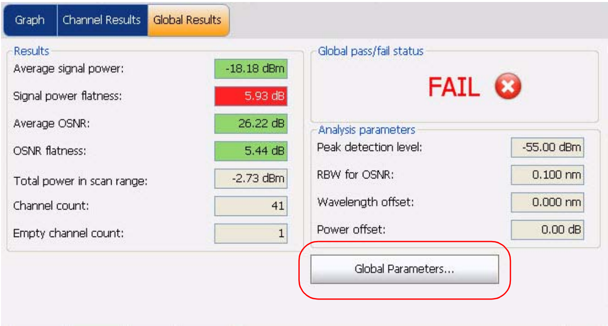
Note: *The modifications made to the global thresholds parameters will be applied to the active trace only.*

To change global threshold parameters:

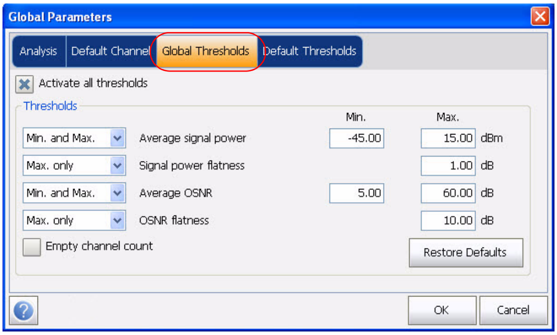
1. From the main window, select the **Global Results** tab.



2. Press **Global Parameters**.



3. Select the **Global Thresholds** tab.



4. Select the **Activate all thresholds** option, to manually set the global threshold values.

5. Enter values in the boxes as explained below:

➤ **Average signal power (dBm)**: is the sum of all signal power of all the peaks detected in the current acquisition, divided by the total number of peaks.

➤ **Signal power flatness (dB)**: is the difference between the maximum and minimum signal power values of the detected peaks, in dB.

- **Average OSNR (dB)**: indicates the sum of the entire OSNR of the peaks detected in the current acquisition, divided by the total number of peaks.
- **OSNR flatness (dB)**: is the difference between the maximum and minimum OSNR values of the detected peaks, in dB.
- **Empty channel count**: Select this option to obtain the number of empty channels from the channel list in the **Global Results** tab. If this option is not selected, it will not display the Pass/Fail verdict in the **Results** tab for the empty channels.

6. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

Changing Default Thresholds Parameters

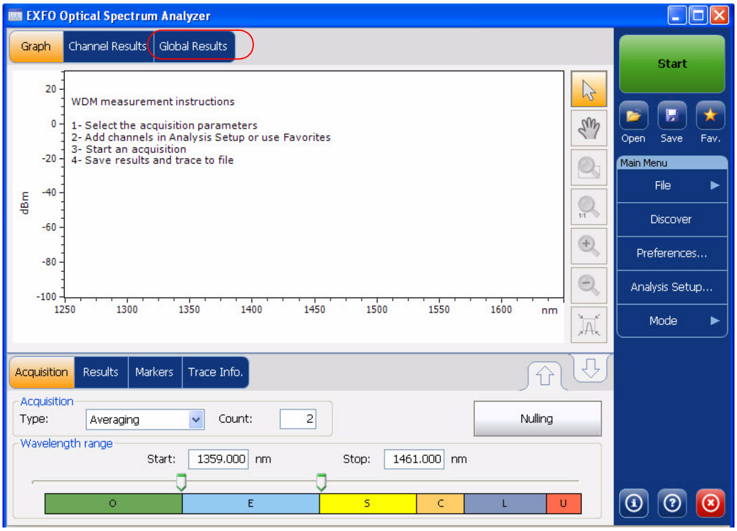
The default thresholds will be applied to any channel found outside the channel list during the current acquisition.

Note: *The default thresholds settings are enabled only when the Activate all thresholds option is selected in the Global Thresholds tab. For more information, see* Changing Global Thresholds Parameters *on page 313.*

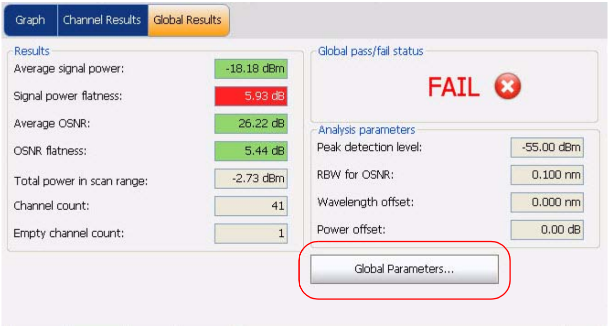
Note: *The modifications made to the default thresholds parameters will be applied to the active trace only.*

To change default threshold parameters:

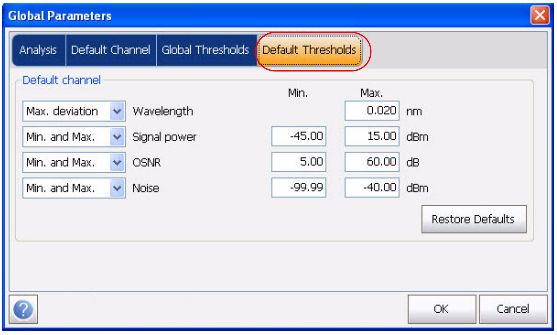
1. From the main window, select the **Global Results** tab.



2. Press **Global Parameters**.



3. Select the **Default Thresholds** tab.



4. Enter values in the boxes as explained below:
   - **Wavelength/Frequency (nm/GHz)**: indicates the channel's central wavelength/frequency.
   - **Signal power (dBm)**: indicates the signal power for the selected channel (excludes noise).
   - **OSNR (dB)**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).
   - **Noise (dBm)**: indicates the level of the noise for the selected channel.
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

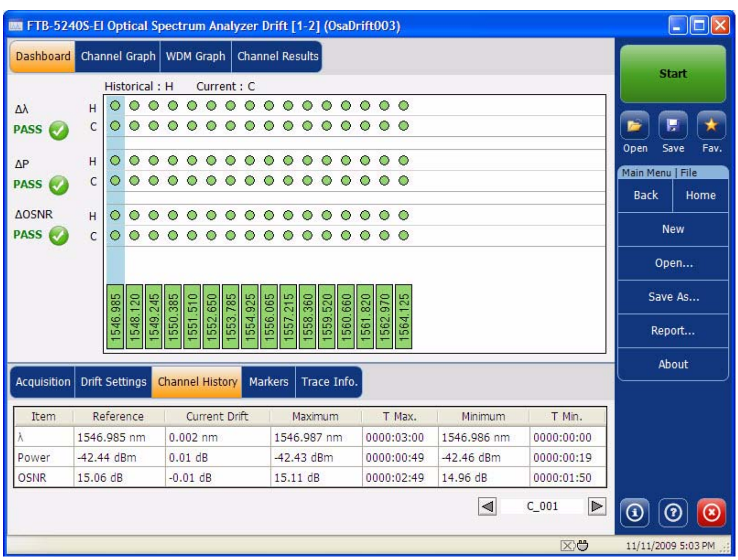
Managing Drift Test Results

The application allows you to view and manage your drift test results. You can view the dashboard, channel graph and WDM graph of your drift acquisition, channel history results for a single channel and information about the trace.

Dashboard Tab

The dashboard shall provide a way to view at-a-glance the pass/fail status of each parameter for each channel that is measured during a drift measurement. When there is no measurement, the Dashboard will be blank.

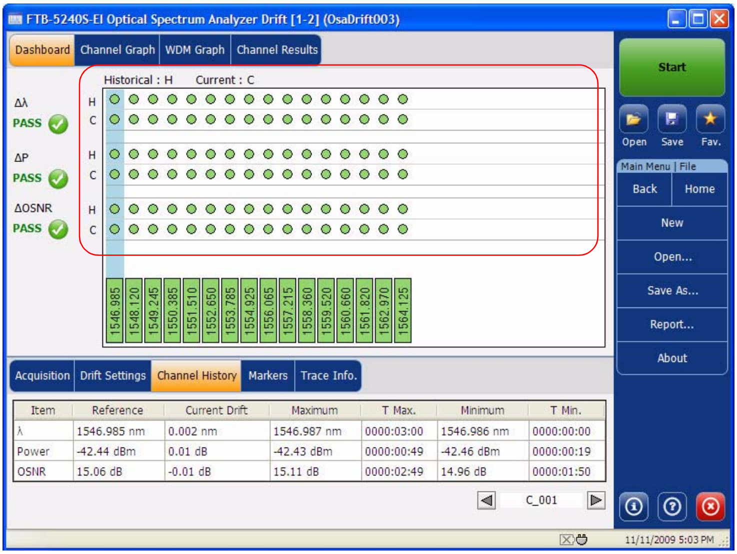
When the measurement is taken (see *Starting a Measurement* on page 273 for details on how to perform a test), the dashboard will display the results as shown below:



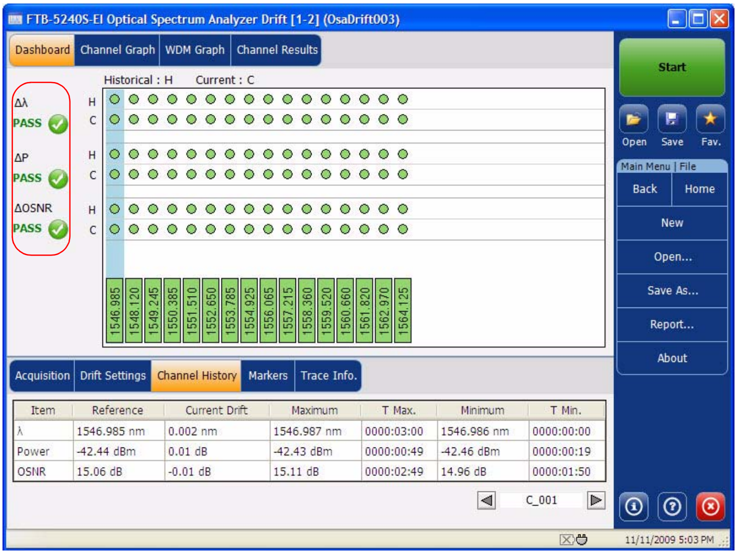
You can select a channel directly from the dashboard or from the **Channel History** tab. For each channel, the dashboard displays the pass/fail status for each of the following parameters:

- Central wavelength/frequency
- Signal power
- OSNR

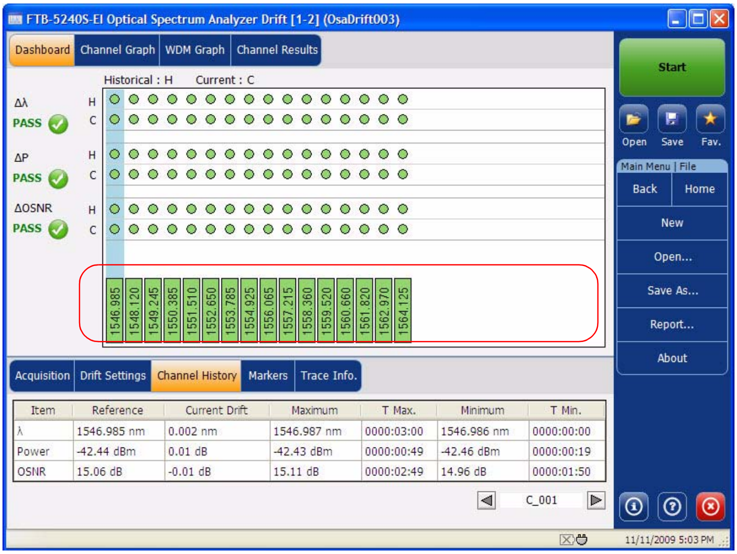
Both the current pass/fail status (last completed acquisition) and the historical pass/fail status are displayed in the dashboard. The historical pass/fail status shall be set to Fail as soon as one occurrence of fail has occurred in the past or in the current acquisition.



➤ The dashboard shows a global status (all channels) for each parameter. This global status is set to Fail if at least a single channel has a failed historical status for that given parameter, else the global status is set to Pass.



➤ The dashboard displays a channel status (all parameters) for a given channel. This channel status is set to Fail if at least one of the parameter has a failed historical status for that given channel, else the channel status is set to Pass.



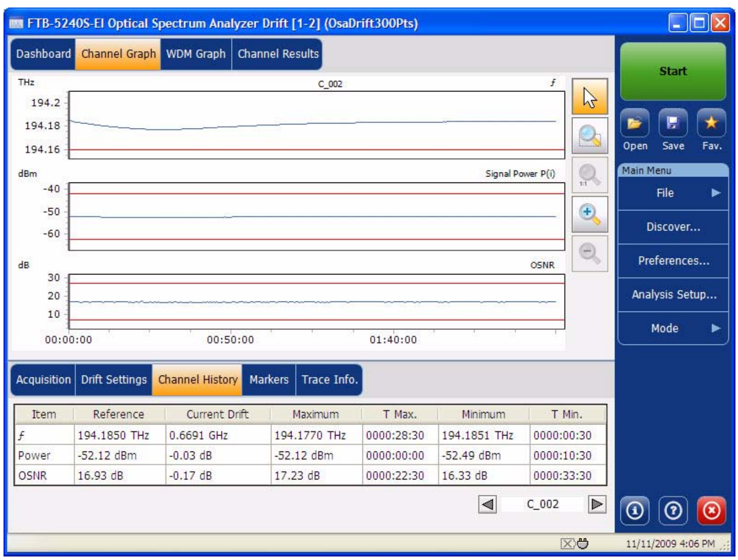
Channel Graph Tab

The **Channel Graph** tab displays three different graphs for the selected channel. You can select which graphs you want to display from the **Drift Results** tab in the **Preferences** Window. The three graphs are X-Y plots of:

➤ Spectral position (center of mass of wavelength or frequency) of the channel against time

➤ Signal power of the channel against time

➤ OSNR of the channel against time



You can also perform the following on your graph:

➤ Use zoom functions to zoom on specific areas or peaks of the graph.

Note: *For more information on zoom, see* Using Zoom Controls *on page 396.*

Channel History Tab

The Channel History table shows channel results for the active trace. The result is displayed for the selected channel only. The pass/fail verdict for thresholds are also displayed in the results table. If the verdict is fail for any parameter, its value will appear in red.

The application displays the progress of the measurement in the status bar, while the acquisition is taken. The **Elapsed time** and **Expected duration** for the measurement to stop is displayed in the **Channel History** tab.

To view channel history results:

From the main window, select the **Channel History** tab.

Item	Reference	Current Drift	Maximum	T Max.	Minimum	T Min.
f	194.1850 THz	0.6691 GHz	194.1770 THz	0000:28:30	194.1851 THz	0000:00:30
Power	-52.12 dBm	-0.03 dB	-52.12 dBm	0000:00:00	-52.49 dBm	0000:10:30
OSNR	16.93 dB	-0.17 dB	17.23 dB	0000:22:30	16.33 dB	0000:33:30

Results for the following parameters related to the selected channel are displayed in the **Channel History** table:

➤ Spectral position (center of mass of wavelength or frequency) of the channel against time (nm or THz)

➤ Signal power of the channel against time (dBM)

➤ OSNR of the channel against time (dB)

For each of the above parameters, the following results are displayed:

➤ **Reference**: indicates the channel reference values for the current drift acquired during the initial acquisition.

➤ **Current Drift**: indicates the current drift values, that is, the current deviation from the channel's reference for the drift's latest acquisition.

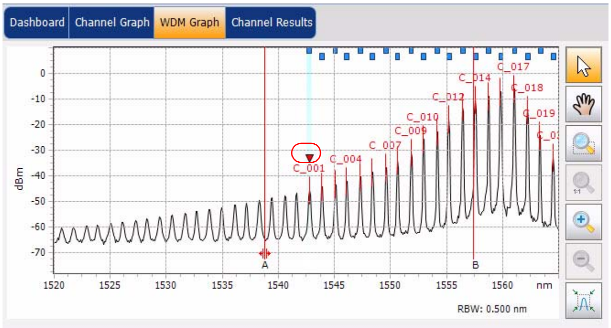
➤ **Maximum**: indicates the maximum values reached during the drift.

➤ **T Max.**: indicates the time of the drift at which the channel was at its maximum value. Displayed time is relative to the time at the start of the drift measurement.

➤ **Minimum**: indicates the minimum values reached during the drift.

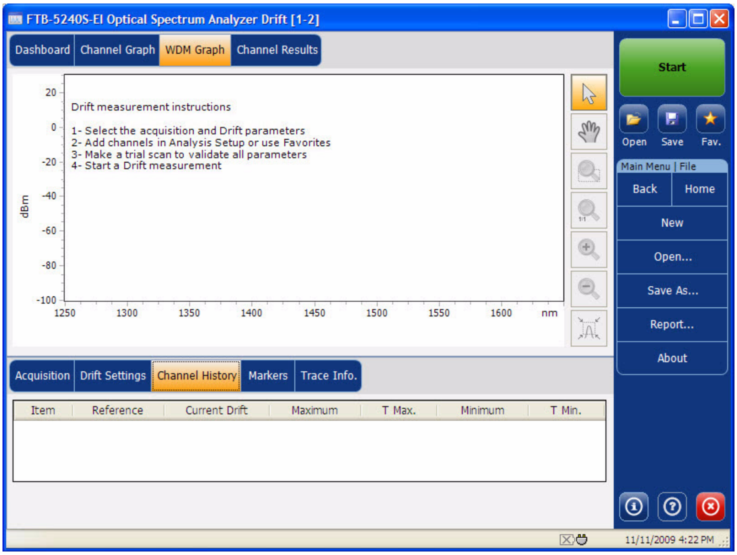
➤ **T Min.**: indicates the time of the drift at which the channel was at its minimum value. Displayed time is relative to the time at the start of the drift measurement.

A small red marker (▼) will point down at the peak in the **WDM Graph** tab when you select a channel in the **Channel History** tab. The red marker will move accordingly to indicate the corresponding peak on the graph, with a focus on the selected channel.



WDM Graph Tab

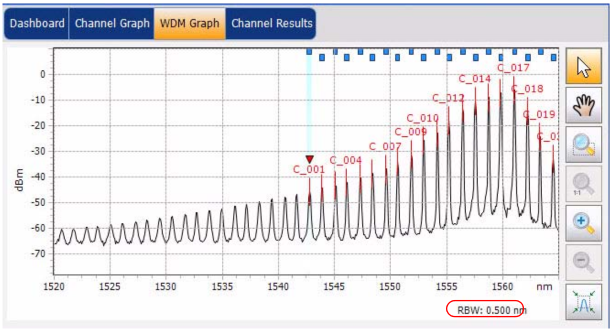
The **WDM Graph** tab allows you to view the spectrum of the active trace for the last WDM acquisition in your drift measurement. This graph represents the optical power against wavelength. When there is no acquisition, the graph zone shows a procedure explaining how to take measurements with the OSA application.



When the acquisition is taken (see *Starting a Measurement* on page 273 for details on how to perform a test), the trace will be displayed in the WDM **Graph** tab with information along the following axis values:

➤ **X axis**: wavelength in nm or frequency in THz

➤ **Y axis**: optical power expressed in dBm, as measured in the optical resolution bandwidth (RBW) of the OSA. This reference RBW is shown at the bottom right of the graph.



The graph will display peak indicators for all the channels found by the application. A blue horizontal bar ( ) will be displayed on the top of the channels, with a red horizontal line over the peaks to indicate the peak position.

You can also perform the following on your graph:

➤ Move markers on the graph.

➤ Use zoom functions to zoom on specific areas or peaks of the graph.

Note: *For more information on these, see* Markers Tab *on page 330 and* Using Zoom Controls *on page 396.*

Markers Tab

You can use markers to perform manual measurements and verification directly on the trace. There are two vertical and two horizontal markers indicating the wavelength and power positions respectively, which you can also move directly on the trace using the zoom select option. You can measure actual power and wavelength values of any point on the trace.

Note: *These markers are visible in the* ***WDM Graph*** *tab only when the markers tab is selected in the main window.*

Note: *Horizontal markers will be displayed only if the markers are activated in the* ***Preferences*** *tab.*

When the vertical markers are set, you will find two markers identified by the letters A and B in the **WDM Graph** tab. Horizontal markers are identified by letters C and D. These markers are represented both in the graph and the markers tab to customize your markers.

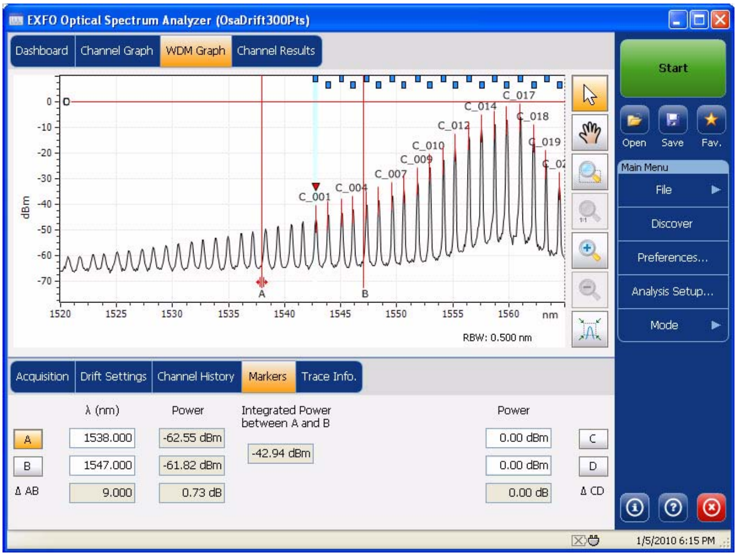
The application allows you to fix the distance between the vertical markers. When this feature is activated, while moving any one marker, both markers shall move at the same rate and distance.

The markers A and B in the **Markers** tab act as toggle buttons to enable selection. When a marker is activated, the colour of the button changes to orange and the selected marker displays a double arrow at the base of the marker in the **WDM Graph** tab, which means that the marker can be moved.

At this point, if you press the other vertical marker in the **WDM Graph** tab, the selection of the toggle switches to this marker. However, if you select the other marker button from the **Markers** tab, both the markers will be selected and the distance between both will be locked. When both the vertical markers are locked for fixed movement, pressing any horizontal marker from the **Markers** tab or the **WDM Graph** tab will remove the selection of both the vertical markers.

To define a marker:

1. From the main window, press the **Markers** tab.
2. Set the marker by entering precise values in the **A**, **B**, **C**, and **D** boxes.



Vertical markers A and B appears on the graph and the following values will be displayed in the **Markers** tab.

➤ power value corresponding to the wavelength position of both the markers.

➤ wavelength difference between the markers (A-B)

➤ power difference in dB between the markers

➤ integrated power between markers in dBm.

Horizontal markers C and D appears on the graph and the power difference between the markers (C-D) related to the horizontal markers will be displayed in the **Markers** tab.

You can also move the markers directly on the **WDM Graph** tab. Drag the marker to the desired area in the display. You will notice that the corresponding box in the **Markers** tab changes according to the marker's position. If you want to set a precise value for the marker, simply enter it in the box.

Note: *After using the zoom tools in the **WDM Graph** tab, you can only move the markers again on the graph after deactivating the zoom. Pressing the arrow in the zoom tools section will deactivate the zoom function.*

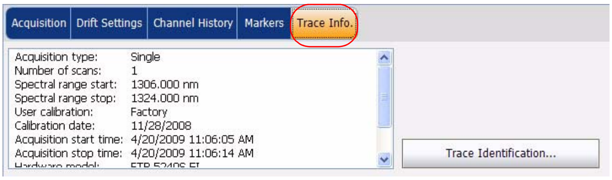
Note: *Markers A and B cannot cross. Displacing a marker over the second one will cause both markers to move together.*

Trace Info. Tab

After acquiring a trace, you might want to view details about the acquisition. The **Trace Info.** tab shows information related to acquisition parameters and conditions. You can also edit information about the tested fiber and job or add comments. This information is saved along with the trace.

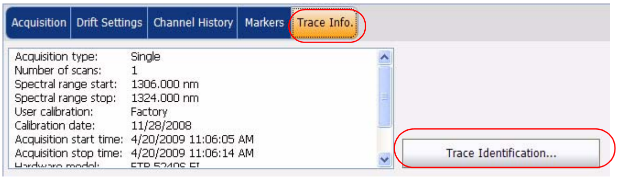
To view trace information parameters:

From the main window, select the **Trace Info.** tab.

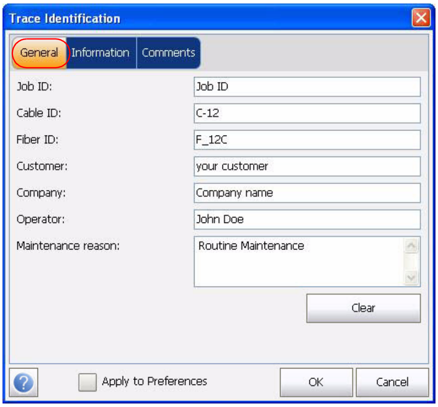


To edit general information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **General** tab.



4. Edit the general information as required.

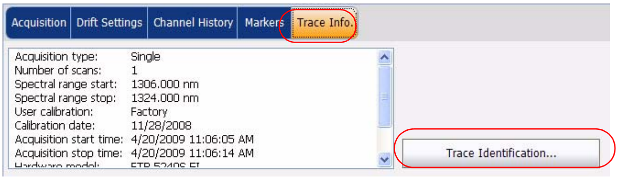
Note: *If you want to apply the above settings to the* ***Preferences*** *tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

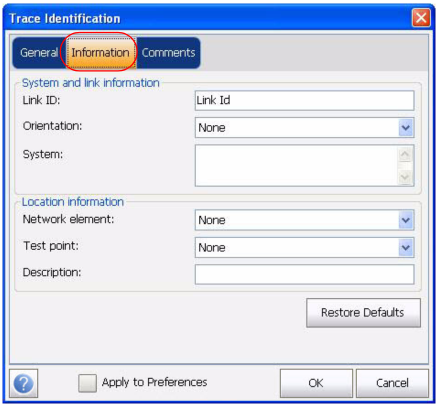
Press **Clear** to clear all the changes made in the **General** tab.

To edit trace information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **Information** tab.



4. Edit the information as required.

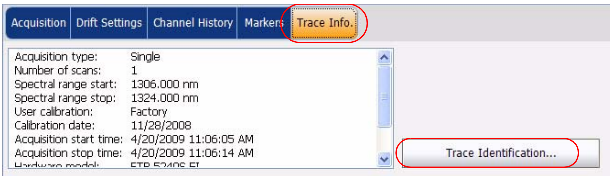
Note: *If you want to apply the above settings to the **Preferences** tab, select the **Apply to Preferences** option and press **OK**.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

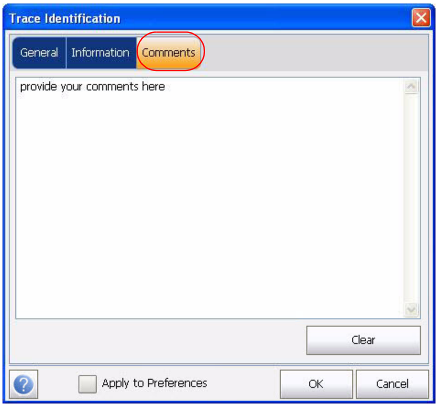
Press **Restore Defaults** to remove all the changes and apply the default values.

To edit comments:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **Comments** tab.



4. Edit comments in the **Comments** window for the current trace.

Note: *If you want to apply the above settings to the preferences tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

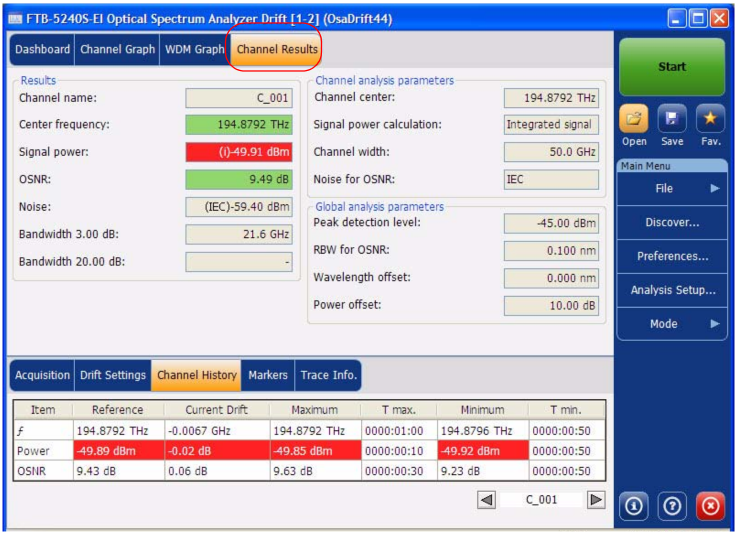
Press **Clear** to clear all the changes made in the **Comments** tab.

Channel Results Tab

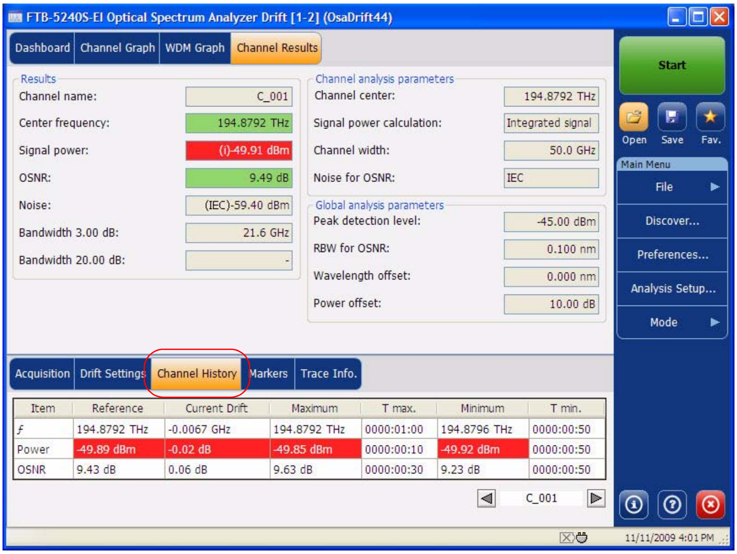
When you select a channel in the **Channel History** tab, the **Channel Results** tab will show complete information about the parameters measured for the selected channel. The pass/fail verdict for thresholds are also displayed in the **Channel Results** tab. If the verdict is fail for any parameter, its value will appear in red. If the verdict is pass, its value will appear in green.

To view channel results:

1. From the main window, select **Channel Results** tab.



2. Select a channel from the **Channel History** tab to view the channel results for the selected channel.



- **Channel name**: indicates the name of the channel as defined in the channel list.
- **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.
- **Signal power**: indicates the signal power for the selected channel (excludes noise).
- **OSNR**: is the Optical Signal to Noise Ratio, given by Signal power (according to the current calculation method, in dBm) minus Noise (according to the current calculation method, in dBm).
- **Noise**: indicates the level of the noise for the selected channel.
- **Bandwidth 3.00 dB**: indicates bandwidth measured by taking the width of a signal at 50 % linear power of the peak or -3 dB from the peak.

- **Bandwidth x dB**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.
- **Channel center**: indicates the channel center for the selected channel.
- **Signal power calculation**: indicates which calculation method to apply for signal power value.

  Integrated signal power: The integrated signal power represents the sum of the power values included between the channel limits of this channel, minus the estimated noise contribution between the same boundaries. In some cases, for instance CATV signals, signals with high-frequency modulation, or signals with an inherent line width similar or larger than the OSA's resolution bandwidth, this calculation becomes a better estimation of the true signal power.

  Peak signal power: The peak signal power represents the maximum power value inside the channel, but it differs a little from the peak measurement on the spectrum due to the fact that the estimated noise is subtracted to get the peak signal power.

  Total channel power: The total channel power represents the sum of the power values included inside the channel width (including noise).

- **Channel width**: indicates the limit within which the power values will be considered in the channel.
- **Noise for OSNR**: indicates which calculation method to use for OSNR value.

  Fixed range IEC based (IEC): The IEC method uses the interpolation of noise measured on both sides of the signal to estimate the noise level. The position at which the noise is estimated from the center wavelength is given by the OSNR distance.

  In-Band (InB): The In-Band method uses a series of scans having different polarization states to calculate the noise level in band.

In-Band narrow filter (InB nf): The In-Band narrow filter method uses the In-Band method with additional processing to provide an accurate OSNR value for the narrow carved noise. This is because with narrow filters, the noise level under the peak is not uniform and the OSNR value depends on the processing width selected.

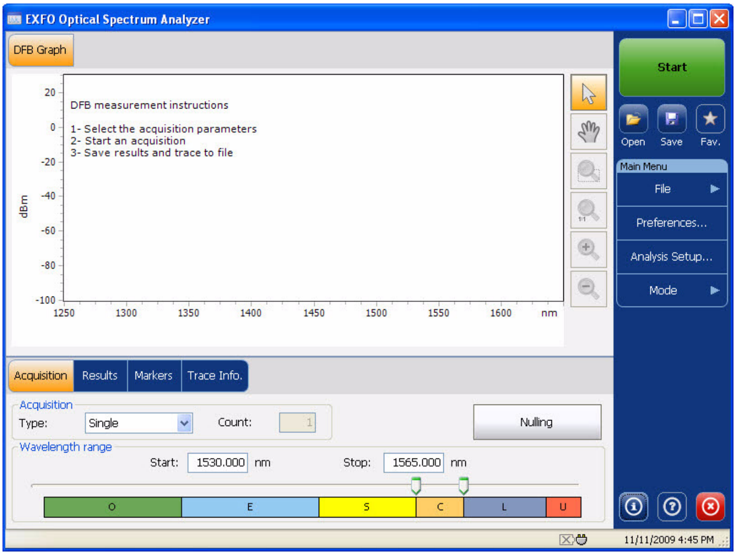
- **Peak detection level**: indicates the minimum power level from where the peak can be considered as a signal.
- **RBW for OSNR**: indicates the resolution bandwidth for the selected OSNR value.
- **Wavelength offset**: indicates the offset value applied on the wavelength.
- **Power offset**: indicates the offset value applied on the power.

Managing DFB Test Results

The application allows you to view and manage your DFB test results. You can view the graph and results for your DFB laser source.

DFB Graph Tab

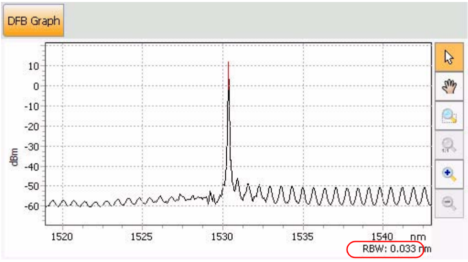
The **DFB Graph** tab allows you to view the spectrum of a DFB laser source. This graph represents the optical power against wavelength. When there is no acquisition, the graph zone shows a procedure explaining how to take measurements with the OSA application.



When the acquisition is taken (see *Starting a Measurement* on page 273 for details on how to perform a test), the trace will be displayed in the DFB **Graph** tab with information along the following axis values:

➤ **X axis**: wavelength in nm or frequency in THz

➤ **Y axis**: optical power expressed in dBm, as measured in the optical resolution bandwidth (RBW) of the OSA. This reference RBW is shown at the bottom right of the graph.



If the current trace was previously saved, the graph will display the file name of the current trace in the title bar.

You can also perform the following on your graph:

➤ Move markers on the graph.

➤ Use zoom functions to zoom on specific areas or peaks of the graph.

Note: *For more information on these, see* Markers Tab *on page 344 and* Using Zoom Controls *on page 396.*

Results Tab

You can view the analysis of the DFB laser source from the **Results** tab.

To view results:

From the main window, select the **Results** tab.

The following information related to the DFB measurement is displayed in the **Results** table:

➤ **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak.

➤ **Peak power (dBm)**: indicates the peak signal power.

➤ **Bandwidth 3.00 dB**: indicates the bandwidth measured by taking the width of a signal at 50 % linear power of the peak or -3 dB from the peak.

➤ **Left SMSR**: indicates the Left side-mode suppression ratio. It is the power difference between the main mode and the most powerful outstanding side-mode on the left.

➤ **Right SMSR**: indicates the Right side-mode suppression ratio. It is the power difference between the main mode and the most powerful outstanding side-mode on the right.

➤ **Worst case SMSR**: indicates the power difference between the main mode and the side-mode with the highest power.

➤ **Worst case SMSR position**: indicates the spectral position of the worst SMSR.

➤ **Left stopband**: indicates the spectral position difference between the main mode and the closest side-mode on the left.

- **Right stopband**: indicates the spectral position difference between the main mode and the closest side-mode on the right.
- **Two-sided stopband**: indicates the spectral position difference between closest left and right side-modes adjacent to the main mode.
- **Central offset**: indicates the spectral position of the main mode minus the mean of the spectral positions of the first adjacent left and right side-modes.
- **Fabry-Perot mode spacing**: indicates the average estimated spectral spacing between adjacent Fabry-Perot modes of the DFB.

Markers Tab

You can use markers to perform manual measurements and verification directly on the trace. There are two vertical markers indicating the wavelength positions, which you can also move directly on the trace using the zoom select option. You can measure actual power and wavelength values of any point on the trace.

Note: *These markers are visible in the* ***DFB Graph*** *tab only when the markers tab is selected in the main window.*

When the markers are set, you will find two markers identified by the letters A and B in the **DFB Graph** tab. These markers are represented both in the graph and the markers tab to customize your markers.

The application allows you to fix the distance between the vertical markers. When this feature is activated, while moving any one marker, both markers shall move at the same rate and distance.

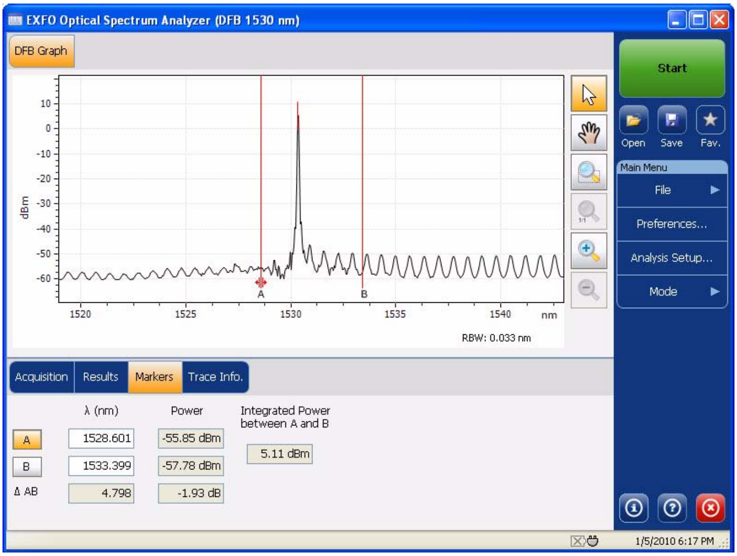
The markers A and B in the **Markers** tab act as toggle buttons to enable selection. When a marker is activated, the colour of the button changes to orange and the selected marker displays a double arrow at the base of the marker in the **DFB Graph** tab, which means that the marker can be moved.

At this point, if you press the other vertical marker in the **DFB Graph** tab, the selection of the toggle switches to this marker. However, if you select the other marker button from the **Markers** tab, both the markers will be selected and the distance between both will be locked.

To define a marker:

1. From the main window, press the **Markers** tab.

2. Set the marker by entering precise values in the **A** and **B** boxes.



Vertical markers A and B appears on the graph and the following values will be displayed in the **Markers** tab.

- power value corresponding to the wavelength position of both the markers.
- wavelength difference between the markers (A-B)
- power difference in dB between the markers
- integrated power between markers in dBm.

You can also move the markers directly on the **DFB Graph** tab. Drag the marker to the desired area in the display. You will notice that the corresponding box in the **Markers** tab changes according to the marker's position. If you want to set a precise value for the marker, simply enter it in the box.

Note: *After using the zoom tools in the **DFB Graph** tab, you can only move the markers again on the graph after deactivating the zoom. Pressing the arrow in the zoom tools section will deactivate the zoom function.*

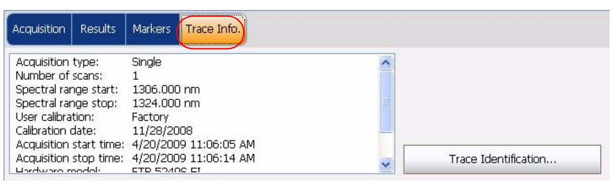
Note: *Markers A and B cannot cross. Displacing a marker over the second one will cause both markers to move together.*

Trace Info. Tab

After acquiring a trace, you might want to view details about the acquisition. The **Trace Info.** tab shows information related to acquisition parameters and conditions. You can also edit information about the tested fiber and job or add comments. This information is saved along with the trace.

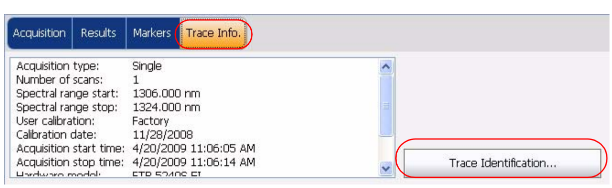
To view trace information parameters:

From the main window, select the **Trace Info.** tab.

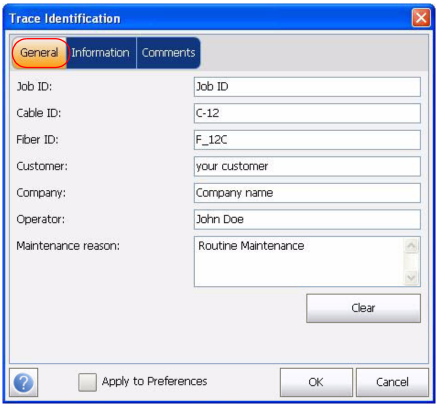


To edit general information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **General** tab.



4. Edit the general information as required.

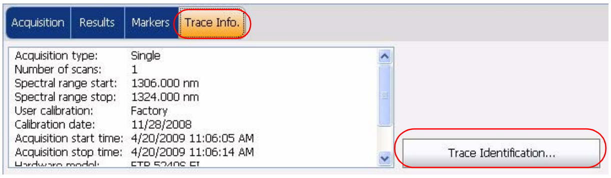
Note: *If you want to apply the above settings to the* ***Preferences*** *tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

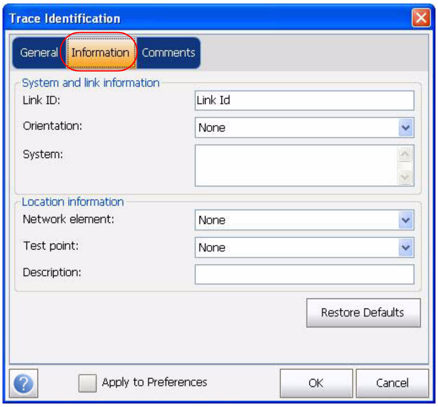
Press **Clear** to clear all the changes made in the **General** tab.

To edit trace information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **Information** tab.



4. Edit the information as required.

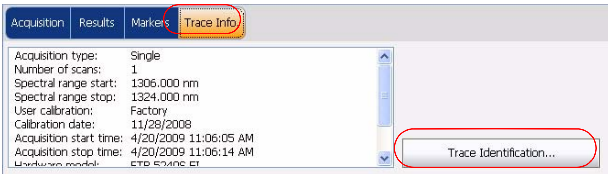
Note: *If you want to apply the above settings to the **Preferences** tab, select the **Apply to Preferences** option and press **OK**.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

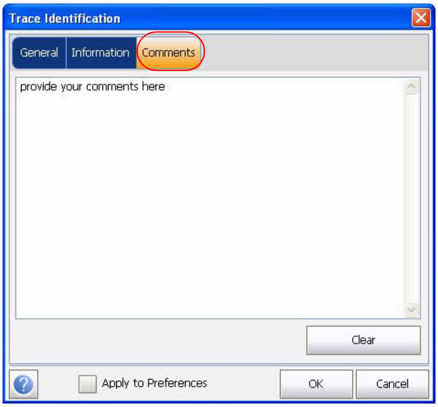
Press **Restore Defaults** to remove all the changes and apply the default values.

To edit comments:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **Comments** tab.



4. Edit comments in the **Comments** window for the current trace.

Note: *If you want to apply the above settings to the preferences tab, select the **Apply to Preferences** option and press **OK**.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

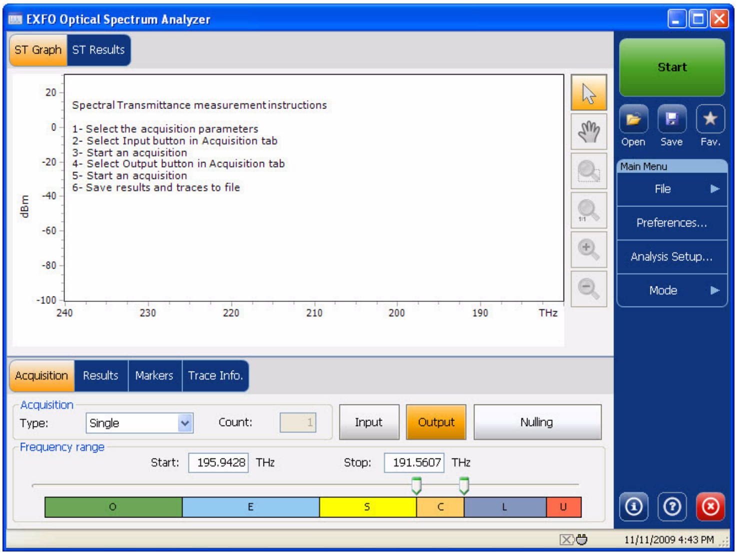
Press **Clear** to clear all the changes made in the **Comments** tab.

Managing Spectral Transmittance Test Results

The application allows you to view and manage your spectral transmittance test results. You can view the graph of your acquisition, results for a single channel, global results and information about the trace.

ST Graph Tab

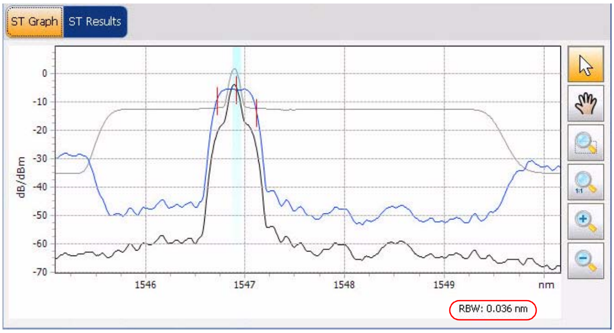
The ST **Graph** tab allows you to view the spectrum of the input trace, the output trace and the calculated ST trace. This graph represents the optical power against wavelength. When there is no acquisition, the graph zone shows a procedure explaining how to take measurements with the OSA application.



When the acquisition is taken (see *Starting a Measurement* on page 273 for details on how to perform a test), the trace will be displayed in the **ST Graph** tab with information along the following axis values:

➤ **X axis**: wavelength in nm or frequency in THz

➤ **Y axis**: optical power expressed in dBm, as measured in the optical resolution bandwidth (RBW) of the OSA. This reference RBW is shown at the bottom right of the graph.



If the current trace was previously saved, the graph will display the file name of the current trace in the title bar.

You can also perform the following on your graph:

➤ Move markers on the graph.

➤ Use zoom functions to zoom on specific areas or peaks of the graph.

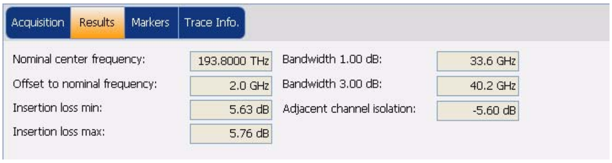
Note: *For more information on these, see* Markers Tab *on page 353 and* Using Zoom Controls *on page 396.*

Results Tab

The results table shows the spectral transmittance results for the active trace. Results for only the channels within the scan range will be displayed.

To view results:

From the main window, select the **Results** tab.



The following results related to the channels are displayed:

➤ **Nominal center wavelength or frequency (nm or Thz)**: indicates a single value that represent the channels center wavelength (in nm) or frequency (in THz).

➤ **Offset to nominal wavelength or frequency (nm or Thz)**: indicates the offset value applied to the nominal wavelength (in nm) or frequency (in THz).

➤ **Insertion loss min (dB):** indicates the minimum difference between a reference power level and the measured power level (in dB).

➤ **Insertion loss max (dB):** indicates the maximum difference between a reference power level and the measured power level (in dB).

➤ **Bandwidth x at (dB)**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.

➤ **Bandwidth y at (dB)**: indicates bandwidth measured by taking the width of a signal at y dB from the peak.

➤ **Adjacent channel isolation (dB)**: is the isolation (in dB) taken at the channel distance on the left or right of the nominal wavelength. The worst value between the left and right isolation is kept.

Markers Tab

You can use markers to perform manual measurements and verification directly on the trace. There are two vertical and horizontal markers indicating the wavelength and power positions respectively, which you can also move directly on the trace using the zoom select option. You can measure actual power and wavelength values of any point on the trace.

Note: *These markers are visible in the* ***ST Graph*** *tab only when the markers tab is selected in the main window.*

Note: *Horizontal markers will be displayed only if the markers are activated in the* ***Preferences*** *tab.*

When the vertical markers are set, you will find two markers identified by the letters A and B in the **ST Graph** tab. Horizontal markers are identified by letters C and D. These markers are represented both in the graph and the markers tab to customize your markers.

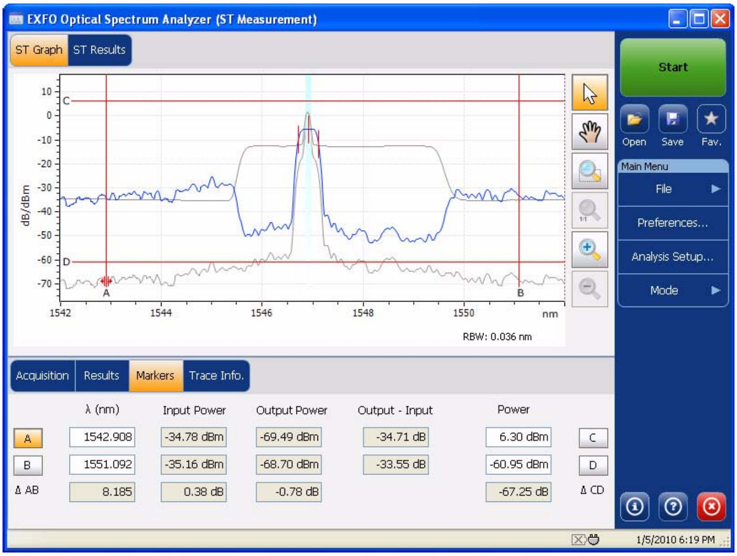
The application allows you to fix the distance between the vertical markers. When this feature is activated, while moving any one marker, both markers shall move at the same rate and distance.

The markers A and B in the **Markers** tab act as toggle buttons to enable selection. When a marker is activated, the colour of the button changes to orange and the selected marker displays a double arrow at the base of the marker in the **ST Graph** tab, which means that the marker can be moved.

At this point, if you press the other vertical marker in the **ST Graph** tab, the selection of the toggle switches to this marker. However, if you select the other marker button from the **Markers** tab, both the markers will be selected and the distance between both will be locked. When both the vertical markers are locked for fixed movement, pressing any horizontal marker from the **Markers** tab or the **ST Graph** tab will remove the selection of both the vertical markers.

To define a marker:

1. From the main window, press the **Markers** tab.
2. Set the marker by entering precise values in the **A**, **B**, **C**, and **D** boxes.



Vertical markers A and B appears on the graph and the following values will be displayed in the **Markers** tab.

- **Input** and **Output** power values corresponding to the wavelength position of both the markers.
- wavelength difference between the markers (A-B)
- power difference in dB between the markers for **Input** and **Output** power values
- power difference between **Output** and **Input** power for both the markers in dB.

Horizontal markers C and D appears on the graph and the power difference between the markers (C-D) related to the horizontal markers will be displayed in the **Markers** tab.

You can also move the markers directly on the **ST Graph** tab. Drag the marker to the desired area in the display. You will notice that the corresponding box in the **Markers** tab changes according to the marker's position. If you want to set a precise value for the marker, simply enter it in the box.

Note: *After using the zoom tools in the **ST Graph** tab, you can only move the markers again on the graph after deactivating the zoom. Pressing the arrow in the zoom tools section will deactivate the zoom function.*

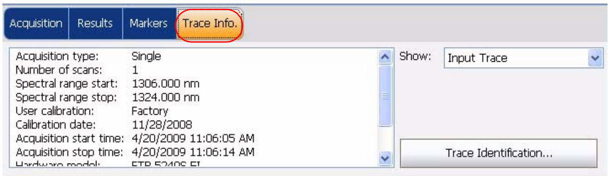
Note: *Markers A and B cannot cross. Displacing a marker over the second one will cause both markers to move together.*

Trace Info. Tab

After acquiring a trace, you might want to view details about the acquisition. The **Trace Info.** tab shows information related to acquisition parameters and conditions. You can also edit information about the tested fiber and job or add comments. This information is saved along with the trace.

To view trace information parameters:

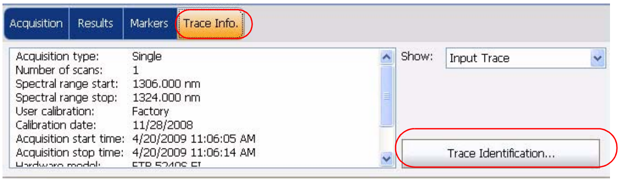
From the main window, select the **Trace Info.** tab.



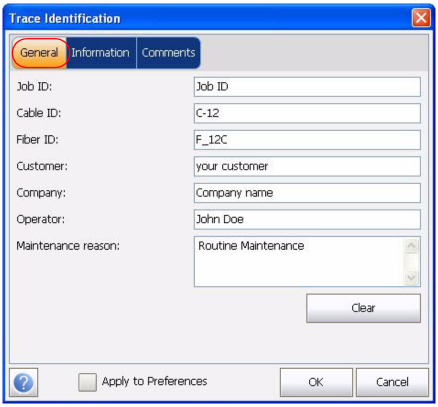
You can select **Input Trace**, **Output Trace** or **ST Trace** based on the information you want to view from the **Show** drop down list.

To edit general information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **General** tab.



4. Edit the general information as required.

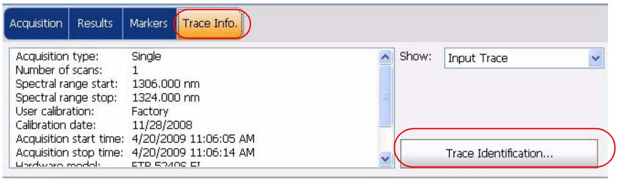
Note: *If you want to apply the above settings to the* ***Preferences*** *tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

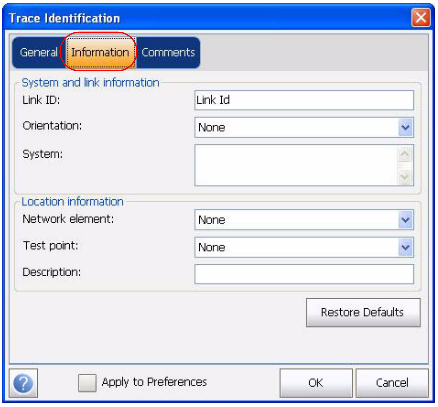
Press **Clear** to clear all the changes made in the **General** tab.

To edit trace information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **Information** tab.



4. Edit the information as required.

Note: *If you want to apply the above settings to the* ***Preferences*** *tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

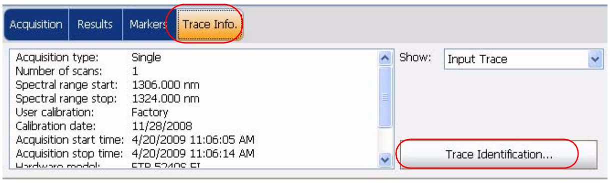
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

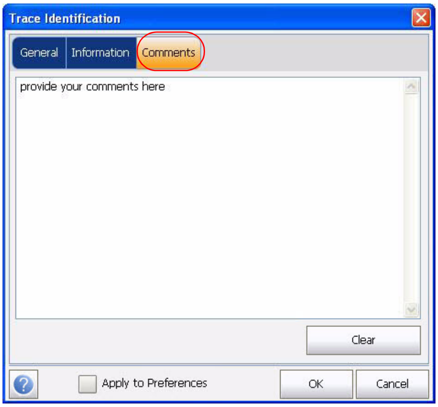
To edit comments:

1. From the main window, select the **Trace Info.** tab.

2. Press **Trace Identification**.



3. Select the **Comments** tab.



4. Edit comments in the **Comments** window for the current trace.

Note: *If you want to apply the above settings to the preferences tab, select the **Apply to Preferences** option and press **OK**.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Clear** to clear all the changes made in the **Comments** tab.

ST Results Tab

The **ST Results** tab will show complete information about the spectral transmittance parameters and the global analysis parameters.

To view ST results:

1. From the main window, select **ST Results** tab.

The following information is displayed in the **ST Results** pane:

- **Nominal center wavelength or frequency (nm or Thz)**: indicates a single value that represent the channels center wavelength (in nm) or frequency (in THz).
- **Offset to nominal wavelength or frequency (nm or Thz)**: indicates the offset value applied to the nominal wavelength (in nm) or frequency (in THz).
- **Insertion loss min (dB):** indicates the minimum difference between a reference power level and the measured power level (in dB).

- **Insertion loss max (dB):** indicates the maximum difference between a reference power level and the measured power level (in dB).
- **Bandwidth 1 at (dB)**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.
- **Bandwidth 2 at (dB)**: indicates bandwidth measured by taking the width of a signal at y dB from the peak.
- **Adjacent channel isolation (dB)**: indicates the isolation (in dB) taken at the channel distance on the left or right of the nominal wavelength. The worst value between the left and right isolation is kept.

The following information is displayed in the **Analysis parameters** pane:

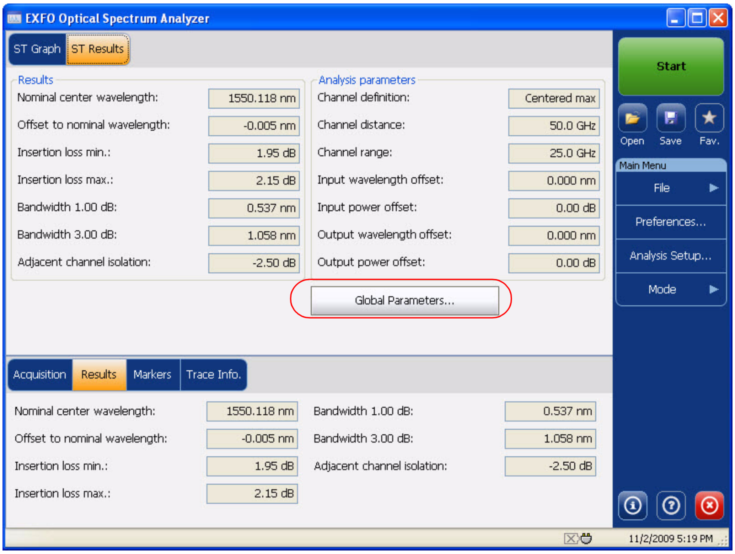
- **Channel definition (GHz)**: indicates the limit inside which the power values will be considered in the channel.
- **Channel distance (GHz or nm)**: indicates distance between the channels. The value of channel distance will be set depending on the selection made for the channel definition option. The channel distance box will be enabled only when the channel center wavelength option is set to custom.
- **Channel range (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel. Integrated power is calculated on channel width.

- **Input wavelength offset (nm)**: indicates the offset value applied on the input wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.
- **Input power offset (dB)**: indicates the offset value applied on the input power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the ↕ arrow.
- **Output wavelength offset (nm)**: indicates the offset value applied on the output wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.
- **Output power offset (dB)**: indicates the offset value applied on the output power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the ↕ arrow.

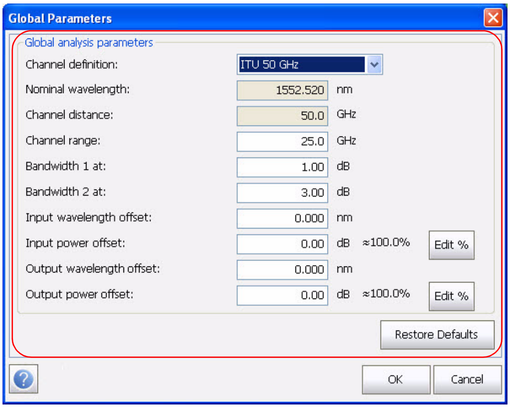
To edit ST analysis parameters:

1. From the main window, select the **ST Results** tab.

2. Press **Global Parameters**.



3. Edit the values as explained below:



➤ **Channel definition (GHz)**: indicates the limit inside which the power values will be considered in the channel.

Centred on max peak: Channel is centered on the lowest insertion loss peak.

ITU Grid: Select the nearest ITU channel from the peak with lowest insertion loss.

CWDM: Select the nearest CWDM channel from the peak with lowest insertion loss.

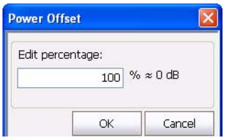
Custom: Channel is centered on value specified by the user.

➤ **Nominal wavelength or frequency (nm or Thz)**: indicates a single value that represent the channels center wavelength (in nm) or frequency (in THz). This field is editable only when **Channel definition** is selected as **Custom**.

➤ **Channel distance (GHz or nm)**: indicates distance between the channels. The value of channel distance will be set depending on the selection made for the channel definition option. The channel distance box will be enabled only when the channel center wavelength option is set to custom.

- **Channel range (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel. Integrated power is calculated on channel width.
- **Bandwidth 1 at (dB)**: indicates bandwidth measured by taking the width of a signal at x dB from the peak.
- **Bandwidth 2 at (dB)**: indicates bandwidth measured by taking the width of a signal at y dB from the peak.
- **Input wavelength offset (nm)**: indicates the offset value applied on the input wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the arrow.
- **Input power offset (dB)**: indicates the offset value applied on the input power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the arrow.

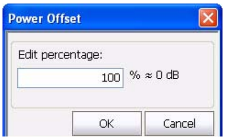
  To edit the power offset value in percentage, press **Edit %** button.

  The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

- **Output wavelength offset (nm)**: indicates the offset value applied on the output wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the arrow.
- **Output power offset (dB)**: indicates the offset value applied on the output power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the arrow.

  To edit the power offset value in percentage, press **Edit %** button.

  The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

4. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

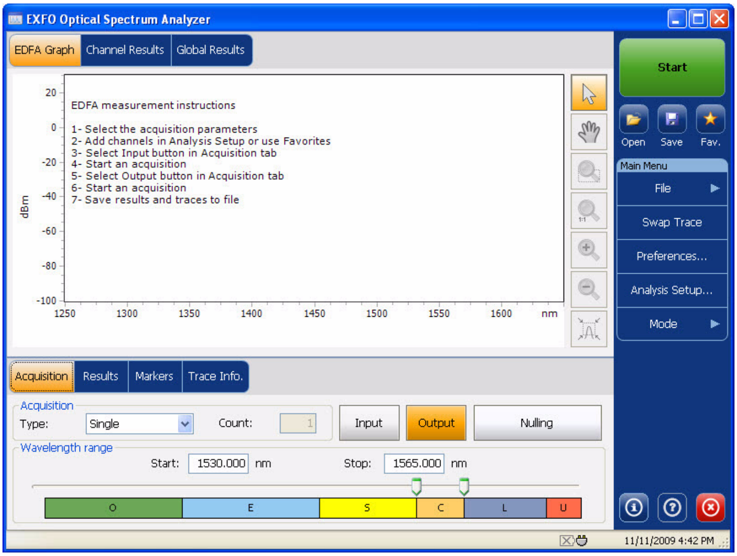
Press **Restore Defaults** to remove all the changes and apply the default values.

Managing EDFA Test Results

The application allows you to view and manage your EDFA test results. You can view the graph of your acquisition, results for a single channel, global results and information about the trace.

EDFA Graph Tab

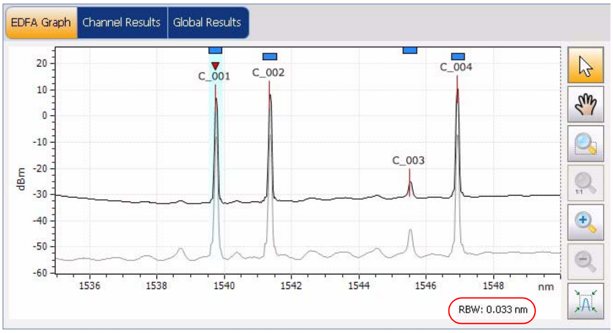
The **EDFA Graph** tab allows you to view the spectrum of the active trace. This graph represents the optical power against wavelength. When there is no acquisition, the graph zone shows a procedure explaining how to take measurements with the OSA application.



When the acquisition is taken (see *Starting a Measurement* on page 273 for details on how to perform a test), the trace will be displayed in the **EDFA Graph** tab with information along the following axis values:

➤ **X axis**: wavelength in nm or frequency in THz

➤ **Y axis**: optical power expressed in dBm, as measured in the optical resolution bandwidth (RBW) of the OSA. This reference RBW is shown at the bottom right of the graph.



If the current trace was previously saved, the graph will display the file name of the current trace in the title bar.

The graph will display peak indicators for all the channels found by the application. A blue horizontal bar ( ) will be displayed on the top of the channels, with a red horizontal line over the peaks to indicate the peak position.

You can also perform the following on your graph:

➤ Move markers on the graph.

➤ Use zoom functions to zoom on specific areas or peaks of the graph.

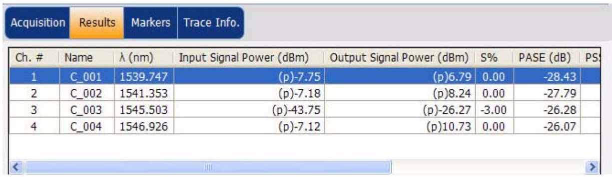
Note: *For more information on these, see* Markers Tab *on page 372 and* Using Zoom Controls *on page 396.*

Results Tab

The results table shows channel results for the active trace. The results for only the channels within the scan range will be displayed.

To view results:

From the main window, select the **Results** tab.



Ch. #	Name	λ (nm)	Input Signal Power (dBm)	Output Signal Power (dBm)	S%	PASE (dB)	PS
1	C_001	1539.747	(p)-7.75	(p)6.79	0.00	-28.43	
2	C_002	1541.353	(p)-7.18	(p)8.24	0.00	-27.79	
3	C_003	1545.503	(p)-43.75	(p)-26.27	-3.00	-26.28	
4	C_004	1546.926	(p)-7.12	(p)10.73	0.00	-26.07	

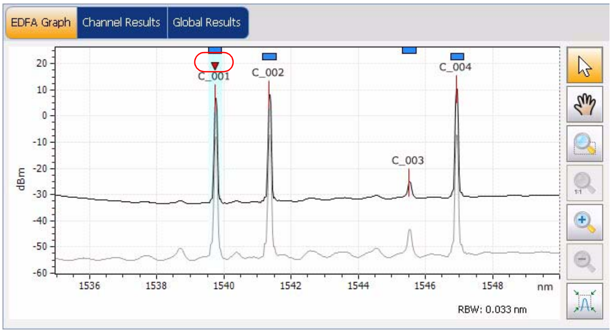
The following information related to the channels is displayed in the **Results** table:

- **Ch#**: indicates the channel number. Channel 1 corresponds to the first channel found.
- **Name**: indicates the channel name as defined in the channel list.
- **Center wavelength/frequency**: indicates the spectral center-of-mass for the peak in that channel.
- **Input Signal Power (dBm)**: indicates the input signal power for the selected channel (excludes noise).
- **Output Signal Power (dBm)**: indicates the output signal power for the selected channel (excludes noise).
- **S %**: indicates the current output power according to the measured output power (Output Signal Power / [Output Signal Power + PASE]).
- **PASE**: indicates the power of the spontaneous emission amplified by EDFA.
- **PSSE**: indicates the power of the spontaneous emission of the source.
- **Gain**: indicates the gain (Output Signal Power - Input Signal Power) for the selected channel.

➤ **Noise Figure**: indicates the EDFA's noise figure measured for the selected channel.

➤ **Gain - Avg. Gain**: indicates the selected channel gain minus the average of all channel gains.

A small red marker (▼) will point down at the peak in the **EDFA Graph** tab when you select a row in the **Results** tab. The red marker will move accordingly to indicate the corresponding peak on the graph, with a focus on the selected channel.



Markers Tab

You can use markers to perform manual measurements and verification directly on the trace. There are two vertical and horizontal markers indicating the wavelength and power positions respectively, which you can also move directly on the trace using the zoom select option. You can measure actual power and wavelength values of any point on the trace.

Note: *These markers are visible in the* ***EDFA Graph*** *tab only when the markers tab is selected in the main window.*

Note: *Horizontal markers will be displayed only if the markers are activated in the* ***Preferences*** *tab.*

When the vertical markers are set, you will find two markers identified by the letters A and B in the **EDFA Graph** tab. Horizontal markers are identified by letters C and D. These markers are represented both in the graph and the markers tab to customize your markers.

The application allows you to fix the distance between the vertical markers. When this feature is activated, while moving any one marker, both markers shall move at the same rate and distance.

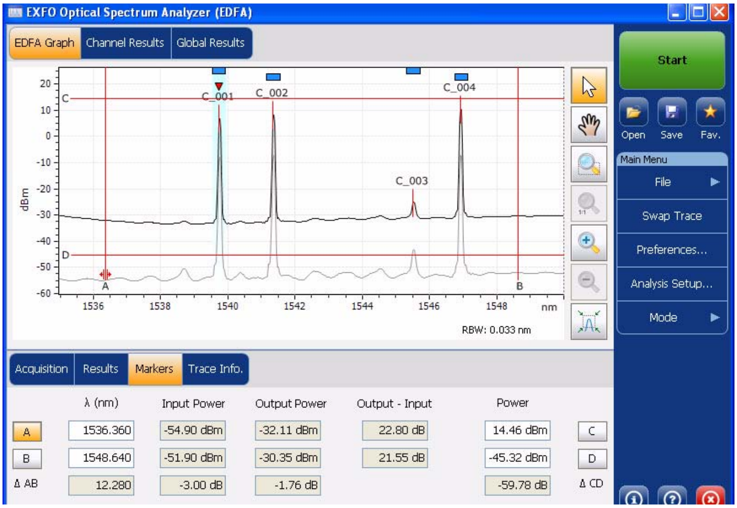
The markers A and B in the **Markers** tab act as toggle buttons to enable selection. When a marker is activated, the colour of the button changes to orange and the selected marker displays a double arrow at the base of the marker in the **EDFA Graph** tab, which means that the marker can be moved.

At this point, if you press the other vertical marker in the **EDFA Graph** tab, the selection of the toggle switches to this marker. However, if you select the other marker button from the **Markers** tab, both the markers will be selected and the distance between both will be locked. When both the vertical markers are locked for fixed movement, pressing any horizontal marker from the **Markers** tab or the **EDFA Graph** tab will remove the selection of both the vertical markers.

To define a marker:

1. From the main window, press the **Markers** tab.

2. Set the marker by entering precise values in the **A**, **B**, **C**, and **D** boxes.



Vertical markers A and B appears on the graph and the following values will be displayed in the **Markers** tab.

- **Input** and **Output** power values corresponding to the wavelength position of both the markers.
- wavelength difference between the markers (A-B)
- power difference in dB between the markers for **Input** and **Output** power values
- power difference between **Output** and **Input** power for both the markers in dB.

Horizontal markers C and D appears on the graph and the power difference between the markers (C-D) related to the horizontal markers will be displayed in the **Markers** tab.

You can also move the markers directly on the **EDFA Graph** tab. Drag the marker to the desired area in the display. You will notice that the corresponding box in the **Markers** tab changes according to the marker's position. If you want to set a precise value for the marker, simply enter it in the box.

Note: *After using the zoom tools in the **EDFA Graph** tab, you can only move the markers again on the graph after deactivating the zoom. Pressing the arrow in the zoom tools section will deactivate the zoom function.*

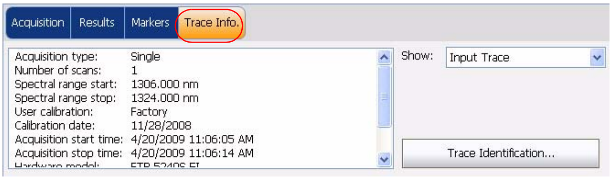
Note: *Markers A and B cannot cross. Displacing a marker over the second one will cause both markers to move together.*

Trace Info. Tab

After acquiring a trace, you might want to view details about the acquisition. The **Trace Info.** tab shows information related to acquisition parameters and conditions. You can also edit information about the tested fiber and job or add comments. This information is saved along with the trace.

To view trace information parameters:

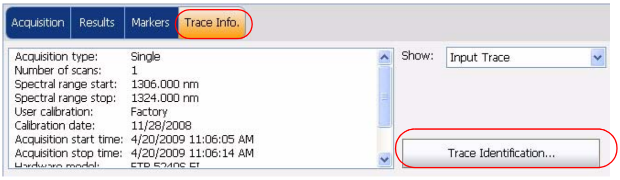
From the main window, select the **Trace Info.** tab.



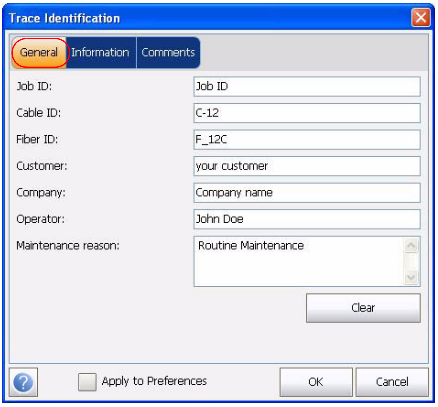
You can select **Input Trace or Output Trace** based on the information you want to view from the **Show** drop down list.

To edit general information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **General** tab.



4. Edit the general information as required.

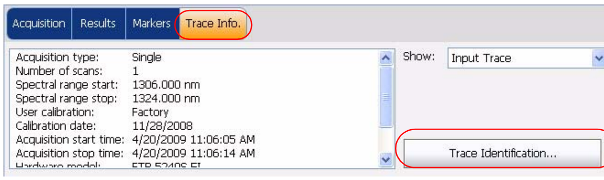
Note: *If you want to apply the above settings to the* ***Preferences*** *tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

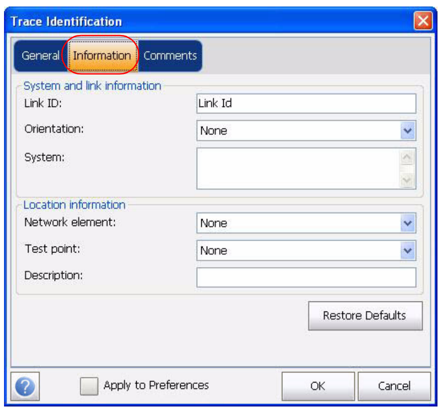
Press **Clear** to clear all the changes made in the **General** tab.

To edit trace information:

1. From the main window, select the **Trace Info.** tab.
2. Press **Trace Identification**.



3. Select the **Information** tab.



4. Edit the information as required.

Note: *If you want to apply the above settings to the* ***Preferences*** *tab, select the* ***Apply to Preferences*** *option and press* ***OK****.*

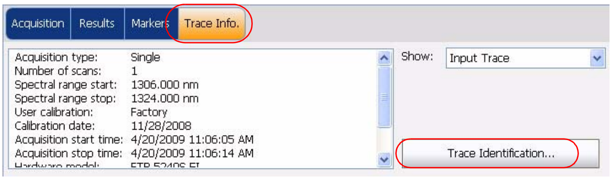
5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

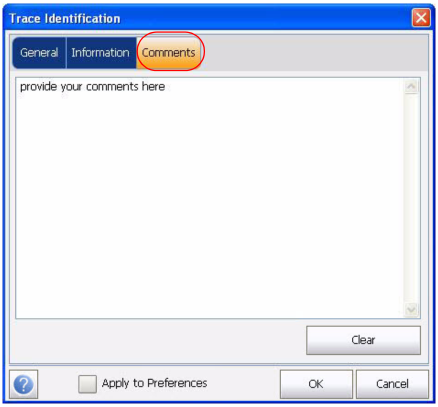
To edit comments:

1. From the main window, select the **Trace Info.** tab.

2. Press **Trace Identification**.



3. Select the **Comments** tab.



4. Edit comments in the **Comments** window for the current trace.

Note: *If you want to apply the above settings to the preferences tab, select the **Apply to Preferences** option and press **OK**.*

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

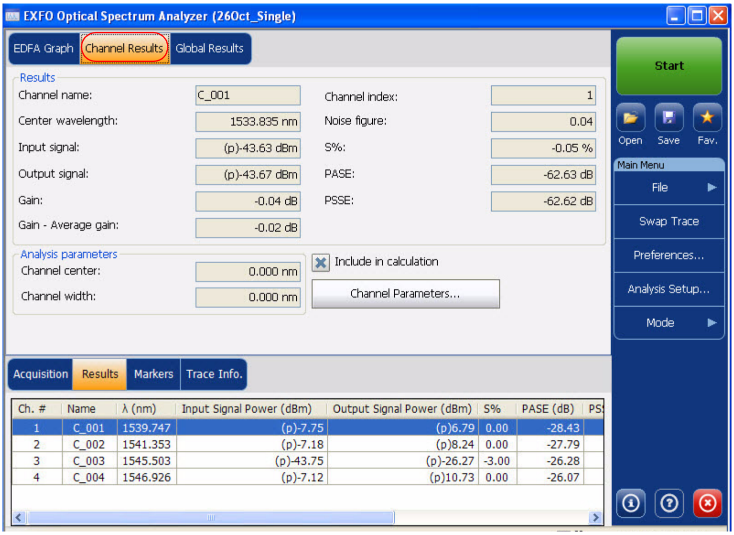
Press **Clear** to clear all the changes made in the **Comments** tab.

Channel Results Tab

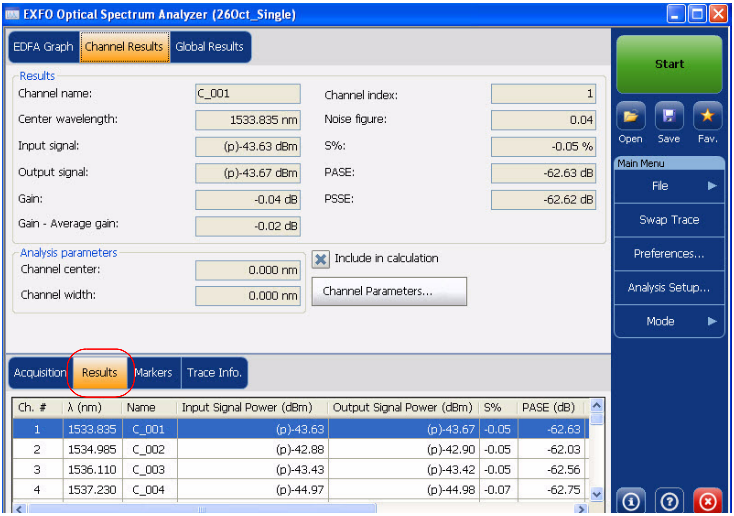
When you select a row from the **Results** tab, the **Channel Results** tab will show complete information about the parameters measured for the selected channel.

To view channel results:

1. From the main window, select **Channel Results** tab.



2. Select a row from the **Results** tab to view the channel results for the selected channel.



3. Under **Results** pane, you can view the following results:

   ➤ **Channel name**: indicates the channel name as defined in the channel list.

   ➤ **Channel Center (nm or THz)**: indicates the central wavelength/frequency for the channel.

   ➤ **Input signal power (dBm)**: indicates the input signal power for the selected channel (excludes noise).

   ➤ **Output signal power (dBm)**: indicates the output signal power for the selected channel (excludes noise).

   ➤ **Gain**: indicates the gain (Output Signal Power - Input Signal Power) for the selected channel.

   ➤ **Gain - Avg. Gain**: indicates the selected channel gain minus the average of all channel gains.

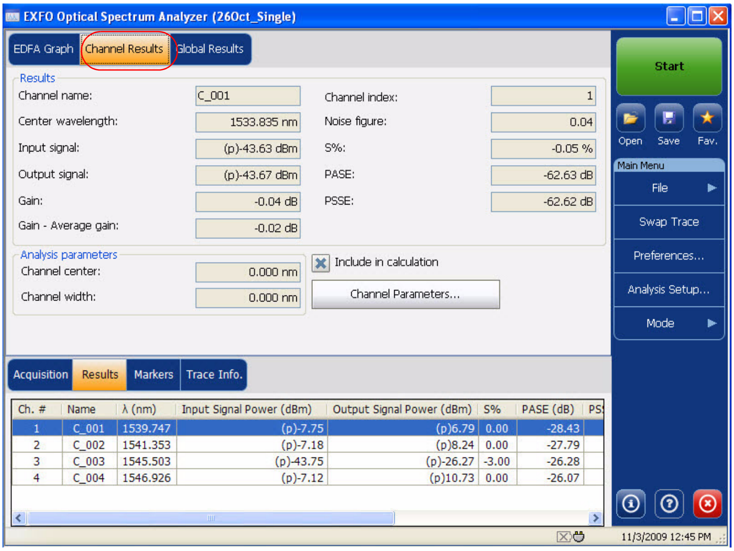
- **Channel index**: indicates the channel number. Channel 1 corresponds to the first channel found.
- **Noise figure**: indicates the EDFA's noise figure measured for the selected channel.
- **S %**: indicates the current output power according to the measured output power (Output Signal Power / [Output Signal Power + PASE]).
- **PASE**: indicates the power of the spontaneous emission amplified by EDFA.
- **PSSE**: indicates the power of the spontaneous emission of the source.

4. Under **Analysis parameters** pane, you can view the following results:
   - **Channel center**: indicates the channel center for the selected channel.
   - **Channel width**: indicates the limit within which the power values will be considered in the channel.

You can include selected channels results in the global results calculation.

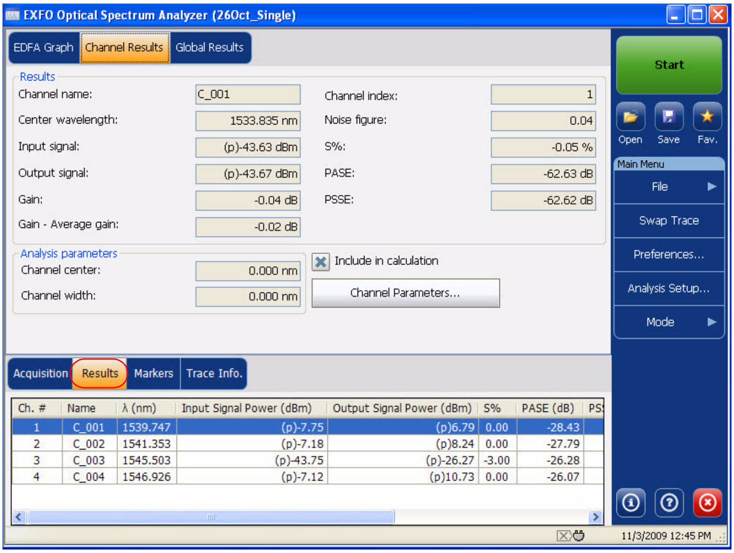
To include channels in global measurement results:

1. From the main window, select **Channel Results** tab.



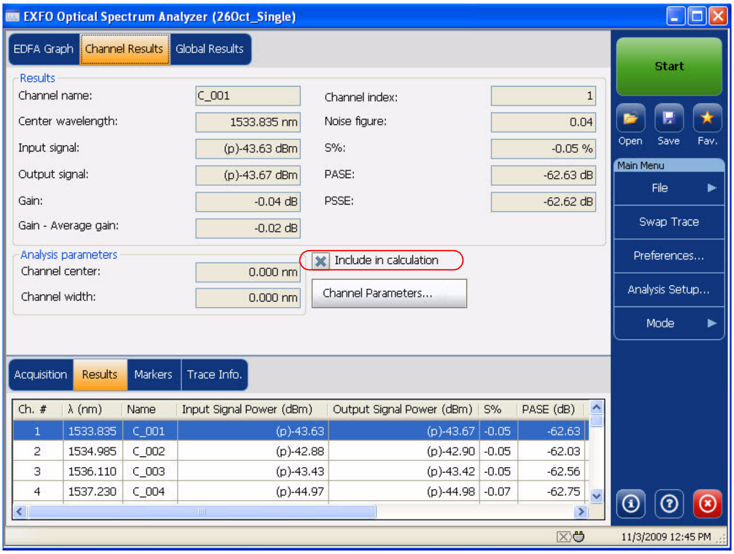
Ch. #	Name	λ (nm)	Input Signal Power (dBm)	Output Signal Power (dBm)	S%	PASE (dB)	PS
1	C_001	1539.747	(p)-7.75	(p)6.79	0.00	-28.43	
2	C_002	1541.353	(p)-7.18	(p)8.24	0.00	-27.79	
3	C_003	1545.503	(p)-43.75	(p)-26.27	-3.00	-26.28	
4	C_004	1546.926	(p)-7.12	(p)10.73	0.00	-26.07	

2. Select a row from the **Results** tab to view the channel results for the selected channel.



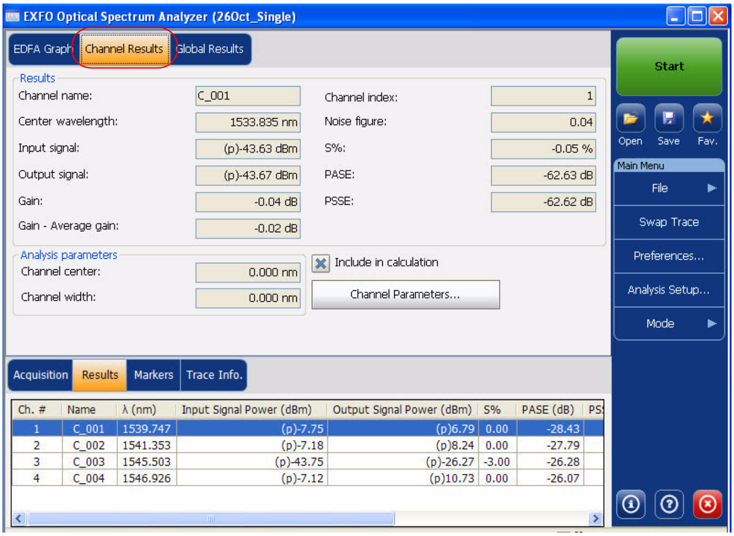
EXFO Optical Spectrum Analyzer (26Oct_Single)
EDFA Graph
Channel Results
Global Results
Results
Channel name: C_001
Center wavelength: 1533.835 nm
Input signal: (p)-43.63 dBm
Output signal: (p)-43.67 dBm
Gain: -0.04 dB
Gain - Average gain: -0.02 dB
Channel index: 1
Noise figure: 0.04
S%: -0.05 %
PASE: -62.63 dB
PSSE: -62.62 dB
Analysis parameters
Channel center: 0.000 nm
Channel width: 0.000 nm
Include in calculation
Channel Parameters...
Start
Open
Save
Fav.
Main Menu
File
Swap Trace
Preferences...
Analysis Setup...
Mode
Acquisition
Results
Markers
Trace Info.
Ch. # | Name | λ (nm) | Input Signal Power (dBm) | Output Signal Power (dBm) | S% | PASE (dB)
1 | C_001 | 1539.747 | (p)-7.75 | (p)6.79 | 0.00 | -28.43
2 | C_002 | 1541.353 | (p)-7.18 | (p)8.24 | 0.00 | -27.79
3 | C_003 | 1545.503 | (p)-43.75 | (p)-26.27 | -3.00 | -26.28
4 | C_004 | 1546.926 | (p)-7.12 | (p)10.73 | 0.00 | -26.07
11/3/2009 12:45 PM

3. Select the **Include in calculation** option.

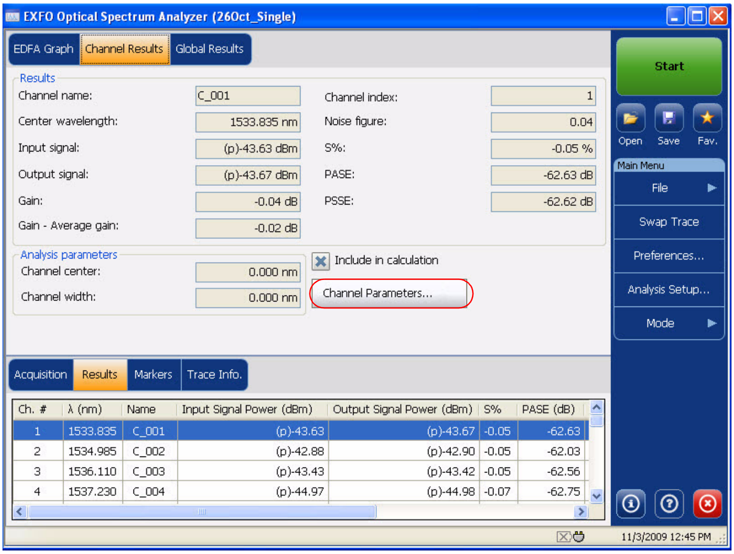


To modify channel results parameters:

1. From the main window, select the **Channel Results** tab.

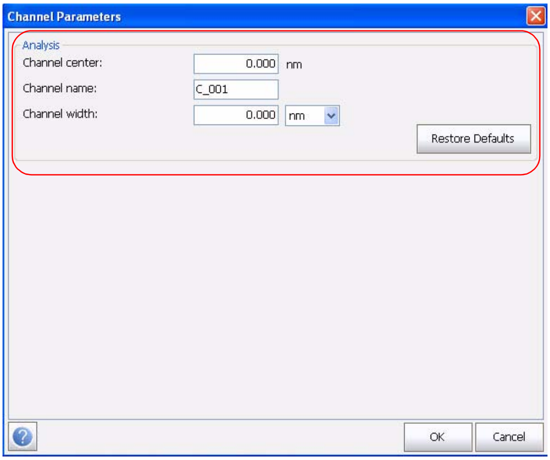


2. Press **Channel Parameters**.



EXFO Optical Spectrum Analyzer (26Oct_Single)
EDFA Graph
Channel Results
Global Results
Results
Channel name:
C_001
Channel index:
1
Center wavelength:
1533.835 nm
Noise figure:
0.04
Input signal:
(p)-43.63 dBm
S%:
-0.05 %
Output signal:
(p)-43.67 dBm
PASE:
-62.63 dB
Gain:
-0.04 dB
PSSE:
-62.62 dB
Gain - Average gain:
-0.02 dB
Analysis parameters
Channel center:
0.000 nm
Include in calculation
Channel width:
0.000 nm
Channel Parameters...
Acquisition
Results
Markers
Trace Info.
Ch. #
λ (nm)
Name
Input Signal Power (dBm)
Output Signal Power (dBm)
S%
PASE (dB)
1 1533.835 C_001 (p)-43.63 (p)-43.67 -0.05 -62.63
2 1534.985 C_002 (p)-42.88 (p)-42.90 -0.05 -62.03
3 1536.110 C_003 (p)-43.43 (p)-43.42 -0.05 -62.56
4 1537.230 C_004 (p)-44.97 (p)-44.98 -0.07 -62.75
Start
Open
Save
Fav.
Main Menu
File
Swap Trace
Preferences...
Analysis Setup...
Mode
11/3/2009 12:45 PM

3. Edit the values as explained below:



- **Channel Center (nm or THz)**: indicates the central wavelength/frequency for the channel.
- **Channel name**: indicates the name of the channel.
- **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.

4. Press **OK** to save the changes for the active trace and close the window, or press **Cancel** to exit without saving.

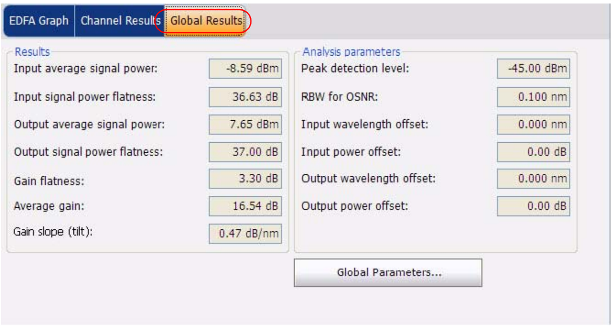
Press **Restore Defaults** to remove all the changes and apply the default values.

Global Results Tab

The application allows you to view the global results of the current measurement. You can also edit global parameters to reanalyze the active trace.

To view global results:

From the main window, select the **Global Results** tab.



Results for the following parameters for all the channels will be displayed:

- **Input average signal power**: indicates the sum of the signal powers of all the peaks detected in the current acquisition, divided by the total number of peaks.
- **Input signal power flatness**: indicates the difference between the maximum and minimum signal power values of the detected peaks, in dB.
- **Output average signal power**: indicates the sum of the signal powers of all the peaks detected in the current acquisition, divided by the total number of peaks.
- **Output signal power flatness**: indicates the difference between the maximum and minimum signal power values of the detected peaks, in dB.
- **Gain flatness**: indicates the difference between the maximum and minimum gain values of the detected channels, in dB.

- **Average gain**: indicates the sum of the gain of all detected channels in the current measurement, divided by the total number of channels.
- **Gain slope (tilt)**: indicates the slope of the linear fit on the gain values of the detected channels.

Note: *To edit global parameters, see* Changing Active Trace Analysis Parameters *on page 390.*

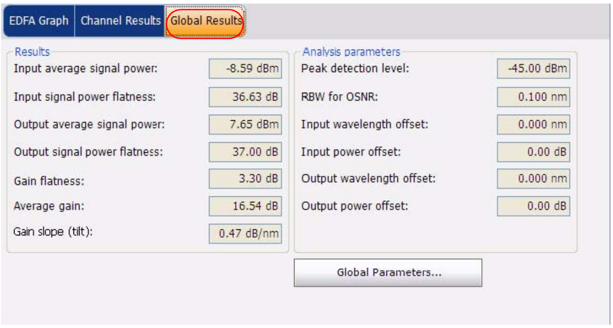
Changing Active Trace Analysis Parameters

The global parameters modifies the active trace. These changes pertain to the analysis and the channel parameters.

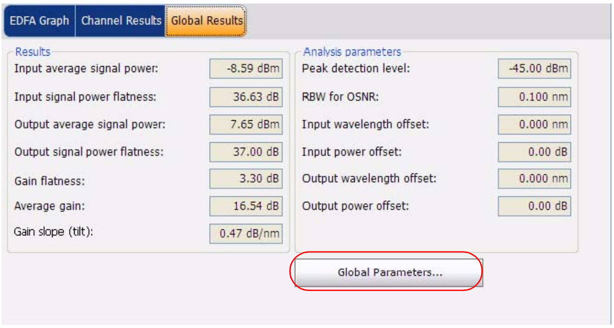
Note: *The modifications made to the global analysis parameters will be applied to the active trace only.*

To change global analysis parameters:

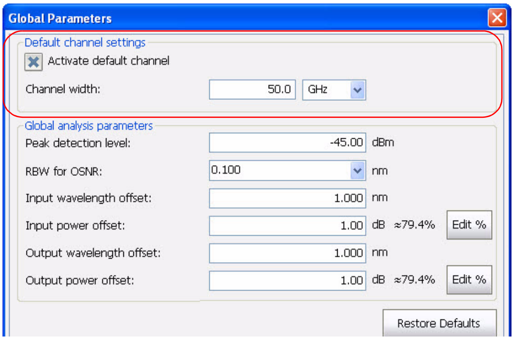
1. From the main window, select the **Global Results** tab.



2. Press **Global Parameters**.



3. Under **Default channel settings**, modify the following parameters as needed:

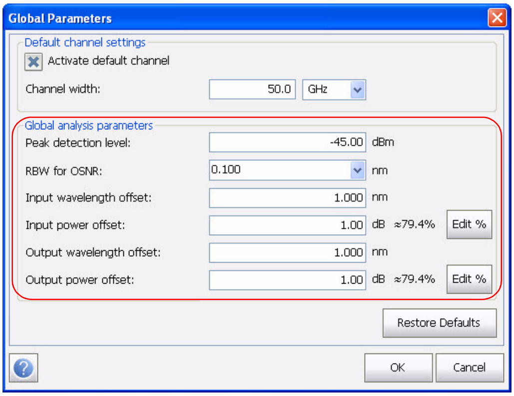


➤ Clear the **Activate default channel** selection, to use the currently defined channel for analysis. This reduces the analysis time by eliminating the peak detection over the complete spectral range. The peaks outside the defined channel will not be detected or analyzed.

➤ **Channel width (GHz or nm)**: indicates the limit inside which the power values will be considered in the channel.

For default channels, the channel width that sets the limits of the channel, should be the same as the channel distance or smaller (channel distance is defined while creating a channel list). If the channel width is not compatible with the channel spacing, either a single peak may be found for two distinct channels and two analysis would be performed and displayed for that peak, or, it is possible that two peaks may be found within the same channel and be considered as one multi-peak signal. With this result, you can use markers to find the spacing between adjacent channels or to find the channel width.

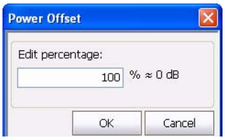
4. Under **Global analysis parameters**, modify the following parameters as needed:



- **Peak detection level (dBm)**: indicates the minimum power level from where the peak can be considered as a signal.
- **RBW for OSNR (nm)**: indicates the resolution bandwidth for the selected OSNR value. The resolution bandwidth of an OSA determines its ability to deal with close optical channel spacing. It is measured as the width of the response curve at half peak power (i.e., 3 dB down) of the instrument to a monochromatic test signal. The instrument's RBW value is written below the graph on the right (below wavelength offset).
- **Input wavelength offset (nm)**: indicates the offset value applied on the input wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the ↔ arrow.

➤ **Input power offset (dB)**: indicates the offset value applied on the input power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the arrow.

To edit the power offset value in percentage, press **Edit %** button.

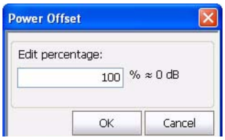


The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

➤ **Output wavelength offset (nm)**: indicates the offset value applied on the output wavelength. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. Entering a value in THz is not possible. The wavelength offset appears below the graph, on the right, next to the arrow.

➤ **Output power offset (dB)**: indicates the offset value applied on the output power. You can use an offset to adjust your unit. This does not replace a calibration performed at EXFO, but it can help you achieve the specifications if you have determined that, for example, your modules are used beyond the normal allowed. The power offset appears in the graph, on the left, above the arrow.

To edit the power offset value in percentage, press **Edit %** button.



The percentage value entered in **Edit percentage** will be converted to a corresponding equivalent value in dB.

5. Press **OK** to save the changes and close the window, or press **Cancel** to exit without saving.

Press **Restore Defaults** to remove all the changes and apply the default values.

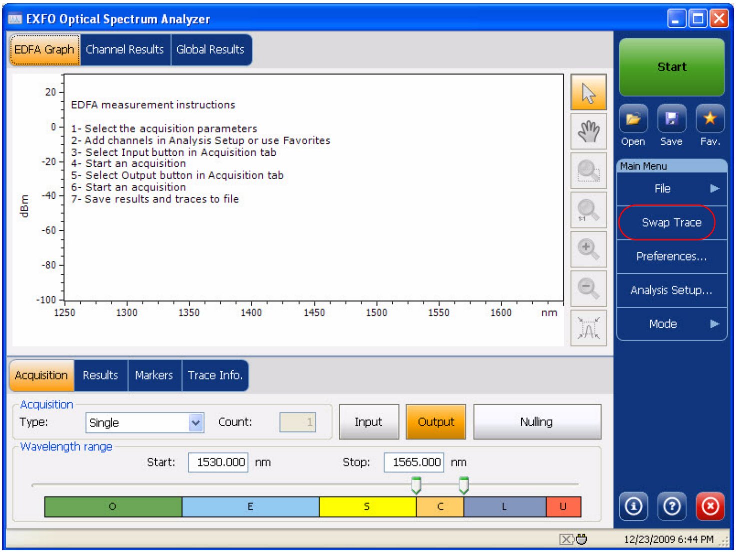
Swapping EDFA Traces

The Swap Trace feature allows you to swap EDFA input and output traces. With this feature, the input trace is replaced with the output trace and vice versa. All results are recalculated.

Note: *Swap Trace feature will not be available if there is no trace in the application.*

To swap EDFA traces:

1. From the **Main Menu**, press **Swap trace**.



All the parameters in the application will be updated according to the modified traces.

Using Zoom Controls

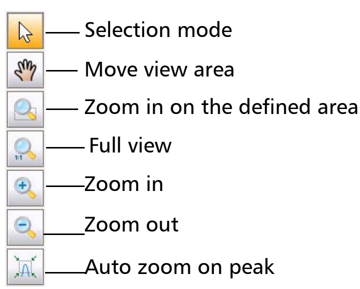
Use the zoom controls to change the scale of the trace display.

You can zoom in on or out of the graph using the corresponding buttons or let the application automatically adjust the zoom on the currently selected peak from the results table.

You can quickly zoom in on or out of a selected peak.

You can also return to the original graph value.

The application provides an automatic zoom on peak feature. When this feature is activated and you press on a row in the peak results grid, the graph will zoom and tab to show that peak covering 75 % of the graph canvas. By default, this option is deactivated.



Note: *You can only move the markers with the button.*

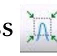
To view specific portions of the graph:

➤ You can define which portion of the graph will be visible by pressing and dragging the graph with the stylus or your finger.

➤ You can also zoom in on a specific area by pressing and defining the zoom area with the stylus or your finger (a rectangle with dotted lines will appear to help you define the area). Once you release the stylus, the application automatically zooms in on the graph.

➤ You can zoom in or out on the center of the portion of the graph that is displayed by using, respectively,  or . The application automatically adjusts the zoom by 50 % and 100 % respectively.

To automatically zoom in on the selected peak:

Select the peak on the graph or in the table results and press .

To revert to the complete graph view:

Press 

.

Managing Trace Files

The application allows you to manage the trace files for all test modes. You can open saved trace files and save current traces.

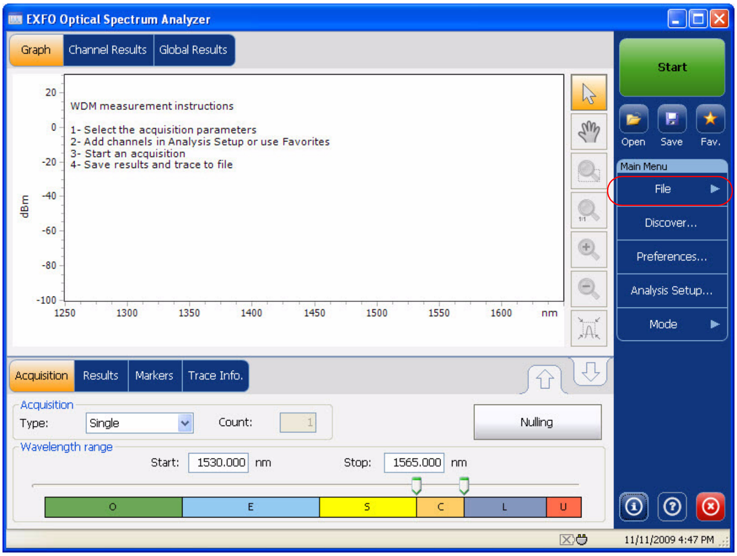
Note: *The procedures explained below for managing the trace files are similar for all test modes.*

Clearing Trace Files

The application allows you to clear the current trace and make room for new traces.

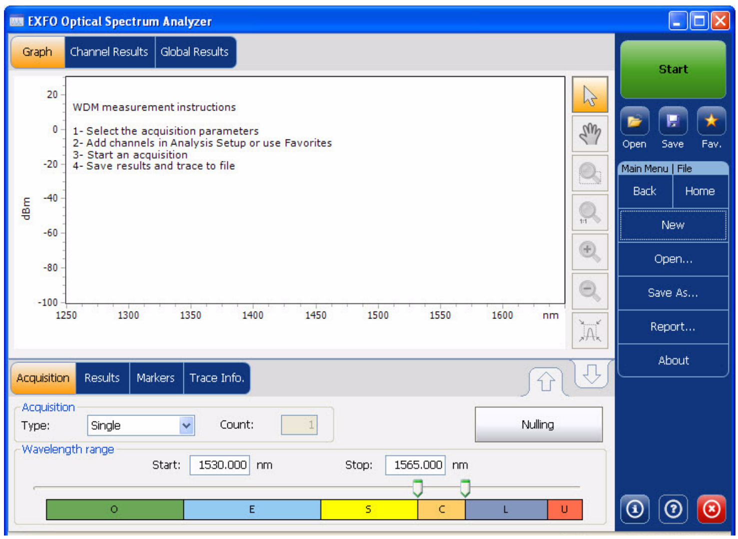
To clear a trace:

1. From the **Main Menu**, Press **File**.



2. Press **New**.

3. If you had already acquired (but not saved) a trace, a warning window appears, asking you if you want to save the current trace. Press **Yes** to save the trace. Once the trace is saved, you can make room for a new trace. Press **No** to create a new trace without saving the previously acquired one. Press **Cancel** to return to the previous window.



Opening Trace Files

Opening a trace file saves you time as you do not need to perform your acquisition again and can continue working where you left off.

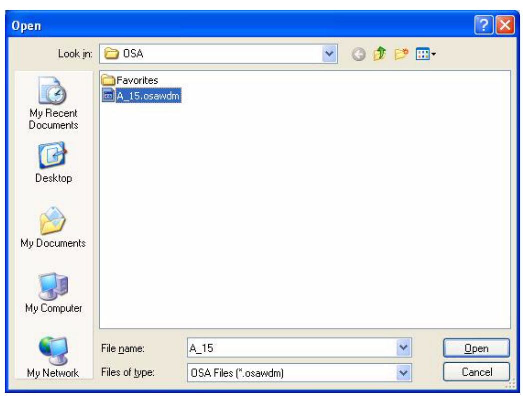
To open a trace file:

1. From the **Main Menu**, Press **File**, and then press **Open**.

   OR

   From the main window, press .

2. If you had already acquired (but not saved) a trace, a warning window appears, asking you if you want to save the current trace. Press **Yes** to save the trace. Once the trace is saved, you can open a new trace. Press **No** to display the new trace without saving the previously acquired one. Press **Cancel** to return to the previous window.



3. Scroll through the list and select a trace to open.

4. Press **Open** to open the file. The trace appears in the **Graph** tab. All the values in the main window except Markers tab will also be updated from the file.

Opening ST, EDFA and Legacy OSA trace files in WDM mode

Your application allows you to open different files types in WDM mode. You can open Spectral Transmittance, EDFA and Legacy OSA file formats also along with the WDM file format, when the application is in the WDM mode.

While loading a spectral transmittance (.osast) file in the WDM mode, the application shall re-analyze the newly imported data using the current **Analysis Setup**.

While loading an EDFA (.osaedfa) file in the WDM mode, the application shall re-analyze the newly imported data using a temporary setup built from the retrieved channel list, retrieved default channel settings and blanks filled using the current **Analysis Setup**.

While loading a legacy OSA (.osw /.osm) file in the WDM mode, the application shall import only the raw trace data and acquisition conditions (date, acquisition type, averaging count and wavelength range). All results will be re-analyzed using the default **Analysis Setup**.

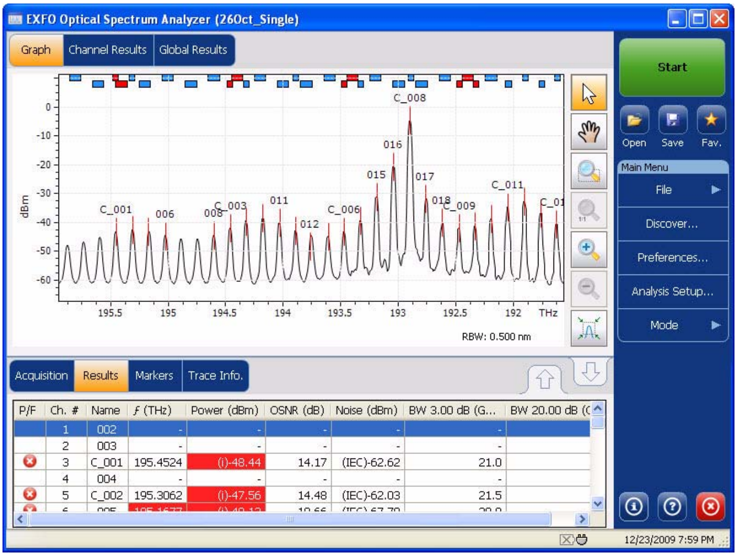
While loading a spectral transmittance or EDFA file, the application verifies the number of traces the file contains. If the file contains a single trace, then the application imports the following data from the selected trace:

- Raw trace data
- Trace information
- Trace identification
- Channel list (if available, in EDFA mode only)
- Default channel settings (if available, in EDFA mode only)

If the file contains two traces (Input and Output), then the application allows you to select which trace to import. Select **Input**, **Output**, or **Cancel** as required. If **Input** or **Output** is selected, application imports the following data from the selected trace.

- Raw trace data
- Trace information
- Trace identification
- Channel list (if available, for EDFA mode only)
- Default channel settings (if available, for EDFA mode only)

An example of EDFA trace file imported in WDM mode is shown below.



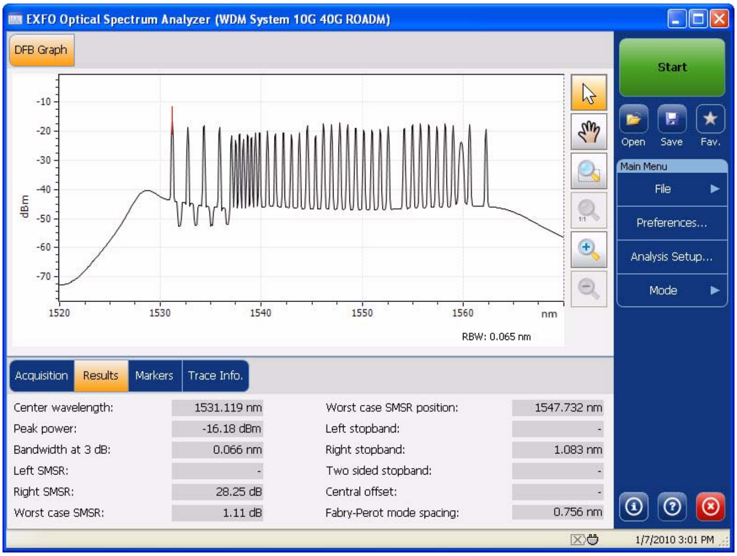
Opening WDM trace file in DFB mode

Your application allows you to open WDM file type in DFB mode.

While loading a WDM (.osawdm) file in the DFB mode, the application shall re-analyze the newly imported data using the current **Analysis Setup** and imports the following data from the selected trace:

- ➤ Raw trace data
- ➤ Trace information
- ➤ Trace identification

An example of WDM trace file imported in DFB mode is shown below.



Opening WDM trace file in ST mode

Your application allows you to open WDM file type in spectral transmittance mode.

While loading a WDM (.osawdm) file in spectral transmittance mode, the application behaves as if a new acquisition is requested. This means that the application does not change the modified state of the current measurement while loading a WDM file.

While loading a WDM file, the application allows you to select in which trace you want to import the WDM file. Select **Input**, **Output**, or **Cancel** as required. If **Input** or **Output** is selected, application imports the following data in the selected trace.

- Raw trace data
- Trace information
- Trace identification

An example of WDM trace file imported in spectral transmittance mode is shown below.

EXFO Optical Spectrum Analyzer (WDM System 10G 40G ROADM)
ST Graph
ST Results
Start
Open
Save
Fav.
Main Menu
File
Preferences...
Analysis Setup...
Mode
dBm
-10
-20
-30
-40
-50
-60
-70
1520
1530
1540
1550
1560
nm
RBW: 0.065 nm
Acquisition
Results
Markers
Trace Info.
Nominal center wavelength:
Offset to nominal wavelength:
Insertion loss min.:
Insertion loss max.:
Bandwidth 1.00 dB:
Bandwidth 5.00 dB:
Adjacent channel isolation:
1/7/2010 3:05 PM

Opening WDM trace file in EDFA mode

Your application allows you to open WDM file type in EDFA mode.

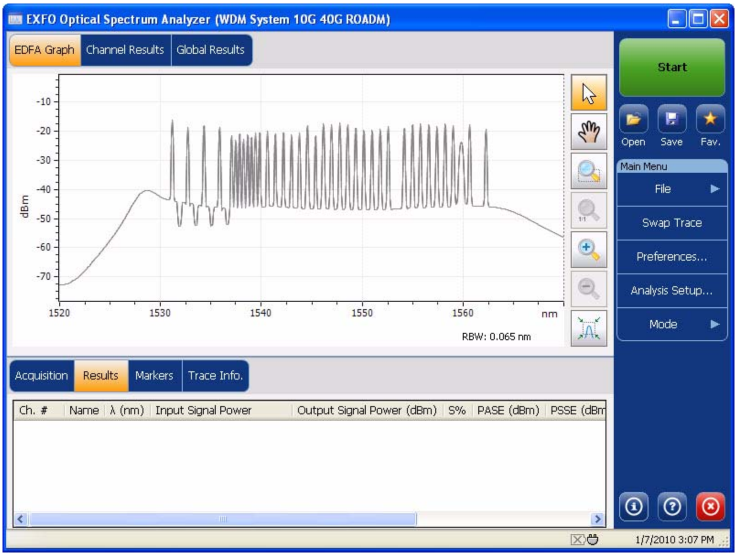
While loading a WDM (.osawdm) file in EDFA mode, the application behaves as if a new acquisition is requested. This means that the application does not change the modified state of the current measurement while loading a WDM file.

While loading a WDM file, the application allows you to select in which trace you want to import the WDM file. Select **Input**, **Output**, or **Cancel** as required. If **Input** or **Output** is selected, application imports the following data in the selected trace.

- Raw trace data
- Trace information
- Trace identification
- Channel list (if available, for input trace only)
- Default channel settings (if available, for input trace only)

When loading a WDM file for the input trace, the application shall overwrite the channel list parameters in the active trace channel. When loading a WDM file for the output trace, the channel list in the active trace is not affected.

An example of WDM trace file imported in EDFA mode is shown below.



Saving Trace Files

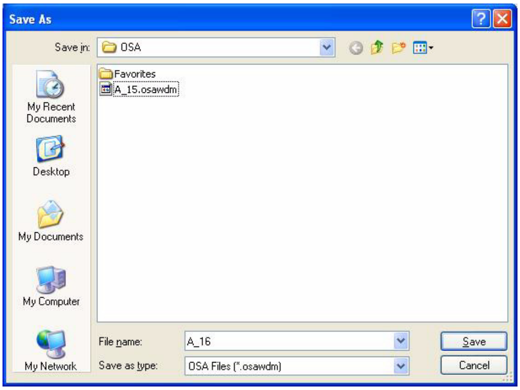
The application allows you to save the trace files for future reference.

To save files:

1. From the **Main Menu**, Press **File**, and then press **Save As**.

OR

From the main window, press .



2. If desired, change the location.

3. If desired, change the file name.

4. Press **Save** to save the trace, else press **Cancel** to exit the window.

Note: *Once a trace is overwritten, you cannot access it anymore.*

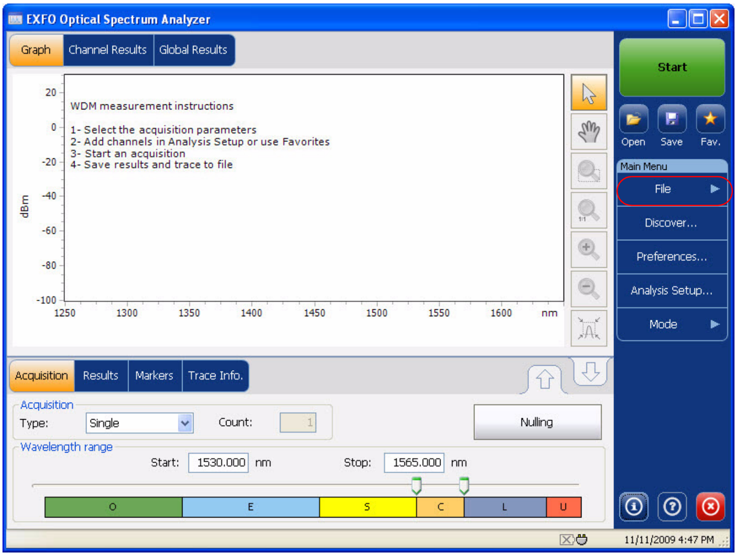
Generating Reports

After performing any acquisition, the application allows you to generate a report for the current acquisition and save it in .html or .txt format depending on the supported file type for WDM, Drift, DFB, ST and EDFA test modes. The report file will include trace information, acquisition conditions and other results and details specific to each test mode.

Note: *The procedure explained below for generating reports is the same for all test modes.*

To generate report:

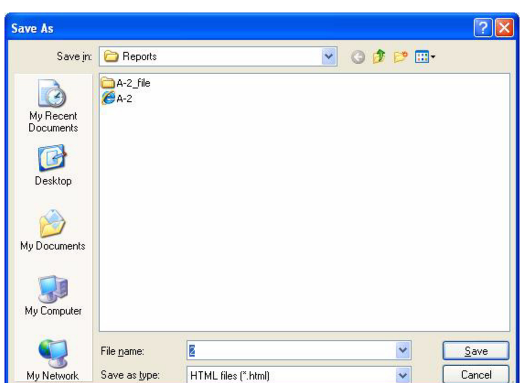
1. From the **Main Menu**, select **File**.



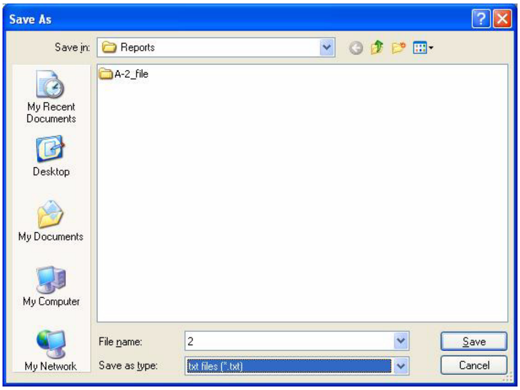
2. Press **Report**.

3. In the **Save As** window, enter a file name.

4. From the **Save as type** list, select **HTML files (*.html)** to save the report in .html format (if the file type is supported for selected test mode).



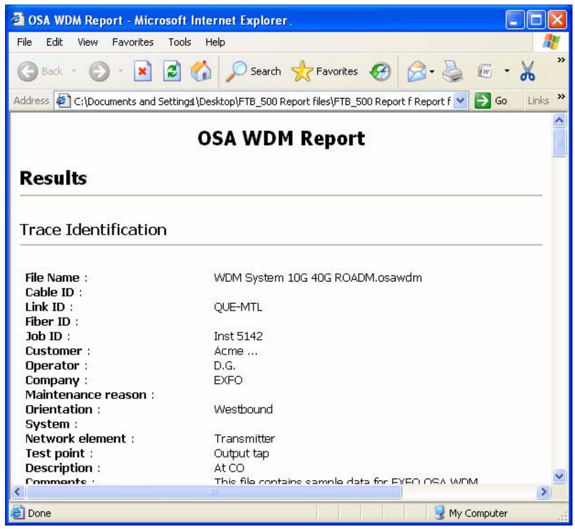
From the **Save as type** list, select **txt files (*.txt)** to save the report in .txt format (if the file type is supported for selected test mode).



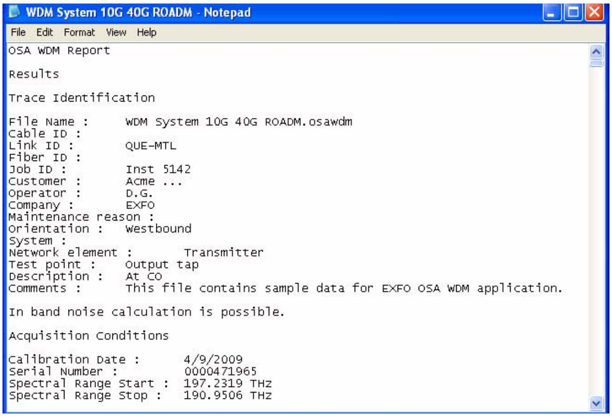
5. Press **Save**. The report will be added to the Reports folder. You can change the location where you want to save the report as desired.

You can view the saved report using the Windows Explorer.

A sample WDM .html report is shown below:



A sample WDM .txt report is shown below:


WDM System 10G 40G ROADM - Notepad

File Edit Format View Help

OSA WDM Report

Results

Trace Identification

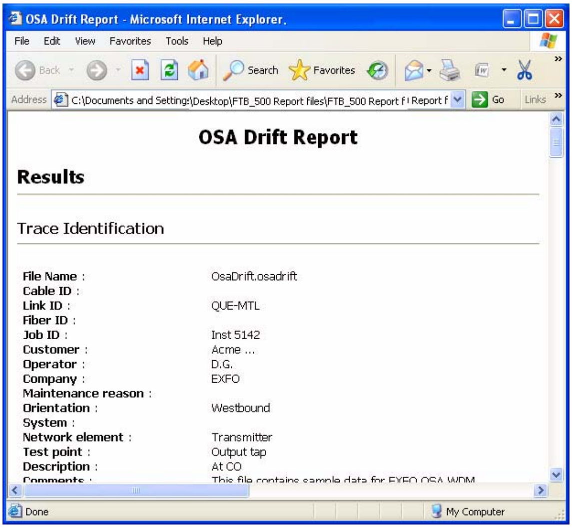
```
File Name :     WDM System 10G 40G ROADM.osawdm
Cable ID :
Link ID :       QUE-MTL
Fiber ID :
Job ID :        Inst 5142
Customer :      Acme ...
Operator :      D.G.
Company :       EXFO
Maintenance reason :
Orientation :   Westbound
System :
Network element :       Transmitter
Test point :    Output tap
Description :   At CO
Comments :      This file contains sample data for EXFO OSA WDM application.
```

In band noise calculation is possible.

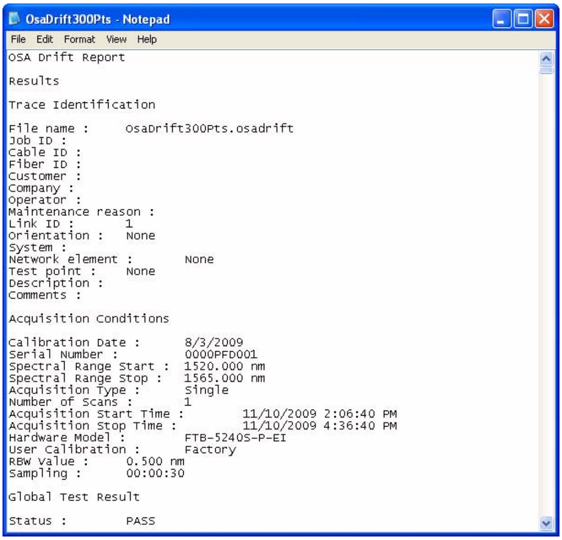
Acquisition Conditions

```
Calibration Date :      4/9/2009
Serial Number :         0000471965
Spectral Range Start :  197.2319 THz
Spectral Range Stop :   190.9506 THz
```

A sample Drift .html report is shown below:



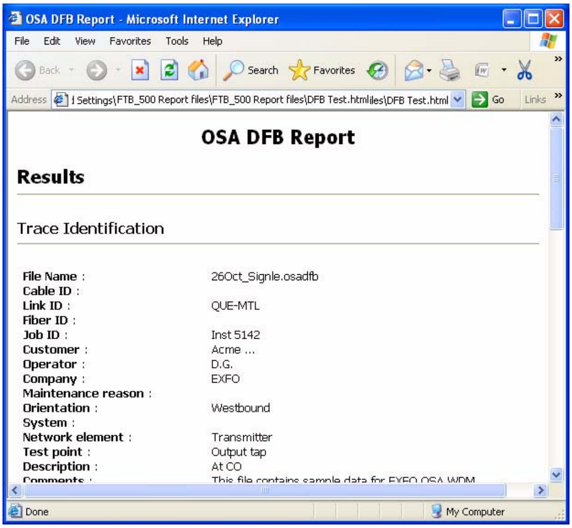
A sample Drift .txt report is shown below:


OsaDrift300Pts - Notepad

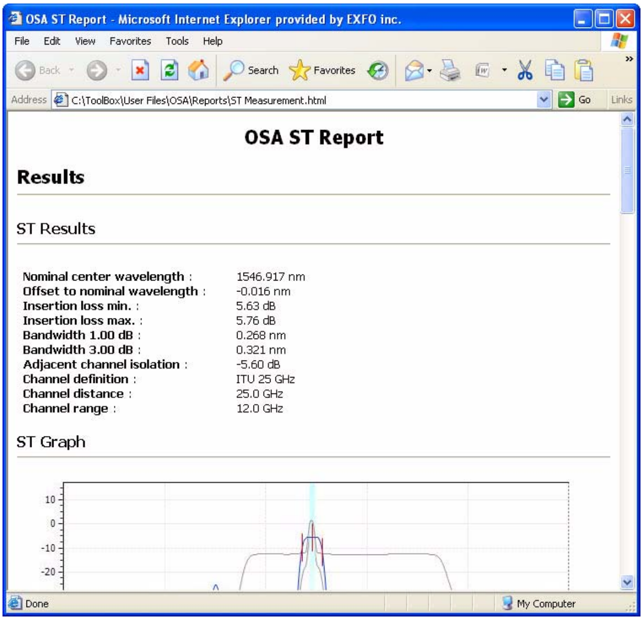
File Edit Format View Help

```
OSA Drift Report

Results

Trace Identification

File name :     OsaDrift300Pts.osadrift
Job ID :
Cable ID :
Fiber ID :
Customer :
Company :
Operator :
Maintenance reason :
Link ID :       1
Orientation :   None
System :
Network element :       None
Test point :    None
Description :
Comments :

Acquisition Conditions

Calibration Date :      8/3/2009
Serial Number :         0000PFD001
Spectral Range Start :  1520.000 nm
Spectral Range Stop :   1565.000 nm
Acquisition Type :      Single
Number of Scans :       1
Acquisition Start Time :        11/10/2009 2:06:40 PM
Acquisition Stop Time :         11/10/2009 4:36:40 PM
Hardware Model :        FTB-5240S-P-EI
User Calibration :      Factory
RBW Value :     0.500 nm
Sampling :      00:00:30

Global Test Result

Status :        PASS
```

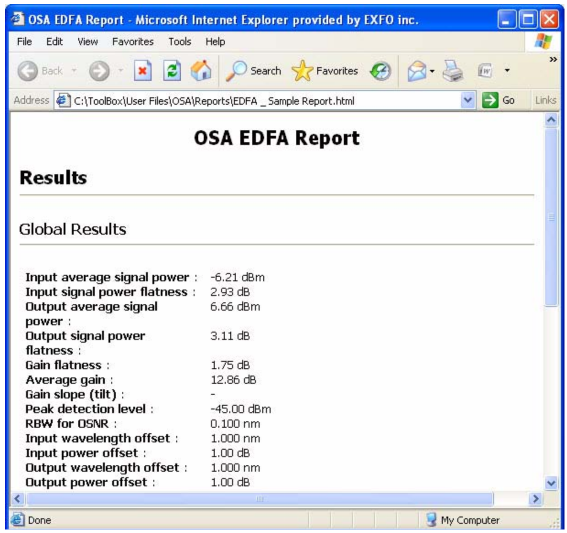
A sample DFB .html report is shown below:



A sample Spectral Transmittance .html report is shown below:



A sample EDFA .html report is shown below:



12 Maintenance

To help ensure long, trouble-free operation:

- Always inspect fiber-optic connectors before using them and clean them if necessary.
- Keep the unit free of dust.
- Clean the unit casing and front panel with a cloth slightly dampened with water.
- Store unit at room temperature in a clean and dry area. Keep the unit out of direct sunlight.
- Avoid high humidity or significant temperature fluctuations.
- Avoid unnecessary shocks and vibrations.
- If any liquids are spilled on or into the unit, turn off the power immediately and let the unit dry completely.

WARNING

Use of controls, adjustments, and procedures for operation and maintenance other than those specified herein may result in hazardous radiation exposure.

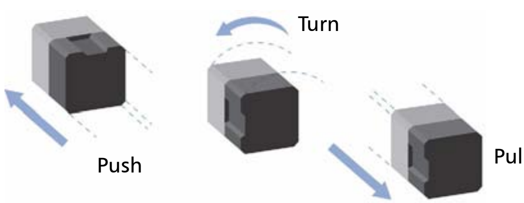
Cleaning EUI Connectors

Regular cleaning of EUI connectors will help maintain optimum performance. There is no need to disassemble the unit.

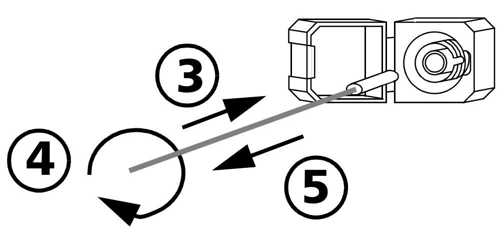
> **IMPORTANT**
> **If any damage occurs to internal connectors, the module casing will have to be opened and a new calibration will be required.**

To clean EUI connectors:

1. Remove the EUI from the instrument to expose the connector baseplate and ferrule.



2. Moisten a 2.5 mm cleaning tip with *one drop* of isopropyl alcohol (alcohol may leave traces if used abundantly).
3. Slowly insert the cleaning tip into the EUI adapter until it comes out on the other side (a slow clockwise rotating movement may help).



4. Gently turn the cleaning tip one full turn, then continue to turn as you withdraw it.

5. Repeat steps 3 to 4 with a dry cleaning tip.

Note: *Make sure you don't touch the soft end of the cleaning tip.*

6. Clean the ferrule in the connector port as follows:

   6a. Deposit *one drop* of isopropyl alcohol on a lint-free wiping cloth.

IMPORTANT

Isopropyl alcohol may leave residues if used abundantly or left to evaporate (about 10 seconds).

Avoid contact between the tip of the bottle and the wiping cloth, and dry the surface quickly.

   6b. Gently wipe the connector and ferrule.

   6c. With a dry lint-free wiping cloth, gently wipe the same surfaces to ensure that the connector and ferrule are perfectly dry.

   6d. Verify connector surface with a portable fiber-optic microscope(for example, EXFO's FOMS) or fiber inspection probe(for example, EXFO's FIP).

WARNING

Verifying the surface of the connector WHILE THE UNIT IS ACTIVE WILL result in permanent eye damage.

7. Put the EUI back onto the instrument (push and turn clockwise).

8. Throw out cleaning tips and wiping cloths after one use.

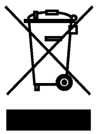
Recalibrating the Unit

Manufacturing and service center calibrations are based on the ISO/IEC 17025 Standard, which states that calibration documents must not contain a recommended calibration interval, unless this has been previously agreed upon with the customer.

Validity of specifications depends on operating conditions. For example, the calibration validity period can be longer or shorter depending on the intensity of use, environmental conditions and unit maintenance. You should determine the adequate calibration interval for your unit according to your accuracy requirements.

Under normal use, EXFO recommends calibrating your unit every year.

Recycling and Disposal (Applies to European Union Only)



Recycle or dispose of your product (including electric and electronic accessories) properly, in accordance with local regulations. Do not dispose of it in ordinary garbage receptacles.

This equipment was sold after August 13, 2005 (as identified by the black rectangle).

- ➤ Unless otherwise noted in a separate agreement between EXFO and a customer, distributor, or commercial partner, EXFO will cover costs related to the collection, treatment, recovery, and disposal of end-of-lifecycle waste generated by electronic equipment introduced after August 13, 2005 to an European Union member state with legislation regarding Directive 2002/96/EC.
- ➤ Except for reasons of safety or environmental benefit, equipment manufactured by EXFO, under its brand name, is generally designed to facilitate dismantling and reclamation.

For complete recycling/disposal procedures and contact information, visit the EXFO Web site at www.exfo.com/recycle.

13 Troubleshooting

Viewing Online Documentation

A PDF version of the FTB-5240S/5240BP Optical Spectrum Analyzer user guide is conveniently available at all times.

To access the online user guide:

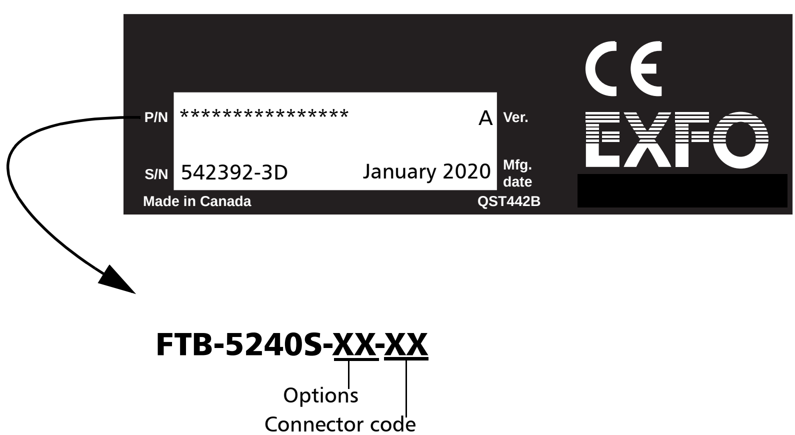
Go to “C:\Program Files\EXFO\ToolBox\Help”. This folder contains a PDF version of the user guide.

Contacting the Technical Support Group

To obtain after-sales service or technical support for this product, contact EXFO at one of the following numbers. The Technical Support Group is available to take your calls from Monday to Friday, 8:00 a.m. to 7:00 p.m. (Eastern Time in North America).

For detailed information about technical support, visit the EXFO Web site at www.exfo.com.

To accelerate the process, please have information such as the name and the serial number (see the product identification label—an example is shown below), as well as a description of your problem, close at hand.

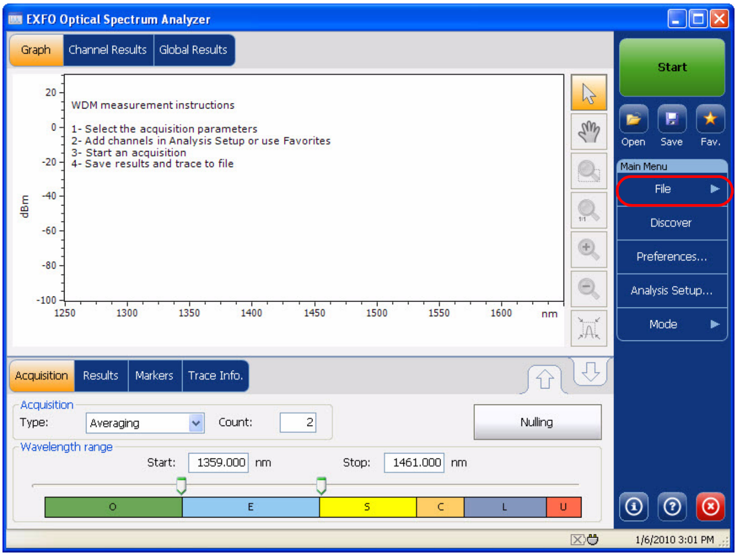


About

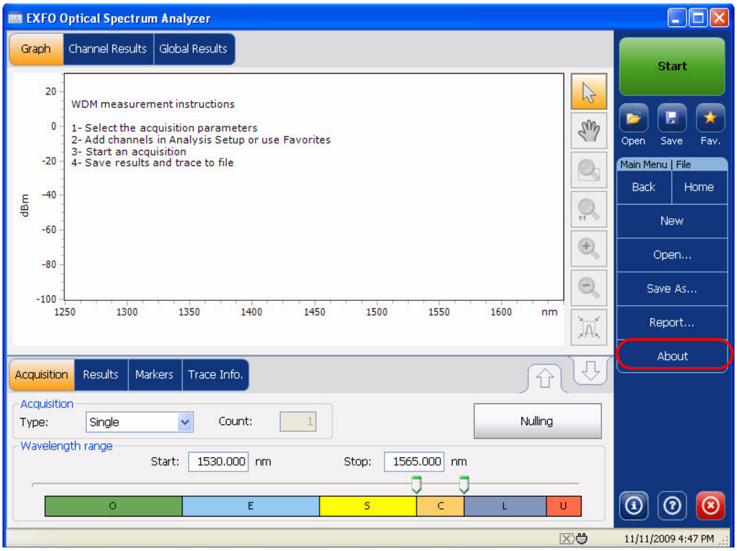
You can view the product version details and technical support information in this window.

To view the information about the product:

1. From the **Main Menu**, press **File**.



2. Press **About**.



Transportation

Maintain a temperature range within specifications when transporting the unit. Transportation damage can occur from improper handling. The following steps are recommended to minimize the possibility of damage:

- Pack the unit in its original packing material when shipping.
- Avoid high humidity or large temperature fluctuations.
- Keep the unit out of direct sunlight.
- Avoid unnecessary shocks and vibrations.

14 Warranty

General Information

EXFO Inc. (EXFO) warrants this equipment against defects in material and workmanship for a period of one year from the date of original shipment. EXFO also warrants that this equipment will meet applicable specifications under normal use.

During the warranty period, EXFO will, at its discretion, repair, replace, or issue credit for any defective product, as well as verify and adjust the product free of charge should the equipment need to be repaired or if the original calibration is erroneous. If the equipment is sent back for verification of calibration during the warranty period and found to meet all published specifications, EXFO will charge standard calibration fees.

IMPORTANT

The warranty can become null and void if:

- **unit has been tampered with, repaired, or worked upon by unauthorized individuals or non-EXFO personnel.**
- **warranty sticker has been removed.**
- **case screws, other than those specified in this guide, have been removed.**
- **case has been opened, other than as explained in this guide.**
- **unit serial number has been altered, erased, or removed.**
- **unit has been misused, neglected, or damaged by accident.**

THIS WARRANTY IS IN LIEU OF ALL OTHER WARRANTIES EXPRESSED, IMPLIED, OR STATUTORY, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. IN NO EVENT SHALL EXFO BE LIABLE FOR SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES.

Liability

EXFO shall not be liable for damages resulting from the use of the product, nor shall be responsible for any failure in the performance of other items to which the product is connected or the operation of any system of which the product may be a part.

EXFO shall not be liable for damages resulting from improper usage or unauthorized modification of the product, its accompanying accessories and software.

Exclusions

EXFO reserves the right to make changes in the design or construction of any of its products at any time without incurring obligation to make any changes whatsoever on units purchased. Accessories, including but not limited to fuses, pilot lamps, batteries and universal interfaces (EUI) used with EXFO products are not covered by this warranty.

This warranty excludes failure resulting from: improper use or installation, normal wear and tear, accident, abuse, neglect, fire, water, lightning or other acts of nature, causes external to the product or other factors beyond the control of EXFO.

IMPORTANT

EXFO will charge a fee for replacing optical connectors that were damaged due to misuse or bad cleaning.

Certification

EXFO certifies that this equipment met its published specifications at the time of shipment from the factory.

Service and Repairs

EXFO commits to providing product service and repair for five years following the date of purchase.

To send any equipment for service or repair:

1. Call one of EXFO's authorized service centers (see *EXFO Service Centers Worldwide* on page 435). Support personnel will determine if the equipment requires service, repair, or calibration.
2. If equipment must be returned to EXFO or an authorized service center, support personnel will issue a Return Merchandise Authorization (RMA) number and provide an address for return.
3. If possible, back up your data before sending the unit for repair.
4. Pack the equipment in its original shipping material. Be sure to include a statement or report fully detailing the defect and the conditions under which it was observed.
5. Return the equipment, prepaid, to the address given to you by support personnel. Be sure to write the RMA number on the shipping slip. *EXFO will refuse and return any package that does not bear number.*

Note: *A test setup fee will apply to any returned unit that, after test, is found to meet the applicable specifications.*

After repair, the equipment will be returned with a repair report. If the equipment is not under warranty, you will be invoiced for the cost appearing on this report. EXFO will pay return-to-customer shipping costs for equipment under warranty. Shipping insurance is at your expense.

Routine recalibration is not included in any of the warranty plans. Since calibrations/verifications are not covered by the basic or extended warranties, you may elect to purchase FlexCare Calibration/Verification Packages for a definite period of time. Contact an authorized service center (see *EXFO Service Centers Worldwide* on page 435).

EXFO Service Centers Worldwide

If your product requires servicing, contact your nearest authorized service center.

EXFO Headquarters Service Center
400 Godin Avenue
Quebec (Quebec) G1M 2K2
CANADA

1 866 683-0155 (USA and Canada)
Tel.: 1 418 683-5498
Fax: 1 418 683-9224
quebec.service@exfo.com

EXFO Europe Service Center
Omega Enterprise Park, Electron Way
Chandlers Ford, Hampshire S053 4SE
ENGLAND

Tel.: +44 2380 246810
Fax: +44 2380 246801
europe.service@exfo.com

EXFO China Service Center/
Beijing OSIC
Beijing New Century Hotel
Office Tower, Room 1754-1755
No. 6 Southern Capital Gym Road
Beijing 100044
P. R. CHINA

Tel.: +86 (10) 6849 2738
Fax: +86 (10) 6849 2662
beijing.service@exfo.com

A Technical Specifications

IMPORTANT

The following technical specifications can change without notice. The information presented in this section is provided as a reference only. To obtain this product's most recent technical specifications, visit the EXFO Web site at *www.exfo.com*.

SPECIFICATIONS [a]

Spectral Measurement	FTB-5240S and FTB- 5240S-P	FTB-5240BP	
Wavelength range (nm)	1250 to 1650	1250 to 1650	
Wavelength uncertainty (nm) [b]	±0.05 ±0.01 [c, d]	±0.03 ±0.01 [c, d]	
Reference	Internal [e]	Internal [e]	
Resolution bandwidth (FWHM) (nm) [f]	0.065 [b, d]	0.033 [b, d]	
Wavelength linearity (nm)	±0.01 [b, d]	±0.01 [b, d]	
Wavelength repeatability 2σ (nm)	±0.003 [g]	±0.002 [g]	

Power Measurements	FTB-5240S and FTB-5240S-P	FTB-5240BP	HPW Option
Dynamic range (dBm) (per channel) [b]	−80 [h] to +18	−80 [h] to +18	−70 [h] to +23
Maximum total safe power (dBm)	+23	+23	+29
Absolute power uncertainty (dB)	±0.5 [i]	±0.5 [i]	±0.5 [i]
Power repeatability 2σ (dB) [d, g]	±0.05	±0.04 [j]	±0.05 [j]

Optical Measurements	FTB-5240S and FTB-5240S-P	FTB-5240BP	HPW Option
Optical rejection ratio at 1550 nm (dB)			
at 0.2 nm (25 GHz)	35 (40 typical)	45 (50 typical)	35 (40 typical)
at 0.4 nm (50 GHz)	45 (50 typical)	50 (55 typical)	45 (50 typical)
Channel spacing			50 to 200 GHz CWDM
PDL at 1550 nm (dB)	±0.08 [d]	±0.06 [d]	
ORL (dB)	≥40	≥40	
Measurement time (s) [d, j] (includes scanning, analysis and display)	<1 (with the FTB-500 platform)	<1 (with the FTB-500 platform)	

In-Band OSNR Measurements [d, k]	FTB-5240S-P only	FTB-5240BP
OSNR dynamic range (dB)	>35 [l]	>35 [l]
OSNR measurement uncertainty (dB)	±0.5 [m]	±0.5 [m]
Repeatability (dB)	±0.2 [n]	±0.2 [n]
Data signals	Up to 100 Gbit/s [o]	Up to 100 Gbit/s [o]
Measurement time (s) [d, j] (includes scanning, analysis and display)	<6 (eight scans with the FTB-500 platform) <75 (eight scans with the FTB-200 platform)	<6 (eight scans with the FTB-500 platform)

Notes

a. All specifications are for a temperature of 23 °C ± 2 °C with an FC/UPC connector unless otherwise specified, after warm-up.

b. From 1520 to 1610 nm.

c. After user calibration in the same test session within 10 nm from each calibration point.

d. Typical.

e. Integrated and wavelength-independent self-adjustment.

f. Full width at half maximum.

g. Over one minute in continuous acquisition mode.

h. With averaging.

i. At 1550 nm, −10 dBm input.

j. 45 nm span, full resolution, 20 peak analysis.

k. In-band OSNR measurement performed with 64 scans.

l. For an optical noise level of > −60 dBm.

m. With PMD ≤15 ps and no crosstalk, uncertainty specification is valid for OSNR ≤ 25 dB. With PMD ≤15 ps and crosstalk, uncertainty specification is valid for OSNR ≤ 20 dB.

n. Repeatability specification is valid for OSNR ≤ 25 dB.

o. Except for pol-mux and fast polarization scrambled signals.

GENERAL SPECIFICATIONS

Temperature			
	operating	0 °C to 40 °C	(32 °F to 104 °F)
	storage	−20 °C to 50 °C	(−4 °F to 120 °F)
Relative humidity		0 % to 95 % non-condensing	
Battery life (hours)		5 (with the FTB-500 platform)	
Connectors		EI (EXFO UPC Universal Interface) EA (EXFO APC Universal Interface)	
Size (H x W x D)	FTB-5240S module	96 mm x 51 mm x 260 mm	(3 3/4 in x 2 in x 10 1/4 in)
	FTB-5240BP module	96 mm x 76 mm x 260 mm	(3 3/4 in x 3 in x 10 1/4 in)
Weight	FTB-5240S module	1.5 kg	(3.3 lb)
	FTB-5240BP module	1.7 kg	(3.8 lb)

LASER SAFETY

LASER SAFETY
21 CFR 1040.10 and IEC 60825-1
CLASS 1 LASER PRODUCT

B SCPI Command Reference

This appendix presents detailed information on the commands and queries supplied with your FTB-5240S/5240BP Optical Spectrum Analyzer.

IMPORTANT

Since the FTB-500 can house many instruments, you must explicitly specify which instrument you want to remotely control.

You must add the following mnemonic *at the beginning of any command or query* that you send to an instrument:

LINStrument<LogicalInstrumentPos>:

where <*LogicalInstrumentPos*> corresponds to the identification number of the instrument.

FTB-500 backplane identification number

1Y

Instrument slot number:

For information on modifying unit identification, refer to your platform user guide.

Quick Reference Command Tree

Command						Parameter(s)	P.
ABORt							443
CALCulate[1..n]	BWAT					<PowerLevel[<wsp>DB]>	444
	BWAT?						445
	CHANnel?					<ChannelIndex>	446
	CHANnel	ADD				<Name>,<CentralWav/Freq.[<wsp>M|HZ]>,<Width>,M|HZ, INTEGRATEDSIGNAL|PEAKSIGNAL|TOTALCHANNEL,FIXEDIEC|INBAND|INBANDNF	448
		COUNt?					451
		DEFault	NFOSnr			FIXEDIEC|INBAND|INBANDNF	452
			NFOSnr?				453
			SIGPowercalc			INTEGRATEDSIGNAL|PEAKSIGNAL|TOTALCHANNEL	454
			SIGPowercalc?				456
			[STATe]			<State>	457
			[STATe]?				458
			WIDTh			<ChannelWidth>,M|Hz	459
			WIDTh?				460
		DELete				<ChannelIndex>	461
			ALL				462
	[IMMediate]	AUTO				<State>	463
		AUTO?					464
	PEAK	[DETEction]	LEVel			<PeakDetectionLevel>	465

Command						Parameter(s)	P.
			LEVel?				466
	POWer	OFFSet	[OUTPut]			<Offset>	467
			[OUTPut]?				468
	RBW					<RbwForOsnr>	469
	RBW?						470
	RBW	INSTrument				<RbwInstrumentState>	471
		INSTrument?					473
	WAVelength	OFFSet	[OUTPut]			<WavelegthOffset>	474
			[OUTPut]?				475
	WDM						476
		CHANnel?					477
		CHANnel	COUNt?				479
		OSNR	AVERage?				480
			FLATness?				481
		SIGPower	AVERage?				482
			FLATness?				483
			INTegrated?				484
CALibration	ZERO	[IMMediate]					485
INITiate[1..n]	[IMMediate]						486
MMEMory[1..n]	LOAD	CONFiguratio				<Filename>	487
		OVERwrite				<Overwrite>	488
		OVERwrite?					489
		TRACe				TRCMAIN,<FileName>	490

Command						Parameter(s)	P.
	STORe	CONFiguratio				<Filename>	492
		OVERwrite				<Overwrite>	493
		OVERwrite?					494
		TRACe				TRCMAIN,<Filename>	495
SENSe[1..n]	AVERage	COUNt				<AverageCount>	496
		COUNt?					497
		INBand	COUNt			<InbandCount>	498
			COUNt?				499
		TYPE				SINGLE|AVERAGING|INBAND	500
		TYPE?					501
	WAVelength	[RANGe]	STARt			<Start[<wsp>NM]>|MAXimum|MINimum	502
			STARt?				504
			STOP			<Stop[<wsp>NM]>|MAXimum|MINimum	505
			STOP?				507
SNUMber?							508
STATus?							509
TRACe[1..n]	[DATA]?					TRCMAIN	511
	[DATA]	PREamble?				TRCMAIN	512
	INFormation?					TRCMAIN	514
	POINts?					TRCMAIN	516
UNIT[1..n]	SPECtrum					M|HZ	517
	SPECtrum?						518

Product-Specific Commands—Description

:ABORt

Description	This command is used to stop running scan, measurement or acquisition in progress. This command is an event and has no associated *RST condition or query form. However, the equivalent of the ABORt command is performed on any acquisition in progress.
Syntax	:ABORt
Parameter(s)	None
Example(s)	ABOR
See Also	INITiate[1..n]:IMMediate INITiate[1..n]:STATe?

:CALCulate[1..n]:BWAT

Description	This command sets the power used to calculate signal bandwidth inside a channel. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, value for Bandwidth at is 20.0 dB.
Syntax	:CALCulate[1..n]:BWAT<wsp><PowerLevel[<wsp>DB]>
Parameter(s)	*PowerLevel:* The program data syntax for <PowerLevel> is defined as a <numeric_value> element followed by an optional <SUFFIX PROGRAM DATA> element. The allowed <SUFFIX PROGRAM DATA> element is DB. Power level used to calculate bandwidth. Range: [0.1 … 50.0] dB
Example(s)	CALC:BWAT 15.00 dB
See Also	CALCulate[1..n]:BWAT? CALCulate[1..n]:WDM:CHANnel?

:CALCulate[1..n]:BWAT?

Description	This query returns the power used to calculate signal bandwidth inside a channel. At *RST, value for Bandwidth at is 20.0 dB.
Syntax	:CALCulate[1..n]:BWAT?
Parameter(s)	None
Response Syntax	<PowerLevel>
Response(s)	*PowerLevel:* The response data syntax for <PowerLevel> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Power level for bandwidth calculation (in dB).
Example(s)	CALC:BWAT?
See Also	CALCulate[1..n]:BWAT CALCulate[1..n]:WDM:CHANnel?

:CALCulate[1..n]:CHANnel?

Description	This query returns available information on a channel. *RST clears the channel list.
Syntax	:CALCulate[1..n]:CHANnel?<wsp><ChannelIndex>
Parameter(s)	*ChannelIndex:* The program data syntax for <ChannelIndex> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Selects the Index of channel. The maximum value for the index varies with the number of channels on the list. The list contains a maximum of 1000 channels. Range: [1 ... 1000]
Response Syntax	<ChannelInfo>
Response(s)	*ChannelInfo:* The response data syntax for <ChannelInfo> is defined as a <DEFINITE LENGTH ARBITRARY BLOCK RESPONSE DATA> element. Information on the channel in A,B,C,D,E,F format, where: A=Channel Name (string)

:CALCulate[1..n]:CHANnel?

	B=Channel Center (always in m) <NR3 NUMERIC RESPONSE DATA> C=Channel Width <NR3 NUMERIC RESPONSE DATA> D=Channel Width unit (m or Hz) (string) E=PowerType (PEAKSIGNAL or INTEGRATEDSIGNAL or TOTALCHANNEL) (string) F=NoisePosition (FIXEDIEC or INBANDNF or INBAND) (string)
Example(s)	CALC:CHAN? 1
See Also	CALCulate[1..n]:CHANnel:ADD

:CALCulate[1..n]:CHANnel:ADD

Description	Adds a new channel to list of channels. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. This command is an event and has no associated *RST condition or query form.
Syntax	:CALCulate[1..n]:CHANnel:ADD<wsp><Name>,<CentralWav/Freq.[<wsp>M|HZ]>,<Width>,M|HZ,INTEGRATEDSIGNAL|PEAKSIGNAL|TOTALCHANNEL,FIXEDIEC|INBAND|INBANDNF
Parameter(s)	➤ *Name:* The program data syntax for <Name> is defined as a <STRING PROGRAM DATA> element. Name of the new channel (case sensitive). Maximum length: 5 characters. ➤ *CentralWav/Freq.:* The program data syntax for <CentralWav/Freq.> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element followed by an optional <SUFFIX PROGRAM DATA> element. The allowed <SUFFIX PROGRAM DATA> elements are: M|HZ.

:CALCulate[1..n]:CHANnel:ADD

Central wavelenght of the channel being added. Range is dependent on OSA module limits.

➤ *Width:*

The program data syntax for <Width> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element.

Width of the channel being added.
Range: [0.008 … 400.000] nm or [1.0 … 58150.0] GHz

➤ *WidthUnit:*

The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are: M|HZ.

Width unit of the channel being added.

➤ *SignalPowerCalculation:*

The program data syntax for the second parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are:
INTEGRATEDSIGNAL|PEAKSIGNAL|TOTALCHANNEL.

Selects the Signal Power Calculation method to apply for signal power value.
INTEGRATEDSIGNAL, selects Integrated signal power method to calculate signal power value.

PEAKSIGNAL, selects Peak signal power method to calculate signal power value.
TOTALCHANNEL, selectsTotal channel power method to calculate signal power value.

:CALCulate[1..n]:CHANnel:ADD

- *NoiseForOSNR:*

The program data syntax for the third parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are: FIXEDIEC|INBAND|INBANDNF.

Selects the Noise for OSNR for the added channel.
FIXEDIEC, selects Fixed range IEC based (IEC) as the OSNR calculation method.
INBAND, selects In-Band (InB) as OSNR calculation method.
INBANDNF, selects In-Band narrow filter (InB nf) as OSNR calculation method.

Example(s)

CALC:CHAN:ADD C23,1550.235 NM,50.1G,Hz,INTEGRATEDSIGNAL,FIXEDIEC

Notes

The new channel must not be in conflict with an existing channel.
The channel being added is in conflict if its name already exists in another channel.

See Also

CALCulate[1..n]:CHANnel:DELete
CALCulate[1..n]:CHANnel:DELete:ALL
CALCulate[1..n]:CHANnel:COUNt?

:CALCulate[1..n]:CHANnel:COUNt?

Description	This query returns the number of channels on the list. *RST clears the channel list.
Syntax	:CALCulate[1..n]:CHANnel:COUNt?
Parameter(s)	None
Response Syntax	<ChannelCount>
Response(s)	*ChannelCount:* The response data syntax for <ChannelCount> is defined as a <NR1 NUMERIC RESPONSE DATA> element. Number of channels on the list.
Example(s)	CALCulate:CHANnel:COUNt? CALCulate:CHANnel:ADD C34,1550.235 NM,50.1G,Hz,INTEGRATEDSIGNAL,FIXEDIEC
See Also	CALCulate[1..n]:CHANnel:ADD

:CALCulate[1..n]:CHANnel:DEFault:NFOSnr

Description	This command sets the Noise For OSNR value. Possible values for sole parameter: FIXEDIEC, INBAND or INBANDNF. At *RST, this value is FIXEDIEC.
Syntax	:CALCulate[1..n]:CHANnel:DEFault:NFOSnr<wsp>FIXEDIEC|INBAND|INBANDNF
Parameter(s)	*NoiseForOSNR* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are: FIXEDIEC|INBAND|INBANDNF. Selects the Noise for OSNR for the default channel. FIXEDIEC, selects Fixed range IEC based (IEC) as the OSNR calculation method. INBAND, selects In-Band (InB) as OSNR calculation method. INBANDNF, selects In-Band narrow filter (InB nf) as OSNR calculation method.
Example(s)	CALC:CHAN:DEFault:NFOSnr FIXEDIEC
See Also	CALCulate[1..n]:CHANnel:DEFault:NFOSnr?

:CALCulate[1..n]:CHANnel:DEFault:NFOSnr?

Description	This query returns the calculation method to use for Noise for OSNR value. At *RST, this value is FIXEDIEC.
Syntax	:CALCulate[1..n]:CHANnel:DEFault:NFOSnr?
Parameter(s)	None
Response Syntax	<NoiseForOsnr>
Response(s)	*NoiseForOsnr:* The response data syntax for <NoiseForOsnr> is defined as a <CHARACTER RESPONSE DATA> element. Display the Noise for OSNR value for the default channel. The returned value can be either FIXEDIEC, INBAND or INBANDNF.
Example(s)	CALC:CHAN:DEFault:NFOSnr?
See Also	CALCulate[1..n]:CHANnel:DEFault:NFOSnr

:CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc

Description	This command sets the Signal Power Calculation method to apply for signal power value. Possible values for sole parameter: INTEGRATEDSIGNAL, PEAKSIGNAL or TOTALCHANNEL. At *RST, this value is INTEGRATEDSIGNAL.
Syntax	:CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc<wsp>INTEGRATEDSIGNAL|PEAKSIGNAL|TOTALCHANNEL
Parameter(s)	*SignalPowerCalculation:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are: INTEGRATEDSIGNAL|PEAKSIGNAL|TOTALCHANNEL. Selects the Signal Power Calculation method to apply for signal power value. INTEGRATEDSIGNAL, selects Integrated signal power method to calculate signal power value.

:CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc

	PEAKSIGNAL, selects Peak signal power method to calculate signal power value. TOTALCHANNEL, selectsTotal channel power method to calculate signal power value.
Example(s)	CALC:CHAN:DEFault:SIGPowercalc INTEGRATEDSIGNAL
See Also	CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc ?

:CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc?

Description	This query returns the Signal Power Calculation. Possible returned values can be INTEGRATEDSIGNAL, PEAKSIGNAL or TOTALCHANNEL. At *RST, this value is INTEGRATEDSIGNAL.
Syntax	:CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc?
Parameter(s)	None
Response Syntax	<SigmalPowerCalculation>
Response(s)	*SigmalPowerCalculation:* The response data syntax for <SigmalPowerCalculation> is defined as a <CHARACTER RESPONSE DATA> element. Display the Signal power calculation value for the dafault channel. The returned value can be INTEGRATEDSIGNAL, PEAKSIGNAL or TOTALCHANNEL.
Example(s)	CALC:CHAN:DEFault:SIGPowercalc?
See Also	CALCulate[1..n]:CHANnel:DEFault:SIGPowercalc

:CALCulate[1..n]:CHANnel:DEFault[:STATe]

Description	This command activates or deactivates the default channel. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 1 (activated).
Syntax	:CALCulate[1..n]:CHANnel:DEFault[:STATe]<wsp><State>
Parameter(s)	*State:* The program data syntax for <State> is defined as a <Boolean Program Data> element. The <State> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0. The <State> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0. The <State> parameter is a boolean value that determines the status of the default channel: 0- deactivates default channel 1- activates default channel
Example(s)	CALC:CHAN:DEF:STATe ON
See Also	CALCulate[1..n]:CHANnel:DEFault:STATe?

:CALCulate[1..n]:CHANnel:DEFault[:STATe]?

Description	This query returns the state of the Activate default channel. At *RST, this value is 1 (activated).
Syntax	:CALCulate[1..n]:CHANnel:DEFault[:STATe]?
Parameter(s)	None
Response Syntax	<State>
Response(s)	*State:* The response data syntax for <State> is defined as a <BINARY NUMERIC RESPONSE DATA> element. The current <State> status, where: 0- the default channel is deactivated 1- the default channel is activated
Example(s)	CALC:CHAN:DEF?
See Also	CALCulate[1..n]:CHANnel:DEFault:STATe

:CALCulate[1..n]:CHANnel:DEFault:WIDTh

Description	This command sets the Channel width value with unit. At *RST,this value is 100.0 GHz.
Syntax	:CALCulate[1..n]:CHANnel:DEFault:WIDTh<wsp><ChannelWidth>,M|Hz
Parameter(s)	➤ *ChannelWidth:* The program data syntax for <ChannelWidth> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Set the default channel width. Range: [0.008 ... 400.000] nm or [1.0 ... 58150.0] GHz ➤ *ChannelWidthUnit:* The program data syntax for the second parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are: M|Hz. Default Channel Width Unit (M/Hz) being added dependent on OSA module limits.
Example(s)	CALC:CHAN:DEFault:WIDTh 0.4N, m
See Also	CALCulate[1..n]:CHANnel:DEFault:WIDTh?

:CALCulate[1..n]:CHANnel:DEFault:WIDTh?

Description	This query returns the Channel width value with unit. At *RST, this value is 100.0 GHz.
Syntax	:CALCulate[1..n]:CHANnel:DEFault:WIDTh?
Parameter(s)	None
Response Syntax	<ChannelWidth>
Response(s)	*ChannelWidth:* The response data syntax for <ChannelWidth> is defined as a <DEFINITE LENGTH ARBITRARY BLOCK RESPONSE DATA>” element. Information on the channel width in A,B, format, where: A=Channel Width <NR3 NUMERIC RESPONSE DATA> B=Channel Width unit (m or Hz)(string)
Example(s)	CALC:CHAN:DEFault:WIDTh?
See Also	CALCulate[1..n]:CHANnel:DEFault:WIDTh

:CALCulate[1..n]:CHANnel:DELete

Description	This command deletes a channel from channel list. The OSA module must not be in acquisition for this command to be accepted. This command is an event and has no associated *RST condition or query form.
Syntax	:CALCulate[1..n]:CHANnel:DELete<wsp><ChannelIndex>
Parameter(s)	*ChannelIndex:* The program data syntax for <ChannelIndex> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Selects the Index of channel. The maximum value for the index varies with the number of channels on the list. The list contains a maximum of 1000 channels. Range: [1 ... 1000]
Example(s)	CALC:CHAN:DEL 2
See Also	CALCulate[1..n]:CHANnel:DELete:ALL

:CALCulate[1..n]:CHANnel:DELete:ALL

Description	This command deletes all channels from channel list. The OSA module must not be in acquisition for this command to be accepted. This command is an event and has no associated *RST condition or query form.
Syntax	:CALCulate[1..n]:CHANnel:DELete:ALL
Parameter(s)	None
Example(s)	CALC:CHAN:DEL:ALL
See Also	CALCulate[1..n]:CHANnel:DELete

:CALCulate[1..n][:IMMediate]:AUTO

Description	This command indicates whether or not the system automatically make a recalculation of all measurements every time a call is made to sub-system "CALC". At *RST, this value is 0 (deactivated).
Syntax	:CALCulate[1..n][:IMMediate]:AUTO<wsp><State>
Parameter(s)	*State:* The program data syntax for <State> is defined as a <Boolean Program Data> element. The <State> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0. Selects the possible values for immediate automatic calculation.
Example(s)	CALC:AUTO ON
See Also	CALCulate[1..n]:IMMediate:AUTO?

:CALCulate[1..n][:IMMediate]:AUTO?

Description	This query returns the current state of whether or not the system automatically make a recalculation of all measurements every time a call is made to sub-system "CALC". At *RST, this value is 0 (deactivated).
Syntax	:CALCulate[1..n][:IMMediate]:AUTO?
Parameter(s)	None
Response Syntax	<State>
Response(s)	*State:* The response data syntax for <State> is defined as a <BINARY NUMERIC RESPONSE DATA> element. Display the immediate automatic calculation value ON/OFF.
Example(s)	CALC:IMMediate:AUTO?
See Also	CALCulate[1..n]:IMMediate:AUTO

:CALCulate[1..n]:PEAK[:DETEction]:LEVel

Description	This command sets level in relation to peak for WDM Analysis test. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is -45.00 dBm.
Syntax	:CALCulate[1..n]:PEAK[:DETEction]:LEVel <wsp><PeakDetectionLevel>
Parameter(s)	*PeakDetectionLevel:* The program data syntax for <PeakDetectionLevel> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Selects the Peak detection level. Range: [-99.99 … 10.00] dBm
Example(s)	CALC:PEAK:DETEction:LEVel -65.08
See Also	CALCulate[1..n]:PEAK:DETEction:LEVel?

:CALCulate[1..n]:PEAK[:DETEction]:LEVel?

Description	This query returns the level in relation to peak from WDM test. At *RST, this value is -45.00 dBm.
Syntax	:CALCulate[1..n]:PEAK[:DETEction]:LEVel?
Parameter(s)	None
Response Syntax	<PeakDetectionLevel>
Response(s)	*PeakDetectionLevel:* The response data syntax for <PeakDetectionLevel> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Display the peak detection level for global channel analysis parameter.
Example(s)	CALC:PEAK:LEVel?
See Also	CALCulate[1..n]:PEAK:DETEction:LEVel

:CALCulate[1..n]:POWer:OFFSet [:OUTPut]

Description	This command sets the Power Offset value. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 0.00 dB.
Syntax	:CALCulate[1..n]:POWer:OFFSet[:OUTPut]<wsp><Offset>
Parameter(s)	*Offset:* The program data syntax for <Offset> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Selects the Offset value. Range: [-99.99 … 99.99] dB
Example(s)	CALC:POWer:OFFSet -0.56
See Also	CALCulate[1..n]:POWer:OFFSet:OUTPut?

:CALCulate[1..n]:POWer:OFFSet[:OUTPut]?

Description	This querry returns the Offset value At *RST, this value is 0.00 dB.
Syntax	:CALCulate[1..n]:POWer:OFFSet[:OUTPut]?
Parameter(s)	None
Response Syntax	<Offset>
Response(s)	*Offset:* The response data syntax for <Offset> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Display the power offset for global channel analysis parameter.
Example(s)	CALC:POWer:OFFSet:OUTPut?
See Also	CALCulate[1..n]:POWer:OFFSet:OUTPut

:CALCulate[1..n]:RBW

Description	This command sets the RBW for OSNR. This value is used in calculation only if RBW Instrument is OFF (or 0). For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 0.100 nm.
Syntax	:CALCulate[1..n]:RBW<wsp><RbwForOsnr>
Parameter(s)	*RbwForOsnr:* The program data syntax for <RbwForOsnr> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Selects the RBW for OSNR. Range: [0.010 … 2.000] nm
Example(s)	CALC:RBW 0.1
See Also	CALCulate[1..n]:RBW? CALCulate[1..n]:RBW:INSTrument CALCulate[1..n]:RBW:INSTrument?

:CALCulate[1..n]:RBW?

Description	This query gets the value for RBW for OSNR. At *RST, this value is 0.100 nm.
Syntax	:CALCulate[1..n]:RBW?
Parameter(s)	None
Response Syntax	<RbwForOsnr>
Response(s)	*RbwForOsnr:* The response data syntax for <RbwForOsnr> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Display the RBW for OSNR for global channel analysis parameter.
Example(s)	CALC:RBW?
See Also	CALCulate[1..n]:RBW CALCulate[1..n]:RBW:INSTrument CALCulate[1..n]:RBW:INSTrument?

:CALCulate[1..n]:RBW:INSTrument

Description	This command sets whether the application uses the Instrument RBW or the user defined RBW value. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 1 (or ON).
Syntax	:CALCulate[1..n]:RBW:INSTrument<wsp><RbwInstrumentState>
Parameter(s)	*RbwInstrumentState:* The program data syntax for <RbwInstrumentState> is defined as a <Boolean Program Data> element. The <RbwInstrumentState> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0. The <RbwInstrumentState> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0.

:CALCulate[1..n]:RBW:INSTrument

The <RbwInstrumentState> parameter is a boolean value that determines the status of the default channel:
0- deactivates Instrument RBW (user defined RBW is used by application)
1- activates instrument RBW

Example(s)	CALC:RBW:INSTrument ON
See Also	CALCulate[1..n]:RBW CALCulate[1..n]:RBW? CALCulate[1..n]:RBW:INSTrument?

:CALCulate[1..n]:RBW:INSTrument?

Description	This query returns whether the application uses the instrument RBW or the user defined RBW value. At *RST, this value is 1 (or ON).
Syntax	:CALCulate[1..n]:RBW:INSTrument?
Parameter(s)	None
Response Syntax	<RbwInstrumentState>
Response(s)	*RbwInstrumentState:* The response data syntax for <RbwInstrumentState> is defined as a <BINARY NUMERIC RESPONSE DATA> element. Display the Instrument RBW state for OSNR state.
Example(s)	CALC:RBW:INSTrument?
See Also	CALCulate[1..n]:RBW:INSTrument CALCulate[1..n]:RBW CALCulate[1..n]:RBW?

:CALCulate[1..n]:WAVelength:OFFSet [:OUTPut]

Description	This command sets the wavelength offset value. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 0.000 nm.
Syntax	:CALCulate[1..n]:WAVelength:OFFSet[:OUTPut] <wsp><WavelegthOffset>
Parameter(s)	*WavelegthOffset:* The program data syntax for <WavelegthOffset> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element. Selects the wavelength Offset value. Range: [-99.999 … 99.999] nm
Example(s)	CALC:WAV:OFFSet:OUTPut 0.000
See Also	CALCulate[1..n]:WAVelength:OFFSet:OUTPut?

:CALCulate[1..n]:WAVelength:OFFSet[:OUTPut]?

Description	This query returns the wavelength offset value. At *RST, this value is 0.000 nm.
Syntax	:CALCulate[1..n]:WAVelength:OFFSet[:OUTPut]?
Parameter(s)	None
Response Syntax	<WavelengthOffset>
Response(s)	*WavelengthOffset:* The response data syntax for <WavelengthOffset> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Display the wavelength offset value.
Example(s)	CALC:WAV:OFFSet:OUTPut?
See Also	CALCulate[1..n]:WAVelength:OFFSet:OUTPut

:CALCulate[1..n]:WDM

Description	When used alone, this command triggers a WDM re-analysis using :CALCulate current values. Pre-condition: trace data must be present in memory (either from an acquisition or from loading a file). This command is an event and has no associated *RST condition or query form.
Syntax	:CALCulate[1..n]:WDM
Parameter(s)	None
Example(s)	CALC:WDM
See Also	CALCulate[1..n]:WDM:CHANnel:COUNt?

:CALCulate[1..n]:WDM:CHANnel?

Description

This query returns the information pertaining to a channel.
A trace must be present in memory for the command to be accepted.

*RST has no effect on the WDM measurement.

Syntax

:CALCulate[1..n]:WDM:CHANnel?

Parameter(s)

ChannelIndex:

The program data syntax for <ChannelIndex> is defined as a <DECIMAL NUMERIC PROGRAM DATA> element.

Selects the Index of channel.

The maximum value for the index varies with thenumber of channels on the results list.

Response Syntax

<ChannelResults>

Response(s)

ChannelResults:

The response data syntax for <ChannelResults> is defined as a <DEFINITE LENGTH ARBITRARY BLOCK RESPONSE DATA> element.

Channel results in A,B,C,D,E,F,G,H,I,J,K,L format where:

A = Channel name <STRING PROGRAM DATA>
B = Spectral position (always in m) <NR3 NUMERIC RESPONSE DATA>
C = Channel power (always dBm) <NR3 NUMERIC RESPONSE DATA>
D = Power type <STRING PROGRAM DATA>

:CALCulate[1..n]:WDM:CHANnel?

	E = Noise (always in dBm) <NR3 NUMERIC RESPONSE DATA> F = Noise type <STRING PROGRAM DATA> G = OSNR (always in dB) <NR3 NUMERIC RESPONSE DATA> H = Bandwidth at 3 dB (always in m) <NR3 NUMERIC RESPONSE DATA> I = Bandwidth at x dB (always in m) <NR3 NUMERIC RESPONSE DATA> J = Empty channel state <NR1 NUMERIC RESPONSE DATA> K = Power saturation state <NR1 NUMERIC RESPONSE DATA> L= Sufficient discrimination for OSNR calculation state <NR1 NUMERIC RESPONSE DATA> For additional information on K and L results, see *Managing Results* on page 277.
Example(s)	MMEM:LOAD:TRAC TRCMAIN,"Trace.osawdm" CALC:WDM:CHAN? 1
See Also	CALCulate[1..n]:WDM

:CALCulate[1..n]:WDM:CHANnel:COUNt?

Description	This query returns the number of channel results from the list. *RST has no effect on the WDM measurement.
Syntax	:CALCulate[1..n]:WDM:CHANnel:COUNt?
Parameter(s)	None
Response Syntax	<ChannelCount>
Response(s)	*ChannelCount:* The response data syntax for <ChannelCount> is defined as a <NR1 NUMERIC RESPONSE DATA> element. Number of channel results on the list.
Example(s)	CALC:WDM:CHAN:COUNt?
See Also	CALCulate[1..n]:WDM

:CALCulate[1..n]:WDM:OSNR:AVERage?

Description	This query returns the average SNR value. *RST has no effect on the WDM measurement.
Syntax	:CALCulate[1..n]:WDM:OSNR:AVERage?
Parameter(s)	None
Response Syntax	<OsnrAverage>
Response(s)	*OsnrAverage:* The response data syntax for <OsnrAverage> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Average optical signal-to-noise ratio from all channels (in dB).
Example(s)	INIT:IMM CALC:WDM:OSNR:AVERage?
See Also	CALCulate[1..n]:WDM:OSNR:FLATness?

:CALCulate[1..n]:WDM:OSNR:FLATness?

Description	This query returns the flatness SNR value. *RST has no effect on the WDM measurement.
Syntax	:CALCulate[1..n]:WDM:OSNR:FLATness?
Parameter(s)	None
Response Syntax	<OsnrFlatness>
Response(s)	*OsnrFlatness:* The response data syntax for <OsnrFlatness> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Flatness optical signal-to-noise ratio from all channels (in dB).
Example(s)	INIT:IMM CALC:WDM:OSNR:FLATness?
See Also	CALCulate[1..n]:WDM:OSNR:AVERage?

:CALCulate[1..n]:WDM:SIGPower:AVERage?

Description	This query returns the average signal power for a channel. *RST has no effect on the WDM measurement.
Syntax	:CALCulate[1..n]:WDM:SIGPower:AVERage?
Parameter(s)	None
Response Syntax	<PowerAverage>
Response(s)	*PowerAverage:* The response data syntax for <PowerAverage> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Average signal power from all channels (in dBm).
Example(s)	INIT:IMM CALC:WDM:SIGPower:AVERage?
See Also	CALCulate[1..n]:WDM:SIGPower:FLATness? CALCulate[1..n]:WDM:SIGPower:INTegrated?

:CALCulate[1..n]:WDM:SIGPower:FLATness?

Description	This query returns the flatness signal power for a channel. *RST has no effect on the WDM measurement.
Syntax	:CALCulate[1..n]:WDM:SIGPower:FLATness?
Parameter(s)	None
Response Syntax	<PowerFlatness>
Response(s)	*PowerFlatness:* The response data syntax for <PowerFlatness> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Flatness signal power from all channels (in dB).
Example(s)	INIT:IMM CALC:WDM:SIGPower:FLATness?
See Also	CALCulate[1..n]:WDM:SIGPower:AVERage? CALCulate[1..n]:WDM:SIGPower:INTegrated?

:CALCulate[1..n]:WDM:SIGPower:INTegrated?

Description	This query returns the integrated signal power for a channel. *RST has no effect on the WDM measurement.
Syntax	:CALCulate[1..n]:WDM:SIGPower:INTegrated?
Parameter(s)	None
Response Syntax	<PowerIntegrated>
Response(s)	*PowerIntegrated:* The response data syntax for <PowerIntegrated> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Integrated signal power from all channels (in dBm).
Example(s)	INIT:IMM CALC:WDM:SIGPower:INTegrated?
See Also	CALCulate[1..n]:WDM:SIGPower:FLATness? CALCulate[1..n]:WDM:SIGPower:AVERage?

:CALibration:ZERO[:IMMediate]

Description	This command performs a nulling operation. This command is an event and has no associated *RST condition or query form.
Syntax	:CALibration:ZERO[:IMMediate]
Parameter(s)	None
Example(s)	CAL:ZERO
See Also	INITiate[1..n]:IMMediate

:INITiate[1..n][:IMMediate]

Description	This command completes one full trigger cycle, returning to IDLE on completion. The operation in progress is instrument-dependent. This command is an event and has no associated *RST condition or query form. However, the equivalent of the ABORt command is performed on any acquisition in progress.
Syntax	:INITiate[1..n][:IMMediate]
Parameter(s)	None
Example(s)	SENS:AVER:TYPE AVERAGING SENS:AVER:COUN 8 INIT
See Also	ABORt[1..n] STATus?

:MMEMory[1..n]:LOAD:CONFiguratio

Description	This command loads a configuration file. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. This command is an event and has no associated *RST condition or query form. However, most settings affected by this command are set to default values at *RST.
Syntax	:MMEMory[1..n]:LOAD:CONFiguratio<wsp><Filename>
Parameter(s)	*Filename:* The program data syntax for <Filename> is defined as a <STRING PROGRAM DATA> element. Selects the configuration file to load. The <Filename> parameter can either be only the filename or the name and its path. If no path is specified, the default path is used. The default path name depend on location of driver libraries.
Example(s)	MMEM:LOAD:CONF "config001.osawdmsetup"
See Also	MMEMory[1..n]:STORe:CONFiguration

:MMEMory[1..n]:LOAD:OVERwrite	
Description	This command selects, whether local data trace should be overwritten or not when loading a trace file. Attempting to load a file while the value is set to OFF will generate an error if local trace already exists in destination. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress for. At *RST, this value is OFF.
Syntax	:MMEMory[1..n]:LOAD:OVERwrite<wsp><Overwrite>
Parameter(s)	*Overwrite:* The program data syntax for <Overwrite> is defined as a <Boolean Program Data> element. The <Overwrite> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0. Selects the local data trace of a trace file. Trace overwrite status: ON = overwrite OFF = Do not overwrite. An error occurs when loading a file if local data already exists in destination.
Example(s)	MMEM:LOAD:OVER ON
See Also	MMEMory[1..n]:LOAD:OVERwrite? MMEMory[1..n]:LOAD:TRACe

:MMEMory[1..n]:LOAD:OVERwrite?

Description	This query lets you know whether local data will be overwritten or not when loading a trace file. At *RST, this value is OFF.
Syntax	:MMEMory[1..n]:LOAD:OVERwrite?
Parameter(s)	None
Response Syntax	<OverwriteState>
Response(s)	*OverwriteState:* The response data syntax for <OverwriteState> is defined as a <BINARY NUMERIC RESPONSE DATA> element. Local trace data overwrite state OFF/ON (0/1).
Example(s)	MMEM:LOAD:OVER?
See Also	MMEMory[1..n]:LOAD:OVERwrite

:MMEMory[1..n]:LOAD:TRACe

Description	This command loads a trace. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. *RST has no effect on traces in memory.
Syntax	:MMEMory[1..n]:LOAD:TRACe<wsp> |TRCMAIN|,<FileName>
Parameter(s)	➤ *TracePosition:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> element for this parameter is TRCMAIN.

:MMEMory[1..n]:LOAD:TRACe

	Selects Trace index of destination. TRCMAIN, selects Main Trace as index of destination. ➤ *FileName:* The program data syntax for <FileName> is defined as a <STRING PROGRAM DATA> element. The <Filename> parameter can either be only the filename or the name and its path. If no path is specified, the default path is used. The default path name depends on location of driver libraries.
Example(s)	MMEM:LOAD:TRAC TRCMAIN,"Trace.osw"
See Also	MMEMory[1..n]:LOAD:OVERwrite MMEMory[1..n]:STORe:TRACe

:MMEMory[1..n]:STORe:CONFiguratio

Description	This command saves a configuration file. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. This command is an event and has no associated *RST condition or query form.
Syntax	:MMEMory[1..n]:STORe:CONFiguratio<wsp><F ilename>
Parameter(s)	*Filename:* The program data syntax for <Filename> is defined as a <STRING PROGRAM DATA> element. Saves a configuration file name. The <Filename> parameter can either be only the filename or the name and its path. If no path is specified, the default path is used. The default path name depends on location of driver libraries.
Example(s)	MMEM:STOR:CONF "config001.cfg"
See Also	MMEMory[1..n]:LOAD:CONFiguration

:MMEMory[1..n]:STORe:OVERwrite

Description	This command sets, whether an existing file will be overwritten when the MMEMory:STORe:OVERwrite command is used. Attempting to save a file while the value is set to OFF will generate an error if file is already present. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is OFF.
Syntax	:MMEMory[1..n]:STORe:OVERwrite<wsp><Overwrite>
Parameter(s)	*Overwrite:* The program data syntax for <Overwrite> is defined as a <Boolean Program Data> element. The <Overwrite> special forms ON and OFF are accepted on input for increased readability. ON corresponds to 1 and OFF corresponds to 0. Overwrite file if existing.
Example(s)	MMEM:STOR:OVER ON
See Also	MMEMory[1..n]:STORe::OVERwrite?

:MMEMory[1..n]:STORe:OVERwrite?

Description	This query lets you know whether existing files already present will be overwritten when the MMEMory:STORe:OVERwrite command is used. At *RST, this value is OFF.
Syntax	:MMEMory[1..n]:STORe:OVERwrite?
Parameter(s)	None
Response Syntax	<OverwriteState>
Response(s)	*OverwriteState:* The response data syntax for <OverwriteState> is defined as a <BINARY NUMERIC RESPONSE DATA> element. Overwrite file status ON/OFF(1/0).
Example(s)	MMEM:STOR:OVER?
See Also	MMEMory[1..n]:STORe:OVERwrite MMEMory[1..n]:STORe:OVERwrite

:MMEMory[1..n]:STORe:TRACe

Description	This command saves trace to a file. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. This command is an event and has no associated *RST condition or query form.
Syntax	:MMEMory[1..n]:STORe:TRACe<wsp>TRCMAIN, <Filename>
Parameter(s)	➤ *TracePosition:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> element for this parameter is TRCMAIN. Selects the Trace index. ➤ *Filename:* The program data syntax for <Filename> is defined as a <STRING PROGRAM DATA> element. The <Filename> parameter can either be only the filename or the name and its path. If no path is specified, the default path is used. The default path name depends on location of driver libraries.
Example(s)	MMEM:STOR:TRAC TRCM,"Trace.osw"
See Also	MMEMory[1..n]:LOAD:TRACe MMEMory[1..n]:STORe:OVERwrite

:SENSe[1..n]:AVERage:COUNt

Description	Sets the number of scans that will be averaged to produce final trace. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 8.
Syntax	:SENSe[1..n]:AVERage:COUNt<wsp><AverageCount>
Parameter(s)	*AverageCount:* The program data syntax for <AverageCount> is defined as a <numeric_value> element. Selects the number of scans that will be averaged to produce final trace. Range: [2 ... 9999]
Example(s)	SENS:AVER:TYPE AVERAGING SENS:AVER:COUN 12 INITiate:IMMediate
See Also	SENSe[1..n]:AVERage:COUNt? SENSe[1..n]:AVERage:TYPE

:SENSe[1..n]:AVERage:COUNt?

Description	This query returns number of scans averaged to produce final trace.Returns 1 when average type is SINGLE. At *RST, this value is 8.
Syntax	:SENSe[1..n]:AVERage:COUNt?
Parameter(s)	None
Response Syntax	<AverageCount>
Response(s)	*AverageCount:* The response data syntax for <AverageCount> is defined as a <NR1 NUMERIC RESPONSE DATA> element. Number of scans used to average for final scan.
Example(s)	SENS:AVER:COUNt?
See Also	SENSe[1..n]:AVERage:COUNt SENSe[1..n]:AVERage:TYPE

:SENSe[1..n]:AVERage:INBand:COUNt

Description	This command sets the number of In-Band scans that will be averaged to produce final trace. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is 64.
Syntax	:SENSe[1..n]:AVERage:INBand:COUNt<wsp> <InbandCount>
Parameter(s)	*InbandCount:* The program data syntax for <InbandCount> is defined as a <numeric_value> element. Selects the Number of scans used to inband for the final scan. Range: [4 ... 9999]
Example(s)	SENS:AVER:TYPE AVERAGING SENS:AVER:INBand:COUN 4 SENS:AVER:COUN 12 INITiate:IMMediate
See Also	SENSe[1..n]:AVERage:INBand:COUNt? SENSe[1..n]:AVERage:TYPE

:SENSe[1..n]:AVERage:INBand:COUNt?

Description	This query returns number of scans Inband to produce final trace. At *RST, this value is 64.
Syntax	:SENSe[1..n]:AVERage:INBand:COUNt?
Parameter(s)	None
Response Syntax	<InbandCount>
Response(s)	*InbandCount:* The response data syntax for <InbandCount> is defined as a <NR1 NUMERIC RESPONSE DATA> element. Number of In-Band scans used to average in final scan.
Example(s)	SENS:AVER:INBand:COUN?
See Also	SENSe[1..n]:AVERage:INBand:COUNt SENSe[1..n]:AVERage:TYPE

:SENSe[1..n]:AVERage:TYPE

Description	This command sets the acquisition type. Possible values: SINGLE, AVERAGING or INBAND. At *RST, this value is SINGLE.
Syntax	:SENSe[1..n]:AVERage:TYPE<wsp>SINGLE|AVERAGING|INBAND
Parameter(s)	*Type:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> elements for this parameter are: SINGLE|AVERAGING|INBAND. Selects the Acquisition type.
Example(s)	SENS:AVER:TYPE SINGLE
Notes	For the OSA, Acquisition type is SINGLE,AVERAGING or INBAND
See Also	INITiate[:IMMediate] SENSe[1..n]:AVERage:TYPE?

:SENSe[1..n]:AVERage:TYPE?

Description	This query returns the acquisition type. Returns: SINGLE, AVERAGING or INBAND. At *RST, this value is SINGLE.
Syntax	:SENSe[1..n]:AVERage:TYPE?
Parameter(s)	None
Response Syntax	<Type>
Response(s)	*Type:* The response data syntax for <Type> is defined as a <CHARACTER RESPONSE DATA> element. Display the Acquisition type
Example(s)	SENS:AVER:TYPE?
Notes	For the OSA, Acquisition type is SINGLE,AVERAGING or INBAND
See Also	INITiate[:IMMediate] SENSe[1..n]:AVERage:TYPE

:SENSe[1..n]:WAVelength[:RANGe]:STARt

Description	This command sets the start wavelength for acquisition range. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is set to 1530.0 nm (C-band lower limit).
Syntax	:SENSe[1..n]:WAVelength[:RANGe]:STARt<wsp><Start[<wsp>NM]>|MAXimum|MINimum
Parameter(s)	*Start:* The program data syntax for <Start> is defined as a <numeric_value> element followed by an optional <SUFFIX PROGRAM DATA> element. The allowed <SUFFIX PROGRAM DATA> element is NM. The <Start> special forms MINimum and MAXimum are accepted on input. MINimum allows to set the instrument to the smallest supported value. MAXimum allows to set the instrument to the greatest supported value. Minimum wavelength for acquisition range. Range is device-dependent.

:SENSe[1..n]:WAVelength[:RANGe]:STARt

Example(s)	SENS:WAV:RANG:STAR 1520 NM
Notes	SENS:WAV:RANG:STOP value will be forced to 5.0 nm larger than SENS:WAV:RANG:STARt.
See Also	SENSe[1..n]:WAVelength:RANGe:STARt? SENSe[1..n]:WAVelength:RANGe:STOP

:SENSe[1..n]:WAVelength[:RANGe]:STARt?

Description	This query returns the start wavelength for acquisition range. At *RST, this value is set to 1530.0 nm (C-band lower limit).
Syntax	:SENSe[1..n]:WAVelength[:RANGe]:STARt?
Parameter(s)	None
Response Syntax	<Start>
Response(s)	*Start:* The response data syntax for <Start> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Minimum wavelength for acquisition range (in m or Hz).
Example(s)	SENS:WAV:RANG:STAR?
See Also	SENSe[1..n]:WAVelength:RANGe:STAR SENSe[1..n]:WAVelength:RANGe:STOP?

:SENSe[1..n]:WAVelength[:RANGe]:STOP

Description	This command sets the stop wavelength for acquisition range. For this command to be accepted, the OSA module must be in Ready state with no acquisition in progress. At *RST, this value is set to 1565.0 nm (C-band upper limit).
Syntax	:SENSe[1..n]:WAVelength[:RANGe]:STOP<wsp><Stop[<wsp>NM]>|MAXimum|MINimum
Parameter(s)	*Stop:* The program data syntax for <Stop> is defined as a <numeric_value> element followed by an optional <SUFFIX PROGRAM DATA> element. The allowed <SUFFIX PROGRAM DATA> element is NM. The <Stop> special forms MINimum and MAXimum are accepted on input. MINimum allows to set the instrument to the smallest supported value. MAXimum allows to set the instrument to the greatest supported value. Maximum wavelength for acquisition range. Range is device-dependent.

:SENSe[1..n]:WAVelength[:RANGe]:STOP

Example(s)	SENS:WAV:RANG:STOP 1570 NM
Notes	SENS:WAV:RANG:STARt value will be forced to 5.0 nm smaller than SENS:WAV:RANG:STOP.
See Also	SENSe[1..n]:WAVelength:RANGe:STOP? SENSe[1..n]:WAVelength:RANGe:STARt

:SENSe[1..n]:WAVelength[:RANGe]:STOP?

Description	This query returns the start wavelength for acquisition range. At *RST, this value is set to 1565.0 nm (C-band upper limit).
Syntax	:SENSe[1..n]:WAVelength[:RANGe]:STOP?
Parameter(s)	None
Response Syntax	<Stop>
Response(s)	*Stop:* The response data syntax for <Stop> is defined as a <NR3 NUMERIC RESPONSE DATA> element. Maximum wavelength for acquisition range (in m or Hz).
Example(s)	SENS:WAV:RANG:STOP?
See Also	SENSe[1..n]:WAVelength:RANGe:STOP SENSe[1..n]:WAVelength:RANGe:STARt?

:SNUMber?

Description	This query returns a value indicating the serial number of the module. *RST has no effect on Serial number.
Syntax	:SNUMber?
Parameter(s)	None
Response Syntax	<SerialNumber>
Response(s)	*SerialNumber:* The response data syntax for <SerialNumber> is defined as a <STRING RESPONSE DATA> element. The <SerialNumber> response represents a string containing the serial number of the module.
Example(s)	SNUM?
See Also	STATus?

:STATus?

Description	This query returns a value indicating the status of the module (READY, BUSY, etc.). This command is an event and has no associated *RST condition.
Syntax	:STATus?
Parameter(s)	None
Response Syntax	<Status>
Response(s)	*Status:* The response data syntax for <Status> is defined as a <CHARACTER RESPONSE DATA> element. The <Status> response represents the module state, where: UNINITIALIZED, means the module is not initialized. INITINPROGRESS, means the module initialization is in progress, READY, means the module is ready, BUSY, means the module is busy, DISCONNECTED, means the module is disconnected,

:STATus?

	DEFECTIVE, means the module is defective and UNCONFIGURED, means the module is not configured.
Example(s)	STATus?
See Also	INITiate[:IMMediate] SNUMber?

:TRACe[1..n][:DATA]?

Description	This query returns the points of a trace. *RST has no effect on traces in memory.
Syntax	:TRACe[1..n][:DATA]?<wsp>TRCMAIN
Parameter(s)	*TracePosition:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> element for this parameter is TRCMAIN. Selects the Trace Index.
Response Syntax	<TraceData>
Response(s)	*TraceData:* The response data syntax for <TraceData> is defined as a <DEFINITE LENGTH ARBITRARY BLOCK RESPONSE DATA> element. List containing trace data (power, power, power format). Each power value represents a point in the trace and is always returned in dBm as <NR3 NUMERIC RESPONSE DATA> type.
Example(s)	TRACe:DATA? TRCM
See Also	TRACe[1..n][:DATA]:PREamble? TRACe[1..n]:INFormation? TRACe[1..n]:POINts?

:TRACe[1..n][:DATA]:PREamble?

Description	This query returns a trace's header. *RST has no effect on traces in memory.
Syntax	:TRACe[1..n][:DATA]:PREamble?<wsp>TRCMAI N
Parameter(s)	*TracePosition:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> element for this parameter is TRCMAIN. Selects the Trace Index.
Response Syntax	<TraceHeader>
Response(s)	*TraceHeader:* The response data syntax for <TraceHeader> is defined as a <DEFINITE LENGTH ARBITRARY BLOCK RESPONSE DATA> element. List containing trace data preamble A,B,C,D,E format where:

:TRACe[1..n][:DATA]:PREamble?

	A = Acquisition type <STRING RESPONSE DATA> B = Spectral range start (always in m) <NR3 NUMERIC RESPONSE DATA> C = Spectral range stop (always in m) <NR3 NUMERIC RESPONSE DATA> D = Number of scans <NR1 NUMERIC RESPONSE DATA> E = Bandwidth Resolution (always in m) <NR3 NUMERIC RESPONSE DATA>
Example(s)	TRAC:PRE? TRCMAIN
See Also	TRACe[1..n][:DATA]? TRACe[1..n]:INFormation? TRACe[1..n]:POINts?

:TRACe[1..n]:INFormation?

Description	This query returns all available information on a trace. *RST has no effect on traces in memory.
Syntax	:TRACe[1..n]:INFormation?<wsp>TRCMAIN
Parameter(s)	*TracePosition:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> element for this parameter is TRCMAIN. Selects the Trace Index.
Response Syntax	<TraceInfo>
Response(s)	*TraceInfo:* The response data syntax for <TraceInfo> is defined as a <DEFINITE LENGTH ARBITRARY BLOCK RESPONSE DATA> element. Selected trace information A,B,C,D,E,F,G format, where:

:TRACe[1..n]:INFormation?

	A = Acquisition start date and time <STRING RESPONSE DATA> B = Acquisition end date and time <STRING RESPONSE DATA> C = Calibration date <STRING RESPONSE DATA> D = User or factory calibration <STRING RESPONSE DATA> E = Model name <STRING RESPONSE DATA> F = Serial number <STRING RESPONSE DATA> G = Software version <STRING RESPONSE DATA>
Example(s)	TRAC:INF? TRCM
See Also	TRACe[1..n][:DATA]? TRACe[1..n][:DATA]:PREamble? TRACe[1..n]:POINts?

:TRACe[1..n]:POINts?

Description	This query returns the number of points on a trace. *RST has no effect on traces in memory.
Syntax	:TRACe[1..n]:POINts?<wsp>TRCMAIN
Parameter(s)	*TracePosition:* The program data syntax for the first parameter is defined as a <CHARACTER PROGRAM DATA> element. The allowed <CHARACTER PROGRAM DATA> element for this parameter is TRCMAIN. Selects the Trace Index.
Response Syntax	<PointsCount>
Response(s)	*PointsCount:* The response data syntax for <PointsCount> is defined as a <NR1 NUMERIC RESPONSE DATA> element. Number of points on the trace.
Example(s)	TRAC:POINts? TRCMAIN
See Also	TRACe[1..n][:DATA]? TRACe[1..n][:DATA]:PREamble? TRACe[1..n]:INFormation?

:UNIT[1..n]:SPECtrum

Description	This command sets the spectrum units. At *RST, this value is m.
Syntax	:UNIT[1..n]:SPECtrum<wsp>M|HZ
Parameter(s)	*Unit:* The program data syntax for the first parameter is defined as a <SUFFIX PROGRAM DATA> element. The allowed <SUFFIX PROGRAM DATA> elements are: M|HZ. Selects the Spectral unit.
Example(s)	UNIT:SPEC M
See Also	UNIT[1..n]:SPECtrum?

:UNIT[1..n]:SPECtrum?

Description	This query returns the current spectrum unit. At *RST, this value is m.
Syntax	:UNIT[1..n]:SPECtrum?
Parameter(s)	None
Response Syntax	<SpectralUnit>
Response(s)	*SpectralUnit:* The response data syntax for <SpectralUnit> is defined as a <CHARACTER RESPONSE DATA> element. Current spectral unit.
Example(s)	UNIT:SPECtrum?
See Also	UNIT[1..n]:SPECtrum

C Formulas Used with Your Optical Spectrum Analyzer

The following formulas are used in the various tests available with your OSA module.

EDFA Noise Figure Calculation

According to EDFA theory, this measurement is obtained using the following equation:

$$\text{EDFA noise figure} = \frac{P_{ASE} - GP_{SSE}}{Gh\nu B} + \frac{1}{G}$$

Where

P_{ASE} is the power of the spontaneous emission amplified by the EDFA,

P_{SSE} is the power of the spontaneous emission of the source,

G is the gain at this channel's wavelength,

h is Plank's constant (6,6256 x 10^{-34} $J \cdot s$),

ν is the frequency of the channel, and

B is the noise equivalent bandwidth, as calibrated at this channel's wavelength.

Central Wavelength Calculation (Spectral Transmittance)

The central wavelength is calculated using the following equation:

$$a = \frac{\lambda_R + \lambda_L}{2}$$

Where

a is the central wavelength,

λ_R is the wavelength on the right at which the power is 3 dB below the power at the nominal wavelength, and

λ_L is the wavelength on the left at which the power is 3 dB below the power at the nominal wavelength.

Bandwidth Calculation (Spectral Transmittance)

Bandwidth is calculated using the following equation:

$$b = 2*\mathrm{Min}\{(\lambda_N - \lambda_{XdBLeft}), (\lambda_{XdBRight} - \lambda_N)\}$$

Where

b is the bandwidth at X dB,

λ_N is the nominal wavelength,

$\lambda_{XdBLeft}$ is the wavelength on the left at which the power is X dB below the power at the nominal wavelength.

$\lambda_{XdbRight}$ is the wavelength on the right at which the power is X dB below the power at the nominal wavelength.

Index

NOTICE

通告

CHINESE REGULATION ON RESTRICTION OF HAZARDOUS SUBSTANCES

中国关于危害物质限制的规定

NAMES AND CONTENTS OF THE TOXIC OR HAZARDOUS SUBSTANCES OR ELEMENTS CONTAINED IN THIS EXFO PRODUCT

包含在本 EXFO 产品中的有毒有害物质或元素的名称和含量

O	Indicates that this toxic or hazardous substance contained in all of the homogeneous materials for this part is below the limit requirement in SJ/T11363-2006 表示该有毒有害物质在该部件所有均质材料中的含量均在 SJ/T11363-2006 标准规定的限量要求以下。
X	Indicates that this toxic or hazardous substance contained in at least one of the homogeneous materials used for this part is above the limit requirement in SJ/T11363-2006 表示该有毒有害物质至少在该部件的某一均质材料中的含量超出 SJ/T11363-2006 标准规定的限量要求。

Part Name 部件名称	Toxic or hazardous Substances and Elements 有毒有害物质和元素					
	Lead 铅 (Pb)	Mercury 汞 (Hg)	Cadmium 隔 (Cd)	Hexavalent Chromium 六价铬 (Cr VI)	Polybrominated biphenyls 多溴联苯 (PBB)	Polybrominated diphenyl ethers 多溴二苯醚 (PBDE)
Enclosure 外壳	O	O	O	O	O	O
Electronic and electrical sub-assembly 电子和电子组件	X	O	X	O	X	X
Optical sub-assembly[a] 光学组件[a]	X	O	O	O	O	O
Mechanical sub-assembly[a] 机械组件[a]	O	O	O	O	O	O

a. If applicable.
如果适用。

MARKING REQUIREMENTS
标注要求

Product 产品	Environmental protection use period (years) 环境保护使用期限 (年)	Logo 标志
This Exfo product 本 EXFO 产品	10	
Battery[a] 电池 [a]	5	

a. If applicable.
如果适用。

P/N: 1059976

www.EXFO.com · info@exfo.com

CORPORATE HEADQUARTERS	400 Godin Avenue	Quebec (Quebec) G1M 2K2 CANADA Tel.: 1 418 683-0211 · Fax: 1 418 683-2170
EXFO AMERICA	3701 Plano Parkway, Suite 160	Plano TX, 75075 USA Tel.: 1 972 907-1505 · Fax: 1 972 836-0164
EXFO EUROPE	Omega Enterprise Park, Electron Way	Chandlers Ford, Hampshire S053 4SE ENGLAND Tel.: +44 2380 246810 · Fax: +44 2380 246801
EXFO ASIA-PACIFIC	151 Chin Swee Road #03-29, Manhattan House	SINGAPORE 169876 Tel.: +65 6333 8241 · Fax: +65 6333 8242
EXFO CHINA	Room 2711, Trade Center, No. 4028 Jintian Road, Futian District	Shenzhen 518035 P. R. CHINA Tel.: +86 (755) 8203 2300 · Fax: +86 (755) 8203 2306
	Beijing Global Trade Center, Tower C, Room 1207, 36 North Third Ring Road East, Dongcheng District	Beijing 100013 P. R. CHINA Tel.: +86 (10) 5825 7755 · Fax: +86 (10) 5825 7722
EXFO SERVICE ASSURANCE	270 Billerica Road	Chelmsford MA, 01824 USA Tel.: 1 978 367-5600 · Fax: 1 978 367-5700
TOLL-FREE	(USA and Canada)	1 800 663-3936

Printed in Canada (2010-11)

ISO 9001

EXPERTISE REACHING OUT