# JDSU Optical Network Management System (ONMS)

**System designed to supervise optical transport, V5.10**

User Manual

**JDSU**

# JDSU Optical Network Management System (ONMS)

**System designed to supervise optical transport, V5.10**

User Manual

**◇ JDSU**

*Manual*    This guide is a product of JDSU's Technical Information Development Department. This manual gives you the main information to install, start and use the ONMS.

*WEEE Directive Compliance*    JDSU has established processes in compliance with the Waste Electrical and Electronic Equipment (WEEE) Directive, 2002/96/EC.

This product should not be disposed of as unsorted municipal waste and should be collected separately and disposed of according to your national regulations. In the European Union, all equipment purchased from JDSU after 2005-08-13 can be returned for disposal at the end of its useful life. JDSU will ensure that all waste equipment returned is

reused, recycled, or disposed of in an environmentally friendly manner, and in compliance with all applicable national and international waste legislation.

It is the responsibility of the equipment owner to return the equipment to JDSU for appropriate disposal. If the equipment was imported by a reseller whose name or logo is marked on the equipment, then the owner should return the equipment directly to the reseller.

Instructions for returning waste equipment to JDSU can be found in the Environmental section of JDSU's web site at www.jdsu.com. If you have questions concerning disposal of your equipment, contact JDSU's WEEE Program Management team at WEEE.EMEA@jdsu.com.

# Table of contents

Table of contents

# About this guide

The ONMS of JDSU is a system designed to supervise optical transport..

The topics discussed in this chapter are as follows:

– "Purpose and scope" on page xviii
– "Assumptions" on page xviii
– "Technical assistance" on page xviii
– "Recycling Information" on page xix
– "Conventions" on page xix

# Purpose and scope

The purpose of this guide is to help you successfully use the ONMS features and capabilities. This guide includes task-based instructions that describe how to install, configure, use, and troubleshoot the ONMS. Additionally, this guide provides a complete description of JDSU's warranty, services, and repair information, including terms and conditions of the licensing agreement.

# Assumptions

This guide is intended for novice, intermediate, and experienced users who want to use the ONMS effectively and efficiently. We are assuming that you are familiar with basic telecommunication concepts and terminology.

# Technical assistance

If you need assistance or have questions related to the use of this product, call or e-mail JDSU's Technical Assistance Center for customer support.

**Table 1**      Technical assistance centers

| Region | Phone Number | |
| --- | --- | --- |
| Americas Telecom Products | 866 228 3762 World Wide: 301 353 1550 | tac@jdsu.com |
| Europe, Africa, and Mid-East | +49 (0) 7121 86 1345 (Europe) | hotline.europe@jdsu.com |
| | +33 (0) 1 30 81 50 60 (JDSU France) | support.france@jdsu.com |
| Asia and the Pacific | +852 2892 0990 (Hong Kong) | |
| Southeast Asia, Australia, and New Zealand | +86 10 6833 7477 (Beijing-China) | |

**Table 1** Technical assistance centers (Continued)

| Region | Phone Number | |
|--------|-------------|---|
| All others | 866 228 3762 | tac@jdsu.com |

During off-hours, you can request assistance by doing one of the following:

– leave a voice mail message at the Technical Assistance number in your region

– e-mail North American Technical Assistance Center, tac@jdsu.com, or European Technical Assistance Center, support.uk@jdsu.com

– submit your question using our online Technical Assistance Request form at www.jdsu.com.

# Recycling Information

JDSU recommends that customers dispose of their instruments and peripherals in an environnmentally sound manner. Potential methods include reuse of parts or whole products and recycling of products components, and/or materials.



**Waste Electrical and electronic Equipment (WEEE) Directive**

In the European Union, this label indicates that this product should not be disposed of with household waste. Il should be deposited at an appropriate facility to enable recovery and recycling.

# Conventions

This guide uses naming conventions and symbols, as described in the following tables.

**Table 2** Typographical conventions

| Description | Example |
|-------------|---------|
| User interface actions appear in this **typeface**. | On the Status bar, click **Start**. |

**Table 2**    Typographical conventions (Continued)

| Description | Example |
|---|---|
| Buttons or switches that you press on a unit appear in this **TYPEFACE**. | Press the **ON** switch. |
| Code and output messages appear in this typeface. | `All results okay` |
| Text you must type exactly as shown appears in this **type-face**. | Type: `a:\set.exe` in the dialog box |
| Variables appear in this ***type-face***. | Type the new ***hostname***. |
| Book references appear in this ***typeface***. | Refer to ***Newton's Telecom Dictionary*** |
| A vertical bar \| means "or": only one option can appear in a single command. | `platform [a\|b\|e]` |
| Square brackets [ ] indicate an optional argument. | `login [platform name]` |
| Slanted brackets < > group required arguments. | `<password>` |

**Table 3**    Keyboard and menu conventions

| Description | Example |
|---|---|
| A plus sign + indicates simultaneous keystrokes. | Press **Ctrl+s** |
| A comma indicates consecutive key strokes. | Press **Alt+f,s** |
| A slanted bracket indicates choosing a submenu from menu. | On the menu bar, click **Start > Program Files**. |

**Table 4**    Symbol conventions

| | |
|---|---|
| ⚠ | **This symbol represents a general hazard.** |

| | |
|---|---|
| ⚡ | **This symbol represents a risk of electrical shock.** |

| | |
|---|---|
| ▬ | **NOTE**<br>This symbol represents a Note indicating related information or tip. |

| | |
|---|---|
| 🗑 | This symbol, located on the equipment or its packaging indicates that the equipment must not be disposed of in a land-fill site or as municipal waste, and should be disposed of according to your national regulations. |
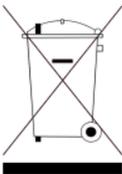
**Table 5**    Safety definitions

| | |
|---|---|
| ⚠ | **WARNING**<br>Indicates a potentially hazardous situation which, if not avoided, could result in death or serious injury. |

| | |
|---|---|
| ⚠ | **CAUTION**<br>Indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate injury. |

# Description

**1**

The ONMS system is designed to supervise optical transport.

Topics discussed in this chapter are as follows:

# Main characteristics of the system

This system ensures the monitoring of the physical and optical layers of the network using autonomous remote Optical Test Units (OTU-95000) placed at the strategic points of the network. They test the network and return alarm reports.

The OTUs can be equipped with two interchangeable optical modules compatible with the JDSU portable optical analyzers MTS 5000$^e$ :

– OTDR multimode or singlemode modules for the OTDR analysis

– WDM modules for optical spectra analysis (maintenance in the C band up to 12,5 GHz).

Result measurements can be easily analyzed via the Web.

Alarm data and cable documentation can be easily transferred to other systems.

# Main benefits of the ONMS system

– Increased network availability due to the continuous monitoring of the network and the indication the problems to operators as soon as they appear.

– The reduction of the average repair times:

The faults on the optical cables are more rapidly localized and identified by the system compared to repair times measured in hours when using traditional portable instruments. The ONMS localizes the detected fault.

– The creation of a predictive maintenance policy:

The system makes systematic and repeated acquisitions of measurement results from the optical elements. The analysis of the evolution of these results provides the operator with a powerful tool for anticipating faults.

– The creation of computerized documentation of the optical network:

The graphical interface of the system associated with a database facilitates computerization of the network documentation. The comparison of this documentation with systematic measurements guarantees the authenticity of the stored data.

– The standardization and aid in network construction operations:

During construction of the network, the association of measurements with the computerized documentation systemizes the creation of acceptance documents.

– The improvement in the efficiency of maintenance teams:

The tests can be carried out from a central point by a single user, consequently reducing the movements of the maintenance personnel. One qualified person can therefore manage the maintenance of a vast zone.

– The reduction in the portable maintenance equipment base:

Portable instruments are no longer required for the parts of the network supervised by the ONMS. All problems inherent to their use (availability, service ability, instrument configuration, etc.) are therefore removed.

# Surveillance Methods

2

Topics discussed in this chapter are as follows:

# Surveillance Methods

### Dark fiber method

A free fiber (or dark fiber) in the cable is used to provide a picture of the whole cable, because generally when the cable is cut all the fibers are cut.

This surveillance method provides approximately 90% faults detection on cables.



Test Signal from the ONMS

Fiber under test that is not used for traffic

Test signal sent to ONMS

Cable under test
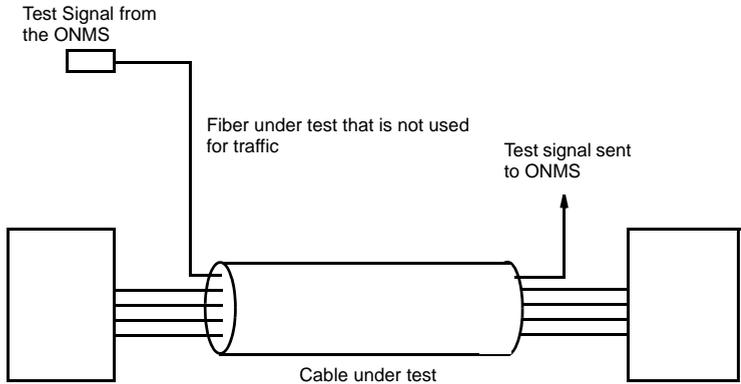
**Fig. 1**     Surveillance of a cable using a free fiber (or dark fiber)

### Advantages of this method

• It is possible to use the same wavelength in the In-Service fibers and in the tested fiber.
• No additional equipment is needed on the optical link.

### Disadvantages

• Loss of one or two fibers for the traffic.
• Other fibers than those tested could have faults.

### Active Fiber Method

A test signal of a different wavelength is superimposed on the traffic signals to monitor In-Service fibers. This method requires additional passive isolating components on the link: at the input, a Wavelength Division Multiplexor (WDM) to insert the test signal; at the output a filter to eliminate the test signal.
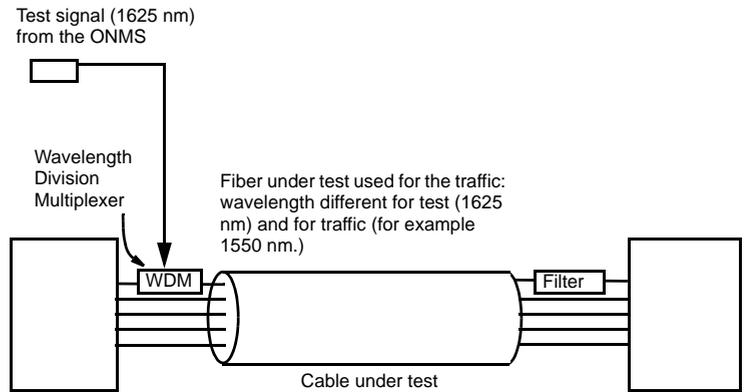
**Fig. 2** Monitoring an active fiber (in service)

***Advantages of this*** • Every fiber can be monitored.
***method:*** • Every fiber can be used for the traffic.

This method is used to monitor critical fibers or when there are no free fibers available in the cable to allow the dark fiber method.

***Disadvantages:*** • Additional loss due to the added components.
• Higher cost.

# Fault Detection Method

### Possible faults

Two types of faults can appear on an optical fiber:

• a break due to: an incident (roadworks, fire, road or railway accident), or sabotage, or an animal attack (rodents).
• damage due to aging of splices and connectors, dust or damp penetration.

### Fault detection process

A schematic is created for each fiber of the network monitored and stored in memory in a database together with the results of a reference measurement. Markers on the reference OTDR traces represent detected events (connectors, splices, etc.).

During monitoring, whenever a measurement is made, a new OTDR trace is acquired. The levels are compared, at the positions identified with markers, to the reference trace. If the difference is outside the defined threshold, an alarm is generated.

The alarm notification will give the location of the fault.
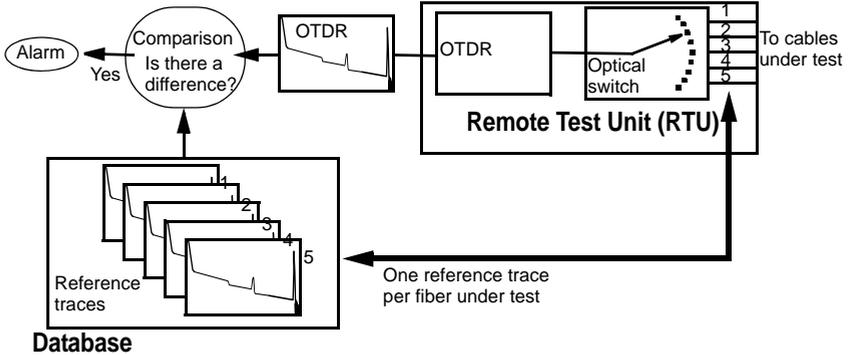


### Conversion of optical lengths to physical lengths (or «Civil works»)

On the backscatter traces the distances correspond to the length of the optical fibers. In other words, these optical fiber lengths do not correspond exactly to the physical distances (or «Civil works») because on one hand, additional loops of fiber are provided as a safety measure; and on the other hand, the fibers are twisted in the cable which adds approximately 3% to their length.

The backscatter trace provides the optical distances to the connectors and splices of the fiber. Knowing the physical distances, it is possible to calculate the ratio between the physical and optical distances on a fiber.

This ratio enables the deduction of the physical distance of the fault from the optical distance.

# WDM analysis principle

The Wavelength Division Multiplex (WDM) technology is a very effective way to increase the capacity of the fiber without having to install new links. Instead of increasing the data rate, it enables the fiber to carry several wavelengths, each wavelength transmitting signals. Channels are defined by the ITU-T recommendations G-692.

With this new technology, new measurements are required. During installation and maintenance of WDM systems, it is important to verify the following parameters:

1. Presence of all channels at corresponding wavelengths, without any drift.
2. Right channel power levels, without any power variation
3. Enough signal over noise ratio (SNR). This value is determined by measuring the ratio of the peak channel power to the noise power level of the ASE[1] noise signals to the left and/or right of the carrier. Usually, the noise measurement point is chosen as the calculated mid-point between two adjacent channels. The measured noise power level is converted to a standardized bandwidth of 0.1 nm.

The most important test equipment for WDM systems able to perform those test is the spectrum analyzer. This analyzers can be connected along the WDM system on all critical measurement points, located at the extremities of the links or at the amplifier locations.

## Measurement results

The optical spectrum analyzer displays a waveform which is showing all the channels. Measured results are presented on a complete spectral analysis and as a WDM table, listing all carriers with their relevant parameters.

The optical spectrum analyzer is performing automatic detection and measurement for all channels.

The number and the composite power are provided like, for each channel:

- the wavelength
- the spacing
- the power level
- the signal over noise ratio

The total power of the system could also be provided.

In order to qualify the amplifier, tilt and slope gain could be calculated. In principle, the amplification shall be flat over the full WDM spectrum.

---

1.Amplified Spontaneous Emission

# Description of system

3

The ONMS system is based on a Client/Server architecture.

This chapter describes the main components of the ONMS system and the main notion it uses:

# Main parts of the system

The main components of the system are:

– **OTUs** (Optical Test Unit)[1] placed at the strategic positions on the controlled optical network. The units include an optical switch module and one or two OTDR or WDM modules for measurement and results processing. The modules are connected to the server by a data communications network (Ethernet or modem) and may be backed up by a second communications tool (modem).

– **a central server***,* the heart of the system, which manages an Oracle database containing all the information concerning the system:
  - network documentation,
  - link references,
  - lists of operators.

– **fixed or portable client terminals** connected to the server by an IP or switched network. The graphic interface of these terminals is used to access all the system data, giving to users having a password the access to these data and to the RTUs according to their privileges. This access is possible from any Client Station connected to the system, whatever its geographical localization. Inversely the server sends the alarms to every client station: they display them or not according to the selected filter.

– **JDSU OFM-500 Optical Fiber Mapping** brings **physical network documentation** to ONMS**.** This software developed by JDSU may be sold with or independantly from JDSU ONMS. Refer to OFM-500 User manuals and other documentation material for further information on how these two products work together.

– as an option, **the Web Client application** enables the connection to the system via Intranet.

---

1. JDSU RTU
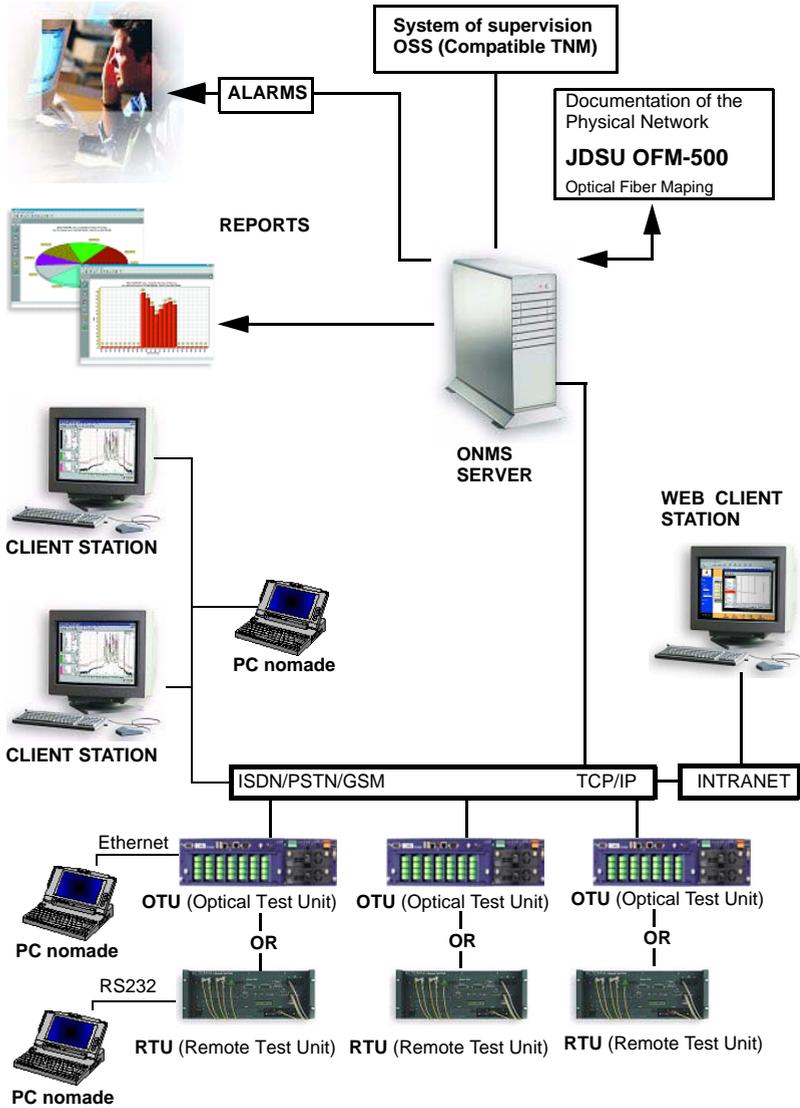
**Fig. 3**      Architecture of an ONMS system

The ONMS is a multi-user system which can be adapted to a regional organization. The definitions of the profiles of the individual users and the regions are logical and do not depend on geographic location. This flexibility means that the system is simple to adapt to your organization.

# Regions

A region meets the following criteria:

– It may contain several RTUs.

– An RTU belongs to only one region.

– For each region, a user has a set of privileges (called a «Profile»): he can open a session from any client terminal and use these privileges.

# Filters on alarms

On every Client Station, filters must be programmed to define the severity, the type and the category of the alarms it will display.

# Profiles

Thanks to its user profiles, the ONMS system offers a multi-user architecture which is flexible and secure.

A profile is made up of a series of authorizations:

– to consult or modify system resources,

– to modify links,

– to acknowledge and save alarms, and to change their severity.

By means of their passwords, users can access the different regions and resources with the privileges and access rights which they have been allocated. For example, a user may have only consultation access for one region but full administration privileges for another region.

To change the privileges of a user, the system administrator simply modifies this user's profile.

Any number of profiles can be defined, depending on requirements. Each profile will contain the particular privileges required and, as far as possible, the name it is given will indicate its use.

The following diagram shows the interaction between regions and profiles for a single user (X).
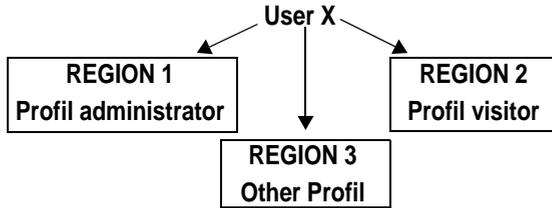
**General administrator**

The system has a pre-installed user, which cannot be modified. This is the general administrator, who is responsible for profile management.

The general administrator can access all the ONMS functions for any region. He alone has authority to:

– Add / delete a region
– Add / modify / delete a profile
– Add / remove a regional administrator
– Modify the parameters of the server.

**Pre-installed profiles**

Three profiles are pre-installed in the system.

### The Regional Administrator profile

The Regional Administrator profile cannot be modified. It may be attributed by the General Administrator to several users for the region concerned. For a given region, only this profile confers the right to:

– Add / Remove a user in the region
– Add / delete a client terminal
– Modify the On Duty Calendar for the region
– Modify the names and comments related to a region.

### The profiles of Expert and User

These profiles can be modified or deleted. They are typical profiles given as examples.

# Organizations

An Organization is a company or the department of a company operating on the DWDM network as owner or client. The ONMS system can allocate wavelengths to organizations.

# Alarms

During a network monitoring, the first event associated to an element of the system, or to an element monitored by the system, opens an alarm identified by a number: this number will remain the same until the alarm is solved. This alarm will change, according to events occurring to the element.

An event corresponds to every change in the status of an element : defect, disappearance of a defect, etc.

### Routing of alarms

When an RTU detects an event:

**1** The RTU sends an alarm message to the server.

**2** The server create an alarm depending on the event type and forwards it to every client terminals. The alarm will be displayed by the station, if the filter allows it. The alarm can also be sent by fax, pager or e-mail if the option «On Duty» is available.

**3** Users who have the corresponding privilege acknowledge the alarm on a client terminal or via the web

**4** The fiber is repaired. If necessary the schematic is modified and a new reference trace is saved.

**5** The RTU detects the repair and informs the server that normal status is restored.

**6** The server forwards the «back to normal status» message to all the concerned client terminals.

**7** The users acknowledge this message.

All these operations are recorded in the database.

# ONMS Key

JDSU assigns to each ONMS system a key giving the installed options. The key consists in the following elements:

– Server identification number (Host ID)

– ONMS software version.

– Number of ONMS client terminals.

– Options (SNMP, on duty management...).

– Number of simultaneous WEB connections that can be opened.

A new ONMS key is given by JDSU with every system configuration update: new options, software updates, etc. The new ONMS key can only be entered by the System Administrator.

To enter a new ONMS key, refer to "Entering the ONMS Key" page 33.

# Client Server linked by telephone network

It is recommended that a LAN/WAN type data communication network be used between the server and the client station. Only a Nomad station can choose between a modem and a network connection.

# ONMS system resource

### The RTU are equipped with:

– Up to two modules (OTDR and/or WDM).

– Up to 24 ports Optical Switch.

– One or two interface: Ethernet, ISDN, PSTN or GSM.

– The optical alarm input/output (option)

For more information report to the RTU user manual.

### The OTU (Optical Test Unit) are equipped with:

– Up to two OTDR modules.

– Up to 24 ports Optical Switch.

– Three relay contacts for alarms (option).

– One Ethernet interface

– One PSTN interface (option)

– One GSM interface (option)

For more information report to the OTU user manual.

### Operating System

– ONMS server: Windows 2003 server
– ONMS clients: Windows XP
– ORACLE embedded License (Data base)

### Minimum configuration of the client stations

– Microprocessor Pentium
– RAM: 2 Gb
– Hard disk: 40 Gb
– CD-ROM unit
– Ethernet Interface
– Possible Options: see the commercial documentation

# Man-machine interface

4

This chapter gives the process to open an ONMS session on a client station and describes the man-machine interface on the client station in the following section:

# Opening an ONMS session

Once the system is installed, to open an ONMS session from a Client Station, proceed as follows:

**1** Press the **Ctrl + Alt + A** keys at the same time.

The **Session Information** dialog box is displayed:



**Fig. 4** Session Information Dialog Box

**NOTE**

When the system is used for the first time, no user is created.
A «general administration» session can thus be opened with the alias ADMIN and the password PASSWORD. **We advise the administrator to change this password.**

**2** If another user has already opened another session on this terminal, click on the CLOSE THE SESSION button.

**3** Enter your alias in the corresponding field.

**4** Enter the password you have been given by the administrator in the PASSWORD field.

**5** Press OK.

The ONMS Client window is displayed to allow you to perform administrative or other operations.

Time zone   User name

ONMS version   Client Station
name
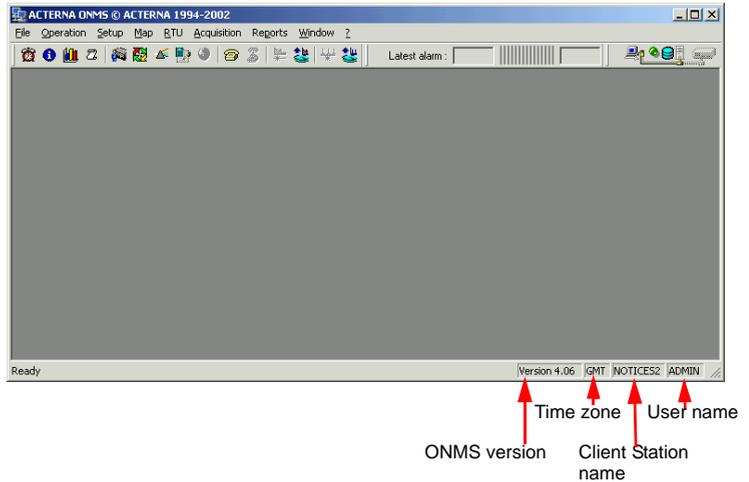
**Fig. 5**        ONMS Session

## Changing your password

To change your password:

**1**    Press CTRL + ALT + A. The SESSION INFORMATION dialog box is displayed.

**2**    Click on the CHANGE PASSWORD button.



**Fig. 6**        Changing a password

**3**    In the USER INFORMATION / PASSWORD field, enter the current password.

    **4**    Enter the new password in the NEW PASSWORD / PASSWORD field.

    **5**    Then enter the new password a second time in the CONFIRMATION field.

    **6**    Press OK.

## Closing an ONMS session

To close an ONMS session on a client terminal:

    **1**    Press **Ctrl + Alt + A**. The SESSION INFORMATION dialog box is displayed.

    **2**    Click on the CLOSE THE SESSION button.

    The icon 🔒 is then displayed next to each element onto the left box and no more action is available.

> **NOTE**
>
> The ONMS session opened closes automatically if a end of automatic session has been configured in Setup > Client > Miscellaneous (see ).

# Description of bars and menus

The ONMS interface is structured in the same way as Windows explorer™:

– management of folders and items as in Windows Explorer™.

– menu bar providing access to all the system functions.

– activity bar displaying the in-progress status of Alarms, Network and RTU.

– tool bar providing rapid access to the main functions via buttons.

– shortcut menu providing rapid access to certain actions using the right mouse button.

– access to menu bar commands via shortcut keys (letter of the command underlined or **Ctrl + letter**).

> **NOTE**
>
> In the following chapters, actions are sometimes indicated only by the tool bar button. The user may still use the equivalent command in the menu bar or the shortcut menu obtained by a right-click.

The different functions are described in detail under separate headings in this manual.

**Menu bar** The menu bar at the top of the screen provides access to all system commands:

| | |
|---|---|
| File | Import |
| | Export |
| | Print Setup... |
| | Quit |
| Operation | Active alarms |
| | Events |
| | Dashboard |
| | Alarm history |
| Setup | Administration |
| | Physical layer |
| | Photonic layer |
| | Measurement |
| | Client |
| | Server |
| | Database |
| Map | Open MAP |
| RTU | Connection |
| | Disconnection |
| | Activity |
| | Rebuilt |
| Acquisition | OTDR measurement on demand... |
| | Open OTDR trace... |
| | Spectrum on demand... |
| | Open Spectrum... |
| Report | OTDR performance report |
| | OTDR alarm history export |
| | Setup: OTDR alarm report |
| |        OTDR performance report |
| Window | Cascade |
| | Tile |
| | Arrange icons |
| ? | About ONMS (software version) |

**Tool bars and activities** The tool bars provide rapid access to certain functions by clicking on buttons.

The activity bars display current status: alarm, network, RTU, connection, progress of tests...

The user may choose whether to display or hide these bars. They can also be moved and rearranged on the screen.

> **NOTE**
>
> You can only display the RTU activity bar if you are connected to the RTU.

### Displaying or hiding a tool bar:

– Right-click in an empty part of the bar to obtain the tool bar display menu.



**Fig. 7**      Tool bar display menu

– Click on the name of the bar to be displayed or hidden. The tool bar names which are checked will be displayed.

### Moving a tool bar

Each tool bar includes a slide handle. On a horizontal bar, this handle is located on the left-hand side, while on the vertical tool bar it is located at the top. Click on the handle and do not release it during the movement.

– Move the handle towards to the extreme left for a vertical position.
– When a bar is separated from the main menu bar, it is highlighted in blue for movement. To group it with the menu bar again, it must be moved towards it.

Use these docking handles to either move or change their positions

Bar separated from the main menu bar
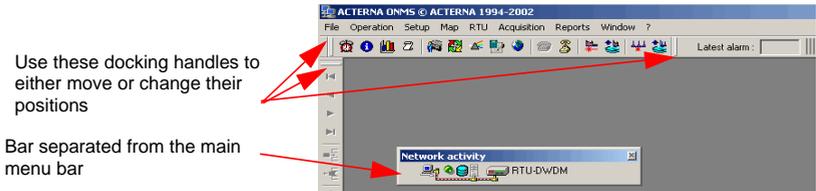


### Control bar

The icons in this bar provide shortcuts to the main commands of the menu bar.

Active alarm table — Dashboard — Resources Administration — Photonic layer — OTDR measurement — Open an OTDR trace

Events — Alarm history — Physical layer — Measurement processing — RTU connection/disconnection — Optical spectrum measurement — Open an optical spectrum
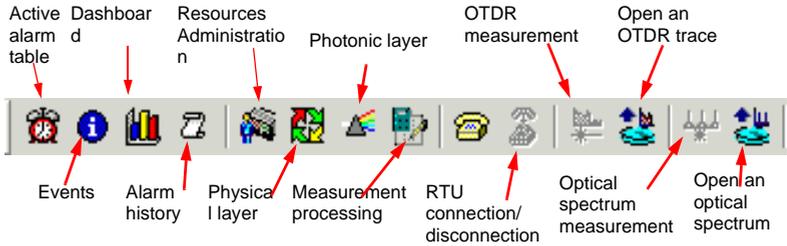
### Document bar

The document bar is used to move around in the menus and dialog boxes and access the database. The document bar commands are also available in the shortcut menu obtained by a right-click.



Move to
First — Previous — Next — Last

Update current configuration from database

Add — Delete — Apply in database

Cancel last data input not saved
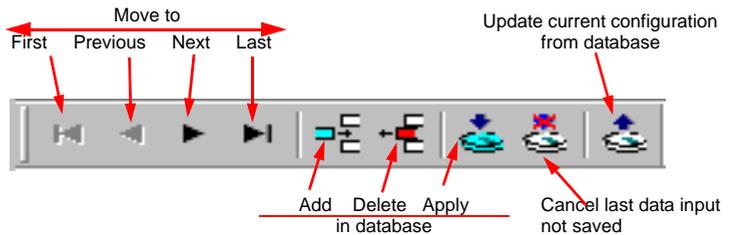
### Alarm activity bar

When a client station receives an event, a color indicating the severity of the alarm scrolls on the alarm activity bar (see "Severity levels of alarms" page 186).



Latest alarm : 5722 | Client
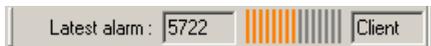
In the bar are also indicated: the number of the last alarm and its type (OSA, RTU, OTDR, etc.). A click on the scrolling bar opens directly the active alarm table.

### Network activity bar

The network activity bar indicates in diagram form:

- whether the RTU is connected and, if so, its name (the RTU and its link are shaded if it is not connected).
- database access by a green flashing signal.

Green during database access

RTU connected

RTU not connected
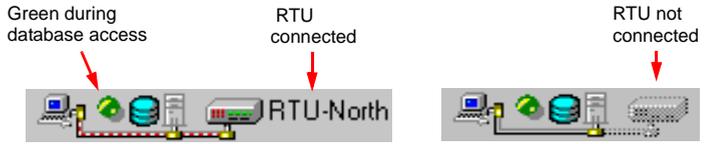
**Fig. 8** Network activity bar (with and without RTU connected)

### RTU activity bar

This bar shows the name of the tested link, type of test in progress, acquisition status and acquisition parameters.



### Status bar

Version of ONMS application

Time zone



System status

Client station name

User name

The Time zone is entered in the SETUP > SERVER >PROPERTIES MENU.

### Shortcut menu



During resource administration, a right-click on a folder or item opens the shortcut menu to allow selection of the action to be performed on this folder or item.

Each action in this menu has an equivalent command in the tool bar:

| Menu | Function | Button |
|------|----------|--------|
| First | go to first item. | |
| Previous | go to previous item. | |
| Next | go to next item | |
| Last | go to last item. | |
| Add | add the selected item | |
| Delete | delete the selected item after confirmation. | |
| Apply | save a new item or modification in the database | |
| Cancel | erase the new data entered if not yet saved in the database | |
| Refresh | update the configuration from the database. | |
| Connection | connect the RTU | |
| Disconnection | disconnect the RTU | |

# Access to configuration and operation menus

During the ONMS administration or operation, the left part of the display is structured like the WINDOWS EXPLORER ™ for directories and files:

– a folder is opened by a left-click on its name or the ⊞ which precedes the name: the content of the folder is displayed.

– a folder is closed by a left-click on its name or the ⊟ which precedes the name.

A left-click on an item opens the corresponding menu in the right-hand part of the screen. The folder or item selected is highlighted.

Selected item                                    Menu for selected item.



Saves data just entered          Deletes information which      Refreshes configuration
in the database                  has just been entered          with information from
                                                                 database
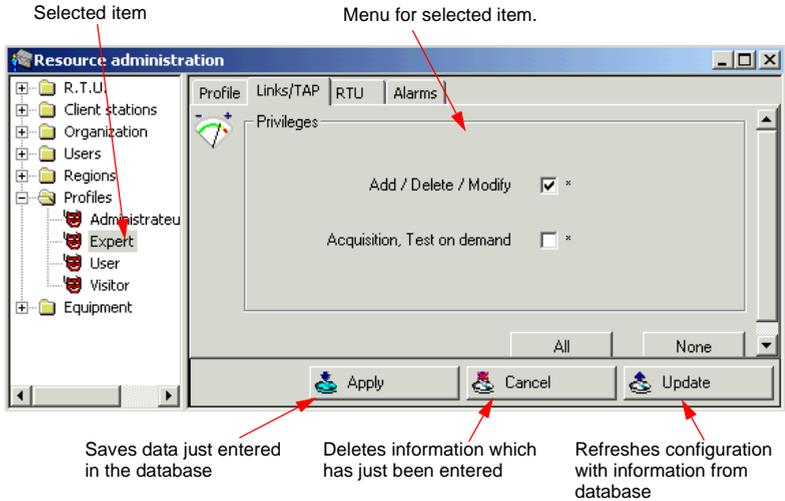
**Fig. 9**        Example

It is now possible to consult or modify the displayed parameters (depending on user privileges).

**Adding an item to a folder**

The following procedure is used to create, for example, a new region, user or profile.

– Select the folder (region, user, profile, etc.).

– Click on the ▣ button in the tool bar or right-click on the folder then choose ADD in the action menu.

– The name and parameters of the new item can then be entered in a dialog box.

– Click on the ▲ icon in the tool bar to save the information in the system database.

**Deleting an item from a folder**

The following procedure is used to delete, for example, a region, user or profile.

– Select the folder (region, user, profile, etc.).

– Click on the ▣ button in the tool bar or right-click on the folder then choose DELETE in the action menu.

The item is deleted from the list after confirmation is given.

In the case of deletion of system resources, see (See: "Conditions required and consequences of removing" page 67).

**Entering parameters (profile, region, etc.)**

To enter a parameter previously defined (RTU, Client station, Organization, User, Profile, Region, etc.) in a dialog box:

– In the left part of the screen, click on the parameter name: name of region, profile, etc. ...

– Hold down the mouse button and drag the parameter to the dialog box field where you wish to enter it; release the mouse button.

# System administration

5

This chapter describes all the ONMS system administration tasks which allows the user to configure the resources.

Administration tasks can be carried out when necessary, but they are given here in the logical order in which they are performed when installing the system:

# Entering the ONMS Key

▬▬▬▬ **NOTE**

The ONMS key can only be entered by the system administrator.

The ONMS key must be entered at the first use of the system. After each system configuration modification (new software, new options, more client stations, etc), to access the new functions, the new key delivered by the supplier must be entered, otherwise an error message is displayed.

To enter the ONMS key:

**1** Open an ADMINISTRATOR session.

**2** Click on the ONMS KEY tab in the SETUP / SERVER menu.

**3** Enter the ONMS key.

**4** Click the APPLY button (or the OK button to validate the new ONMS key and close the SERVER SETUP menu).
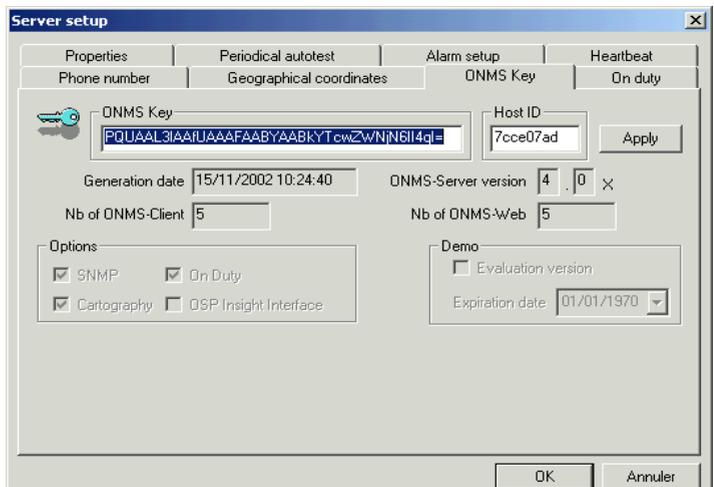


**Fig. 10**     Entering the ONMS Key

# Access to administration functions

To perform an administration function, the system administrator must:

**1** Open a session on a client station connected to the server (see "Opening an ONMS session" page 20).

**2** Click on the [icon] icon in the tool bar.

The RESOURCES ADMINISTRATION window opens. The directories containing the RTUs, the Client Stations, Organizations, Users, Regions, Profiles and Devices can be opened (see in the Chapter 3 'Description of system" the definition of these items).
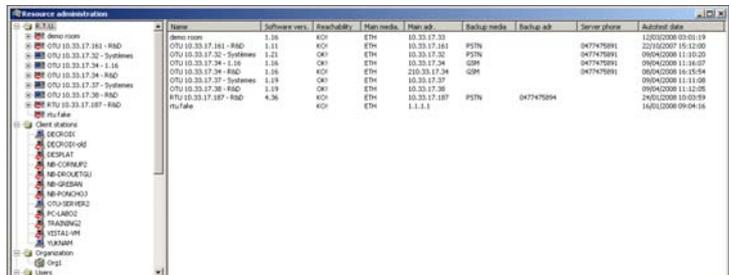


**Fig. 11**        Resource Administration window (for administrator)

The elements contained in the directories are marked with a pictogram:

– Those which cannot be changed by the user (read only) are displayed greyed out.

– The pictogram of those which are not accessible to the user is replaced by a padlock.

# Adding a region

**NOTE**

This function is accessible only to the general administrator.

To add a region:

**1** Click on the REGION directory

**2** Add a region (see "Adding an item to a folder" page 28).
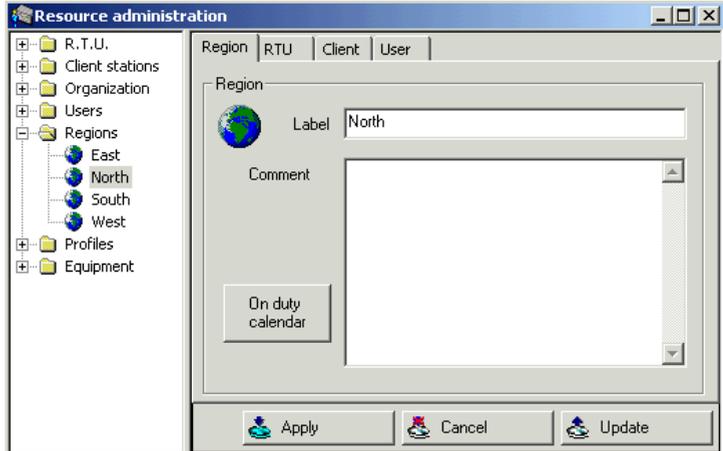
**Fig. 12** Defining a region (example with On Duty option)

**3** Enter a name to define the region in the LABEL field and any comments required for the region in the COMMENT field.

**4** Click on the Apply button to save the region in the database.

The RTU, CLIENT and USER tabs provide access to the lists of RTUs (name, address and availability), clients (name, address, availability and type) and users (name and profiles) allocated to this region.

**NOTE**

Double-click a line in the lists to go directly to the item to modify.

## Adding a profile

**NOTE**

This function is accessible only to the general administrator (see "Profiles" page 14).

To add a profile:

**1** Click on the PROFILES directory and add a profile (see "Adding an item to a folder" page 28).

**2** The Profile configuration window displays: enter the profile's name in the LABEL field.

**3** If required, enter a comment for the profile in the COMMENT window.
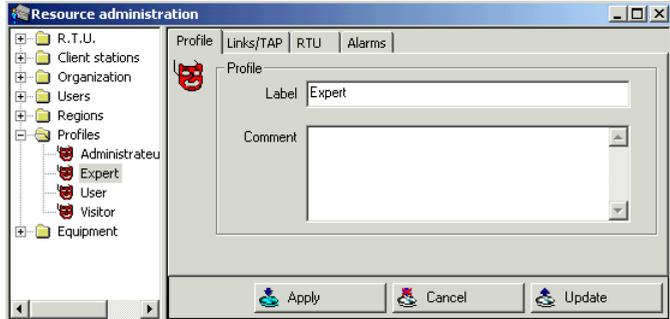


**Fig. 13** Profile Name

In the pages displayed when clicking the following tabs, privileges are granted by checking the appropriate boxes.

**NOTE**

In the configuration menus, the parameters not authorized to the current user are displayed as padlocks.

**4** Click on the LINKS/TAP tab

Check the boxes to provide the profile chosen with the following privileges:

- add, delete, and modify links and TAPs (viewing remains possible).

- acquire an OTDR trace or a spectrum.

- run a TEST ON DEMAND on a link or a TAP.
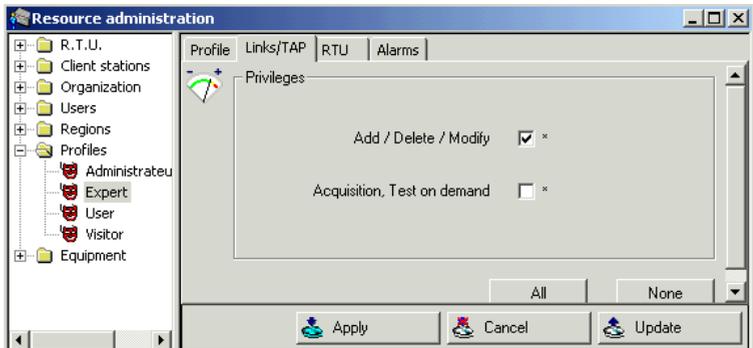


**Fig. 14** Allocating links privileges

**NOTE**

Following the TEST ON REQUEST, an alarm message displays: it can be acknowledged only if the user has the appropriate privilege (see "Allocating privileges on alarms" page 37)

**5** Click on the RTU tab.

Check the boxes to set the authorizations related with RTUs:

- modify the hardware configuration.

- run the RTU autotest.
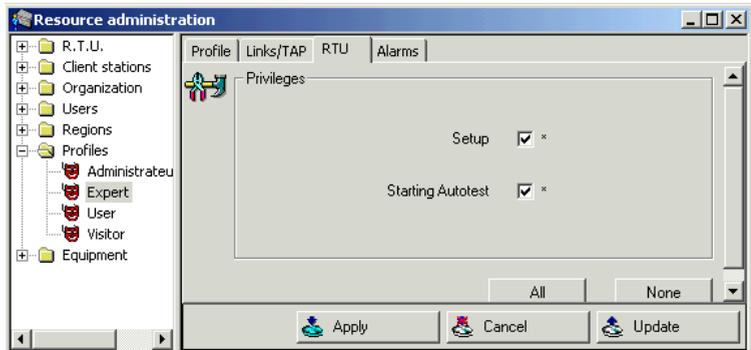


**Fig. 15** Allocating privileges on RTUs

**6** Click on Alarms tab
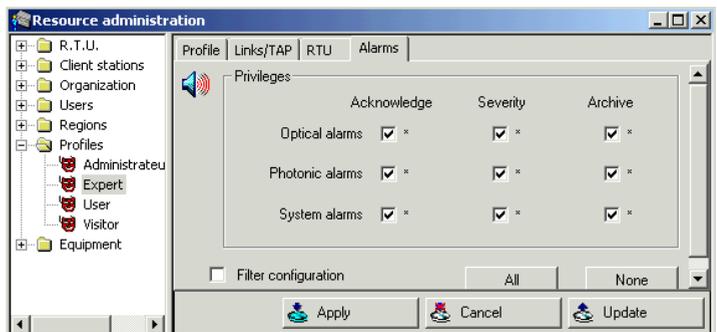


**Fig. 16** Allocating privileges on alarms

**7** Define whether the profile enables :

– acknowledging physical, photonic, or system alarms

– modifying an alarm's severity (e.g. from MAJOR to CRITICAL)

– archiving the alarms acknowledged with a RESOLVED severity.

– If the Filter Configuration case is notched, the user will have the authorization to modify an alarm filter. If not, he will only see the alarms from the actual configured filter, but will not be able to modify it (see page 200).

See Chapter 9 'Alarms management", for more information on alarms.

# Adding a user

> **NOTE**
>
> This function is accessible only to the general and regional administrators.

The ONMS user still has web access, if the option is installed. Access is also possible using a client station, if its configuration enables it.

A Web user is authorized to view :

– several regions with the profile associated to each region.

– one or several independent links with a profile associated to each link.

– one or several TAPs with an associated profile.

Each user is associated with a name, a nickname, and a password, as well as an authorization for certain regions and a profile per authorized region.

In addition, if the ONDUTY MANAGEMENT option is installed, one or several onduty regions can be defined for each user. See chapter .

**User identification**

1  Click on the USER directory

2  Add a user (see "Adding an item to a folder" page 28).

3  The user configuration window displays, with the USER page active: enter the user's nickname in the ALIAS field.
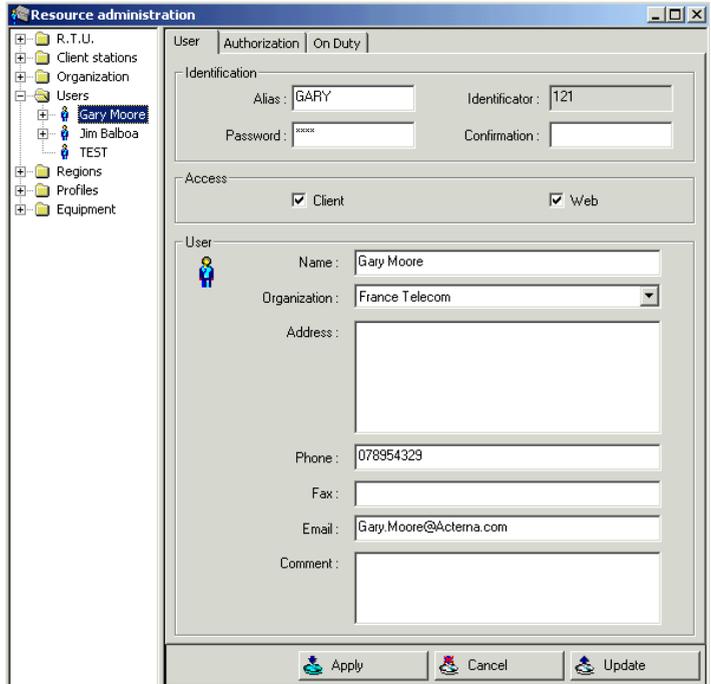
**Fig. 17** Identifying a user

**4** Enter the user's password in the PASSWORD field, then confirm it by entering it again in the CONFIRMATION field.

The identification number allocated by the system is used to enable the on duty user to identify (if the ON DUTY management option is present).

**5** Set the user's access rights (Client Station, Web access, Client + Web access).

**6** Enter the user name in the appropriate field.

**7** If the user is part of an organization, select it.

**NOTE**

The password and nickname are indispensable, as they enable logging on ONMS (see "Opening an ONMS session" page 20). The Address, Telephone number, Fax number, Email and Comments fields can be entered, but are optional.

**User authorizations** Define the regions, links and TAPs the user will be able to access from a client station or the web, including the appropriate profile.

Web users are provided with access to the ONMS only through the Web, and will see only the links allocated to them. The ONMS users always have Web access.

**1** Click on the AUTHORIZATIONS tab.

You can then enter the information related with access by Client station and the Web.

*Clients/Web Authorizations*

**1** In the left-hand field, open the REGIONS and PROFILES directories to display the regions and the profiles they contain.

**2** Drag and drop the region selected in the left-hand list into the right-hand authorization field. You can also right-click on the REGION column and select the region in the list displayed.

**3** Drag and drop the profile selected into the authorization field on the right. You can also right-click on the PROFILE column and select the profile in the list displayed.

**NOTE**

If the two items REGION <u>and</u> PROFILE are not entered, you cannot save the information.

**4** Repeat this procedure for all regions accessible to the user.

– To remove a «REGION + PROFILE» configuration from the authorization list, select it, then click on the REMOVE button.

– Click on the  icon to save the new user information into the database.
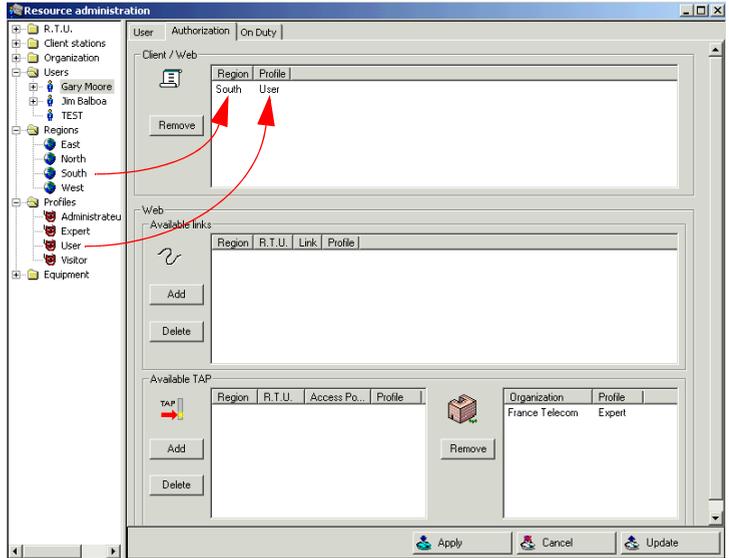
**Fig. 18**          Allocating user authorizations

*Web Authorizations*  In the WEB field, the administrator can enter the Links, TAPs and Organizations the user will have access to through the Web, with a profile associated to each.

1   **To configure the links**, under AVAILABLE LINKS , click on the ADD button.

–   Select the profile to be applied to a link or a group of links.

–   Select a region: the list of RTUs monitoring this region displays in the RTU field.

–   Select an RTU: the list of links monitored by this RTU displays in the List of links field.

–   Select one or several links, and move them into the right-hand field using the > key,
or move all the links to the right-hand field using the  >> key.

–   Click on the OK button.

–   To cancel the Web user's access to one or several links, select these links in the right-hand field, then click on the < button to move them to the left-hand field, or to move all the links to the left-hand field, click on the << button.
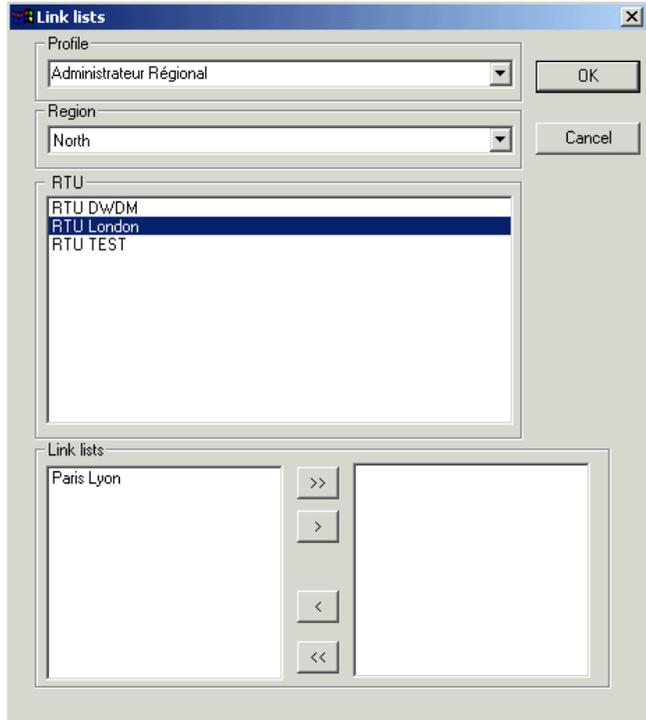
**Fig. 19**      Adding a link to the Web user

**2** **To configure the TAPs**, in the TAPS ACCESSIBLE field, click on the ADD button, then apply the same procedure as with links.

**3** **To configure the Organizations**, in the left-hand field, open the Profiles and Organizations directories to be provided with access to their contents.

– Drag and drop the organization selected in the left-hand list into the right-hand authorization field. You can also right-click on the Organization column, and select the organization in the list displayed.

– Drag and drop the profile selected into the authorization field on the right. You can also right-click on the Profile column and select the profile in the list displayed.

**NOTE**

If the two ORGANIZATION and PROFILE items are not entered, you cannot save the information.

**Defining on duty levels**

If the ONMS system includes the On duty management option:

**1** Click on the **On Duty** tab.
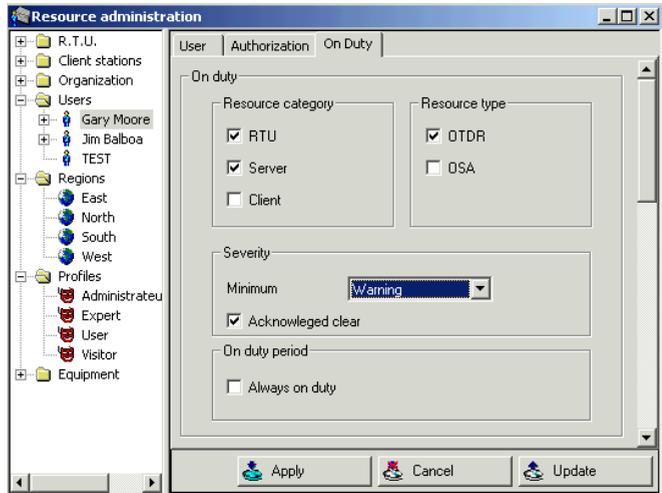


**Fig. 20** Configuring a user's on duty

The regional administrator must then specify which alarms will trigger user calling during the latter's standby period, by selecting:

– the origin of the system alarms: RTU, SERVER, CLIENT STATION

– the monitoring type: OTDR, OSA

– the minimum level of severity of optical alarms: WARNING, MAJOR, MINOR, CRITICAL.

You can select **Acknowledged clear** in order to not take into account the clear alarms.

Generic users can also be created, such as supervision centre, maintenance centre, etc.

By checking the ALWAYS ON DUTY box, the user is always notified for the type(s) of alarms selected.
The user then has a MEDIA subdirectory, enabling to define the communication means used, and allocating standby periods (refer to chapter )**.**

**NOTE**

A user can be validated only when at least one region was allocated to him, in one of the dialogue boxes opened in the AUTHORIZATION or ON DUTY tabs.

# Adding an ONMS client station

▬▬▬▬ **NOTE**

Configuring a Client Station is accessible only to general and regional administrators. The other users can only change its availability.

When the administrator (regional or general) opens the ONMS software on the Client Station for the first time, the following box displays:
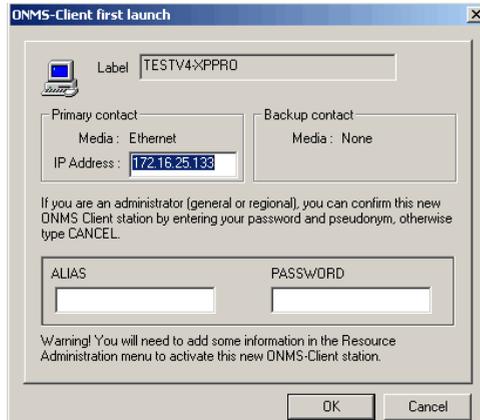


**Fig. 21**      ONMS client Declaration

To confirm the automatic addition of this station, the administrator enters his name and password, then clicks on OK. The station then displays, under the CLIENT STATION directory. To configure this station, the administrator proceeds as follows:

**1**   Click on the station name, in the Client Station directory.

The configuration window displays, with the CLIENT STATION page active.
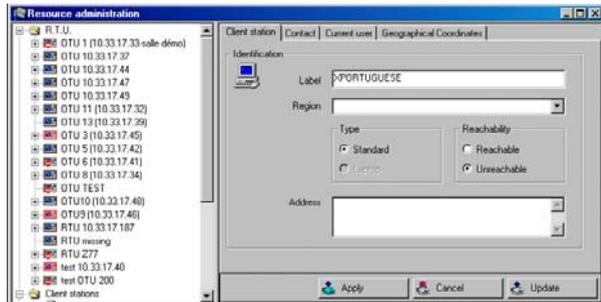
**Fig. 22**     Client identification

The Client name in the LABEL field is added automatically (this station's «network» name).

Select the Region to which the client's station belongs in the list.

The type of client is indicated in the TYPE field:
STANDARD: he receives and displays the alarms according to the filters set (see "Filtering alarms" page 189).

LAPTOP:     he uses a laptop PC and will not receive the alarms.

The station's reachability can be modified in the appropriate field:
REACHABABLE or UNREACHABLE.

---

**NOTE**

Unreachability inhibits the reception of all alarms, but does not prevent the Client Station from operating.

---

**NOTE**

It is recommended to set a client station unreachable when not connected to the network anymore, or in case you don't wish to receive alarms on this machine for a while. Nevertheless, the server will detect if you are not reachable even if you did not set it unreachable. (See "Automatic unreachability detection (for workstations)" page 73)

**2**   Click on the «Contact» tab.

The media for communication used displays: ETHERNET, ISDN or PSTN.

–   Using the ETHERNET communication means the client's IP address is allocated automatically upon the client's first connection to the server.

–   Notch the case «Use server public IP address» to establish communication client <->server via Internet.

– Using the ISDN or PSTN communication means, the administrator must enter the telephone number to be called by the server to escalate the Client Stations.
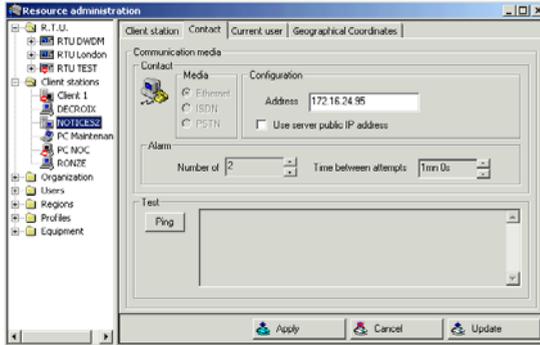


**Fig. 23** Primary contact using the Ethernet media

– Regardless of the communication means, enter in the appropriate fields, the maximum number of attempts to establish contact and the time-out for each attempt.

– Using the Ethernet and ISDN communication means, click on the TEST button to test the link between the client and server. Once the link is established, the server's response time and address display.

**3** Click on the CURRENT USER tab to display the identity of the last user having logged on using this Client Station, and the log on date.
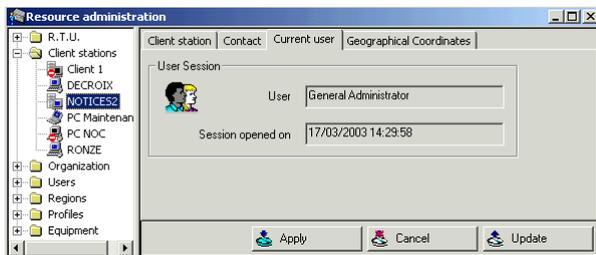


**Fig. 24** Current user

**4** Click on the GEOGRAPHICAL COORDINATES tab if the option is present to enter or view the station's coordinates. These coordinates are provided for information only.

**5** Click on the APPLY button to record the information you have entered for the client in the database.

# Adding an organization

Each organization intervening over the DWDM network is defined by its name, address, telephone number, etc.

To add an organization:

**1** Click on the ORGANIZATION directory. The list of organizations displays on the right.

**2** Adding an organization (see "Adding an item to a folder" page 28).

The configuration window displays, with the ORGANIZATION page active. Enter the identification parameters of the organization.
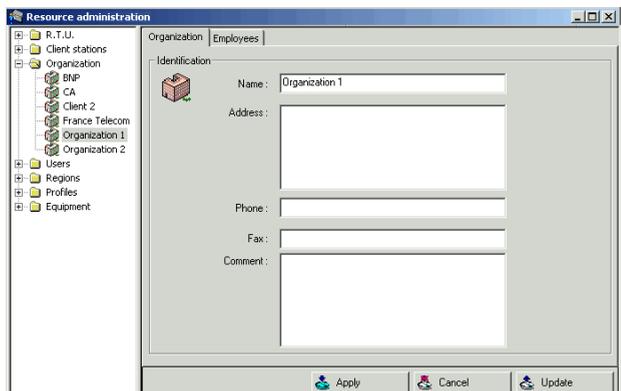


**Fig. 25**     Organization Identification

**3** Click on the EMPLOYEES tab

The list of users allocated to this organization then displays, as well as their means of access to the system (client station and/or web).

If the interveners do not display, add them (see "Adding a user" page 38) by entering the name of their organization.

# Adding an RTU

Each RTU is defined by its name, the region it monitors, its IP address, and its geographical coordinates. It can be fitted with OTDR modules for the reflectometry readings or WDM modules for DWDM monitoring.

<table>
<tr><td>████</td><td>**NOTE**<br><br>The fields which are greyed out in the RTU dialogue boxes cannot be modified. To modify them, you must usually log onto the RTU.</td></tr>
</table>

### To add an RTU:

**1 Click on the R.T.U. directory**

– Adding an RTU (see "Adding an item to a folder" page 28). The RTU configuration window displays with the CASE page active.
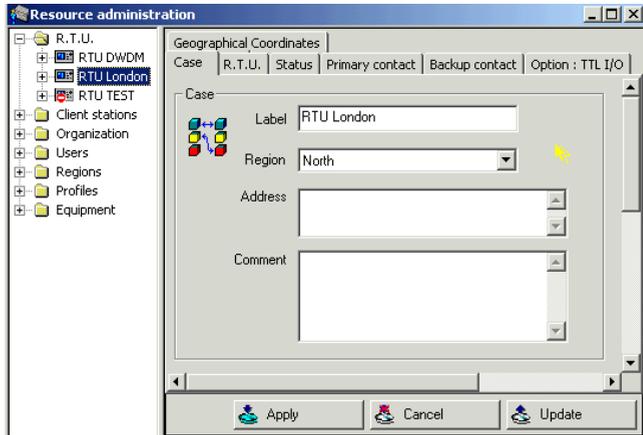


**Fig. 26**        Identifying the RTU (with an OTDR module)

– Enter the RTU's name in the LABEL field.

– Select the region containing this RTU in the Region scroll menu.

<table>
<tr><td>████</td><td>**NOTE**<br><br>When the RTU detects a fault, the alarm is sent to the client stations available through the server.</td></tr>
</table>

– If required, enter the RTU's address and any appropriate comments concerning this RTU. Both fields are optional.

**2 Click on the Primary contact tab to define the primary means of communication between the server and the RTU.**

– Select the type of media in the Media menu:
  - ETHERNET
  - ISDN or PSTN (modem).

**NOTE**

The ISDN or PSTN media is available only if:
- the RTU is fitted with an appropriate modem.
- the RRAS service is installed on the server.
To add these options to the system, call customer service JDSU.

– Enter the parameters required (RTU's IP ADDRESS, telephone number, etc.) matching the media selected.

– Enter the number of contact attempts, in case of failure upon the first attempt, and the time-out between the attempts.

– Notch the case «Use Server Public IP Address» to establish the communication via Internet.

– Click on the TEST button to test the Ethernet link between the RTU and the server. Once the link is established, the server's response time and address display.
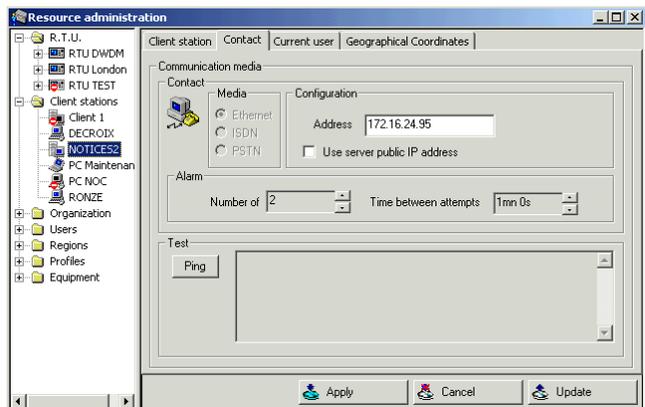


**Fig. 27**      Primary contact using the Ethernet media

– In the case of contact using the ISDN or PSTN media contact has been selected, additional parameters must be entered concerning the server:

– CLIENT IP: enter a virtual address which does not exist in the system. To prevent address conflicts, use addresses of the type 220.0.0.xxx

– TEL #: enter the RTU's telephone number.

– TEL PREFIX: enter the prefix to use when the RTU call the server. For example, 0 when the RTU accesses the outside network through a switchboard or the country code when the server is located in a foreign country.

**3 Click on the «Backup contact» tab to configure the backup contact means between the server and the RTU.**

Proceed as with the primary contact means, but as the link is possible only through a modem, only the ISDN and PSTN options are proposed.
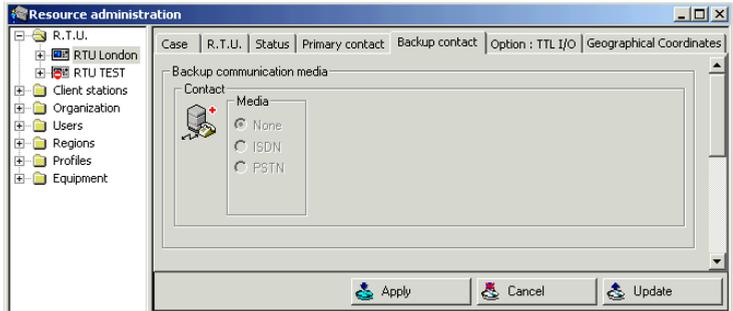


**Fig. 28** Backup contact

━━━━━ **NOTE**

In the case of a GSM modem, select the PSTN media.

**4 Click on the «RTU» tab**

The RTU configuration must display in the RTU field.

– If the configuration does not display, log onto the RTU, by clicking on , then click on the AUTOCONFIGURATION button. The auto-configuration process is launched, and the configuration of the RTU selected displays.

━━━━━ **NOTE**

To retrieve the initialization parameters of the modems and the port configuration implemented locally or by a client, press on the CTRL key while clicking on the AUTO-CONFIGURATION button.
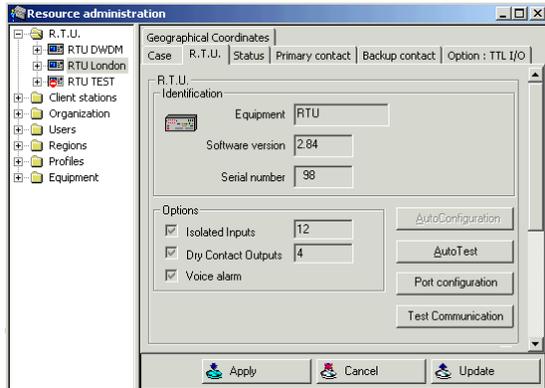
**Fig. 29**        RTU identification

–   To launch an autotest of the RTU, log off the RTU ( ![icon] ) if connected, then click on the AUTOTEST button. Wait a few minutes to receive the report.



**Fig. 30**        Example of autotest result

–   To test the communication between the 2 medias (Ethernet and Modem), click on the Test Communication button. Wait a few minutes to receive the following dialog box:



**Fig. 31**        Exemple of Test Communication result

The measurement equipment is often composed of:

–   an RTU fitted with a measurement module (OTDR or WDM) and internal switches
    or
–   an OTU fitted with one or two OTDR modles
–   external switches.

### Configuring the ports

The optical path between the cable head monitored and the RTU measurement module is normally entered on the level of the RTU (using the RTU SetUp software).
However, it can be modified using the following menu, displayed by clicking on the PORTS CONFIGURATION button.
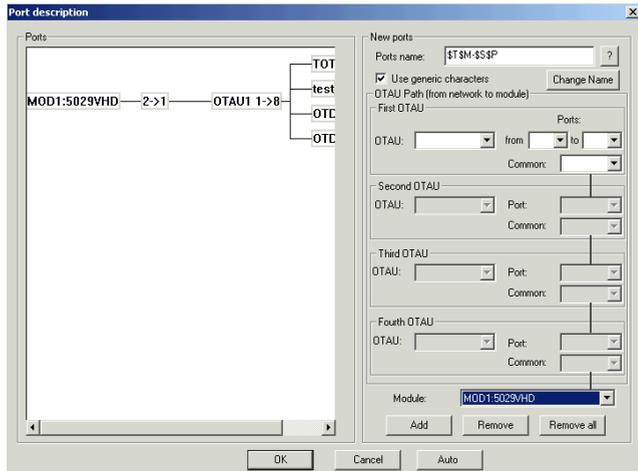


**Fig. 32**     Editing the RTU ports

Existing optical paths display in the left-hand field with the following indications for each of them:

– module name (e.g. MOD1) and type of module (e.g. 5027 HD) for the RTU used for the test,

– name of the successive OTAUs used, together with the successive input and output ports .

The name of a selected port can be modified:

– either by entering it directly in the PORT NAME field and clicking the CHANGE NAME button.

– or by checking the USE GENERIC CHARACTERS box, then entering in the PORT NAME field   codes which will change the port name automatically: the type of module ($T), its position ($M), the position of the optical switch connected to the network ($S) and the port on this switch connected to the network ($P).

To display the generic characters, click the «?» button. The following box lists them:

**Fig. 33**        Generic characters

To enter a new optical path:

– select from top to bottom:
  - the various OTAUs used since the cable head, and for each: the number of the output port used, as well as the common port number.
  - the type of measurement module used.

– then click the ADD button.

To add a port available on an OTAU to a path: select the path, enter the name of the OTAU and the port number, and click on ADD.

To delete a port not used by a TAP or a link, select this port, then click on the DELETE button.

To delete all paths, click on the DELETE ALL button.

If the RTU is set with only one OTAU, which is of type 1->N, click on the AUTO button to get the automatic configuration of the ports.

**5    Click on the Status tab**
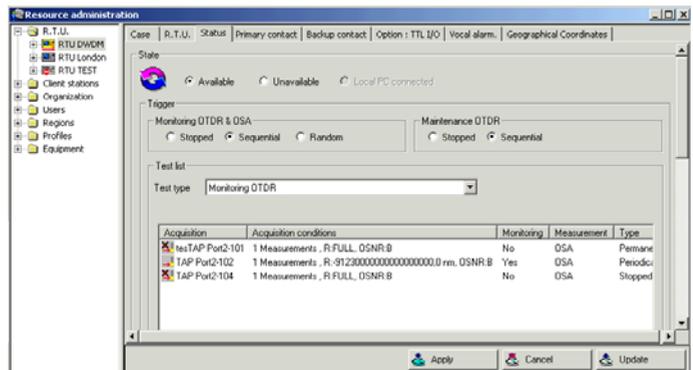


**Fig. 34**        RTU status

– Choose the RTU availability: if it is declared unavailable, it is excluded from the automatic autotest procedure. However, a

manual autotest can be performed, and the RTU will resume its monitoring and/or maintenance cycles.

– Select the type of launch for OTDR and OSA monitoring cycles and OTDR maintenance cycles:

STOPPED: the selected cycle stops.

SEQUENTIAL: the cycle is always run according to the same sequence.

RANDOM: the monitoring cycle is applied in an undefined sequence.

**NOTE**

Regardless of the type of test, **the sequencer must be on the Sequential or Random** setting to run the test cycles. The STOPPED position of the Sequencer does not prevent from running tests manually, but stops sequencing automatically.

– Select the **Test Type** in the list:
  – Monitoring OTDR
  – Maintenance
  – Monitoring OSA (with RTU only)

The list of links monitored by the RTU is displayed in the LIST OF TESTS field.

**6 Click on the «Geographical Coordinates» tab**

If necessary, enter the X, Y and Z coordinates of the RTU.
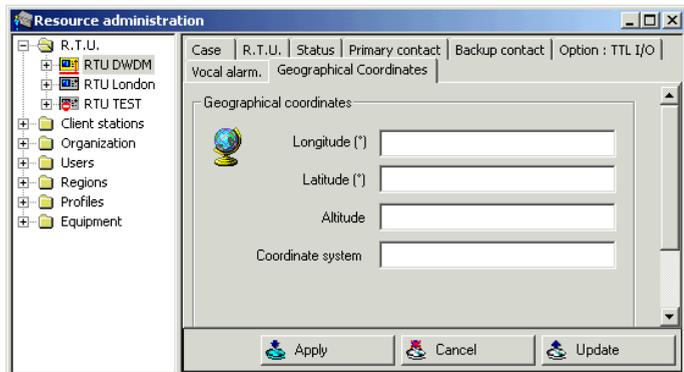


**Fig. 35** Geographical coordinates

**7 Click on the TTL I/O tab if the option is present in the RTU.**

If the option is present, the following screen displays:
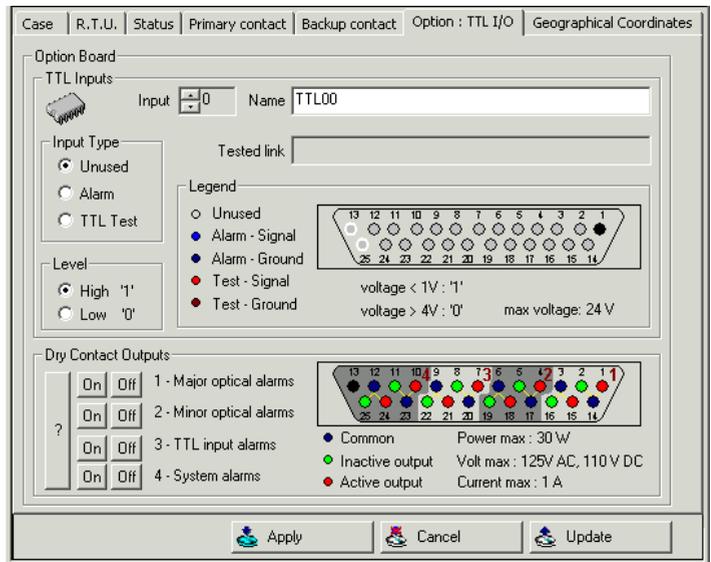


**Fig. 36** TTL I/O Option

It is possible to trigger the test of a link upon each change in the TTL signal's status on the input connector and take into account an external event (e.g. rack door opening) to trigger an alarm.

The input connector displays in the TTL Inputs, with as a caption:

– the color code indicates pin status:
  non used, used for an alarm or used for a test.
– the value of the TTL's signal «1» or «0» statuses.

**To program the test of a link when the TTL signal status changes on an input:**
– click on the appropriate connector pins
– in TYPE OF INPUT, select TEST TTL. The corresponding pins display in red.
– name this test in the NAME field.

The name of the link tested displays in the LINK TESTED field.

**To trigger an alarm upon the arrival of an event on a TTL input.**

– click on the appropriate connector pins
– in TYPE OF INPUT, select ALARM. The corresponding pins display in blue.
– name the alarm. This name will be used in the alarm message sent to the client stations concerned.
– in LEVEL indicate whether the alarm occurs upon the high level «1» or the low level «0»

**To view the «Dry Contact Outputs» the relays of the I/O boards**

The graphic representation of the output connector indicates the pin allocation to the 4 relays, with, in caption:

– the color code indicates pin status: common, output enabled or output disabled.
– the relay characteristics: power, voltage, and current.

These relays are enabled in case of major (contact 1) and minor (contact 2) optical alarms, alarms on TTL input connector (contact 3), and system alarms (contact 4).

To view the status of the relays, click on the «?» button: if the corresponding ON button is depressed, the relay is enabled; if the OFF button is depressed, the relay is disabled.

In case of an optical alarm, to set the relay to the disabled status, click on the corresponding OFF button. The relay also shifts into the disabled status upon alarm acknowledgment.

The ON and OFF buttons enable forcing the corresponding contacts of the relays to test them.

**8   Click on the Apply ⬇ button to save the information entered for the RTU in the database.**

The RTU is configured.

After auto configuration of the RTU, the RTU directory contains the MODULES directory, containing the identification page for the module(s) fitted in the RTU.

This page indicates the type of module, its position in the RTU, its serial number and, in the case of the OTDR module, its calibration parameters. Comments can be entered for this module.
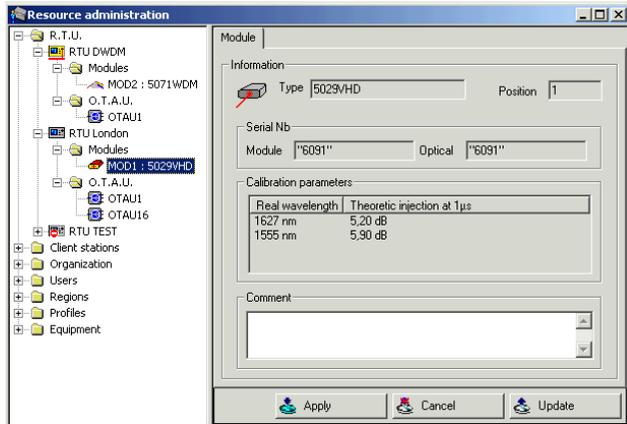
**Fig. 37**          Identification of an OTDR module
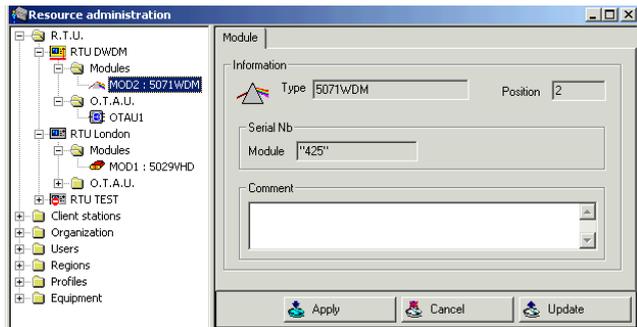


**Fig. 38**          Identification of a WDM module
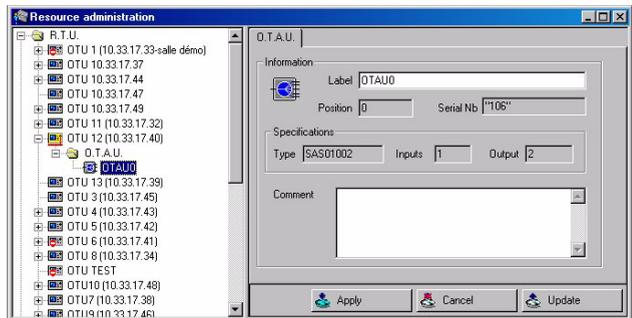


**Fig. 39**          Identification of an OTAU

# Adding an OTU

Each OTU is defined by its name, the region it monitors, its IP address, and its geographical coordinates. It can be fitted with OTDR modules for the reflectometry readings. It can also be fitted with an OTAU module (Optical Test Access Unit), which provide switching of up to 94 ports.

> **NOTE**
>
> The fields which are greyed out in the OTU dialog boxes cannot be modified. To modify them, you must usually log onto the OTU concerned.

**To add an OTU:**

**1 Click on the O.T.U. directory**

– Adding an OTU (see "Adding an item to a folder" page 28). The OTU configuration window displays with the CASE page active.
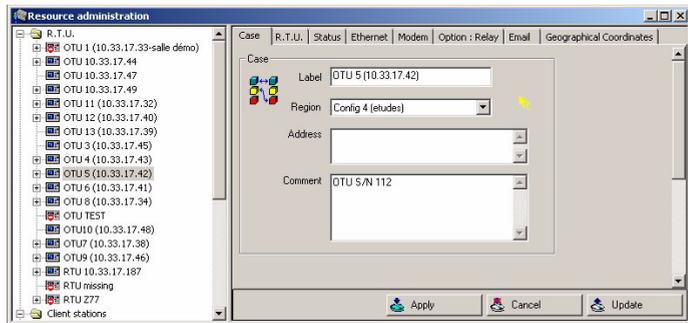


**Fig. 40** Identifying the OTU

> **NOTE**
>
> Until you start the AutoConfiguration, the box displayed will be for the addition of an RTU, and not for an OTU. The modification will be done automatically once the Autoconfiguration is executed (see step 4).

– Enter the OTU's name in the LABEL field.
– Select the region containing this OTU in the Region scroll menu.

> **NOTE**
>
> When the OTU detects a fault, the alarm is sent to the client stations available through the server.

- If required, enter the OTU's address and any appropriate comments concerning this OTU. Both fields are optional.

**2 Click on the Ethernet tab to define the communication between the server and the OTU.**

- Enter the IP address
- If necessary, notch «Use Server Public IP address» to establish the communication via Internet.
- Enter the time-out between the attempts, in case of failure upon the first attempt.
  The first attempt will be made alternately on the Ethernet media, then on the Modem (if included, see step 6).
- Click on the PING button to test the Ethernet link between the OTU and the server. Once the link is established, the server's response time and address display.
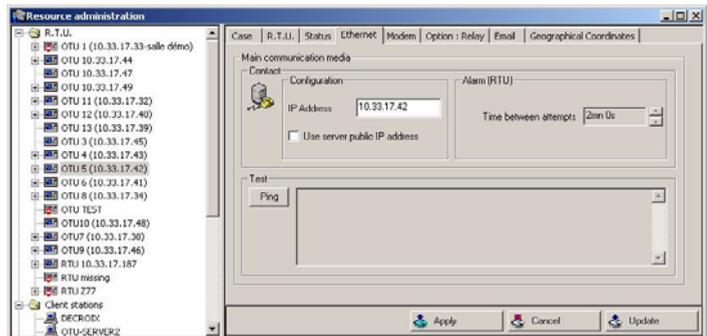


**Fig. 41**      Ethernet media

**3 Log onto the OTU, by clicking on the icon**

**4 Click on the RTU tab, then on the AutoConfiguration button.**

- The auto-configuration process is launched, and the configuration of the OTU selected displays.

**NOTE**

At this time of the process, the tabs corresponding to the OTU configuration and the OTU configuration detected are displayed.

OTU tabs
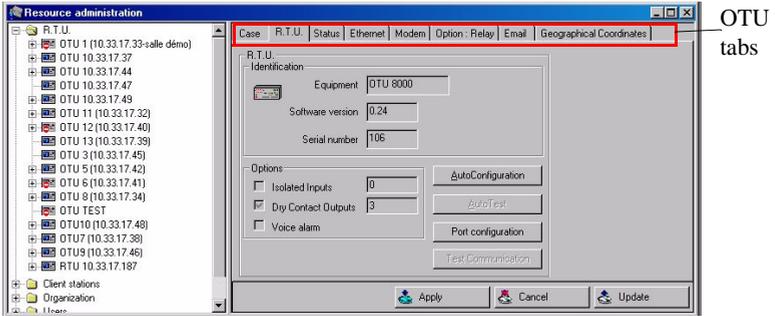
**Fig. 42**     Result after the AutoConfiguration

– To launch an autotest of the OTU, log off the RTU (  ) if connected, then click on the AUTOTEST button. Wait a few minutes to receive the report.



**Fig. 43**     Example of autotest result

– To test the communication between the 2 medias (Ethernet and Modem), click on the TEST COMMUNICATION button. Wait a few minutes to receive the following dialog box:



**Fig. 44**     Exemple of Test Communication result

The measurement equipment is often composed of:

– a OTU fitted with a measurement module (OTDR) and internal switches.
– external switches.

### Configuring the ports

The optical path between the cable head monitored and the OTU measurement module is normally entered on the level of the OTU (using the Web Interface).

However, it can be modified using the Port Description menu, displayed by clicking on the PORTS CONFIGURATION button. The process is the same as for an RTU (see "Configuring the ports" page 52).

## 5   Click on the Status tab
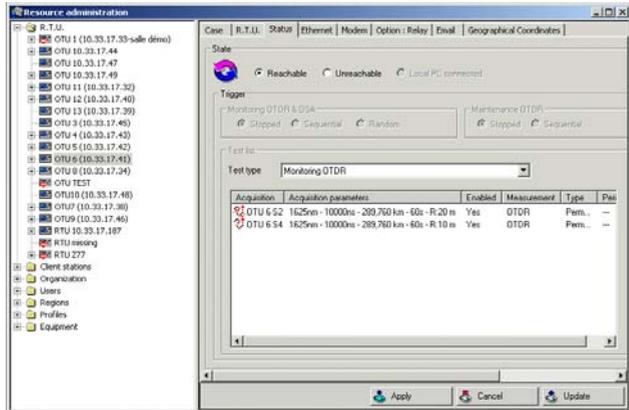


**Fig. 45**        OTU status

– Follow the same process as for an RTU, for the configuration of the OTU status (see page 53).

## 6   Click on the Modem tab

If the OTU is configured with a modem (option), this page allows to configure the backup communication for alarms notification.



**Fig. 46**        Modem configuration

– Choose None if you do not want the alarms notifications to be sent.
– Select PSTN or GSM (the type of modem is automatically detected).
– You can configure the sever phone number.

### 7 Click on «Option:relay»

This option allows to see the relays configuration. This configuration has already been set onto the OTU Web Interface but can be performed on this screen.
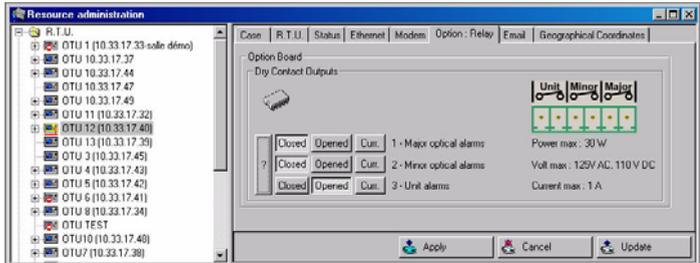


**Fig. 47** Relay Option

– Click on the icon ? to display the relay status set by default, into the Web Interface.

– Click on Opened or Closed and then on the Apply button, to modify the relay status.

   Closed : the key is not selected

   Closed : the key is selected

– Click on Curr. to go back to the initial configuration.

On the right side is available a description of the relay and the supply specifications.

If changes have been performed from the ONMS, click on the **Refresh** button into the Web Interface to apply the modification.

### 8 Click on the «Email» tab

If a PSTN modem is set into the OTU-8000, you can configure the alarm notifications, which can be sent by e-mail.

– Notch the case **Enable email notification**, in order to display the e-mail settings.

– Fill the parameters available (recipients, subject etc.)

– According to the SMTP server configured, you may need to enter the Login and Password, to access to the e-mail.

– Test the e-mail configuration by entering  message (not mandatory) and by clicking on the Test button.

– Check the message has been received by the person(s) entered in
the Recipient(s) address.
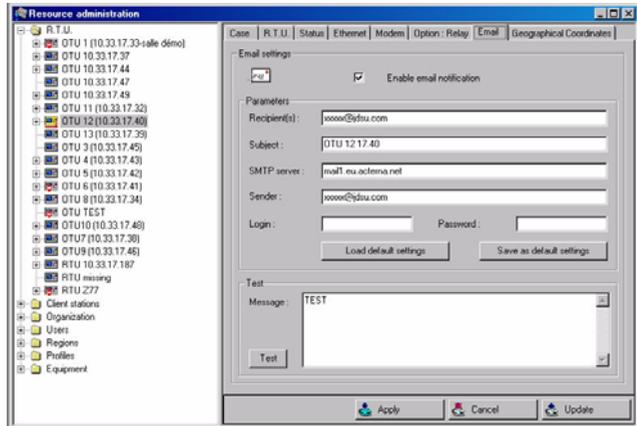


**Fig. 48**     E-mail configuration

**9   Click on the «SMS» tab**

If a GSM modem is set into the OTU-8000, on option, you can configure
the alarm notification, which can be sent by SMS toward a mobile phone.

**NOTE**

The notification by SMS in case of alarm can only be available when
the server is unreachable.
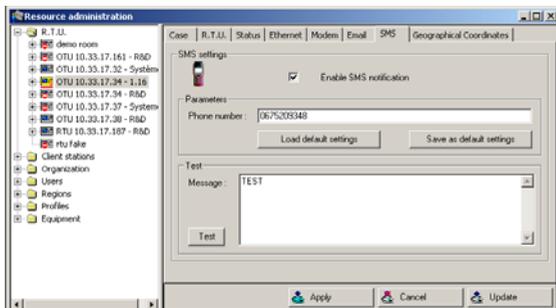


**Fig. 49**     SMS configuration

**1**   Select **Enable SMS notification** in order to receive the alarm by
SMS if the server is not reachable.

**2** In the **Parameter** field, enter the phone number on which the SMS will be sent.

– Click on **Load default settings** in order to display the phone number by default.

– Click on **Save as default settings** in order to make this phone number as the number by default.

**3** To test the SMS notification: enter the message to send via SMS (by default: TEST) and click on the **Test** button...

Wait for the dialog box informing the SMS has been sent.

Wait for the reception on the media to validate the test.

The SMS received displays: the OTU ID, its region, date and time of the SMS sending, INFO-Medium test, and the text typed previously in the line *Message* on the ONMS, in brackets.



**Fig. 50** Example SMS notification

**4** Click on Apply once all parameters are valid.

## 10 Click on the «Geographical Coordinates» tab

If necessary, enter the X, Y and Z coordinates of the OTU (see figure Figure 35 on page 54).

## 11 Click on the Apply 🔺 button to save the information entered for the OTU in the database.

The OTU is configured.

After auto configuration of the OTU, the OTU directory contains the MODULES directory, containing the identification page for the module(s) fitted in the OTU. and the OTAU directory, containing the identification page for the switches fitted in the OTU.

This page indicates the type of module, its position in the OTU, its serial number and:

– in the case of the OTDR module: its calibration parameters.
– in the case of an OTAU, its specifications

Comments can be entered for both modules.

See Figure 37 on page 57 for the identification of the OTDR module.
See Figure 39 on page 57 for the identification of the OTAU module.

### Remote Optical Switch

This configuration is useful for high density fiber network, such as access network.

The ROTAU is an OTU-8000 without the OTDR module(s).

The standard communication between the OTU and the ROTAU(s) is made over the TCP/IP network.

– Up to 15 ROTAUs can be controlled from the OTU-8000, when different fibers are used.
– Up to 512 links can be monitored from the OTU-8000.
– Up to 4 ROTAU can be cascaded when one single fiber is used.



Optical Test Access Path

OTU-8000

ROTAU

TCP/IP Network

ROTAU: OTU-8000 without OTDR Module
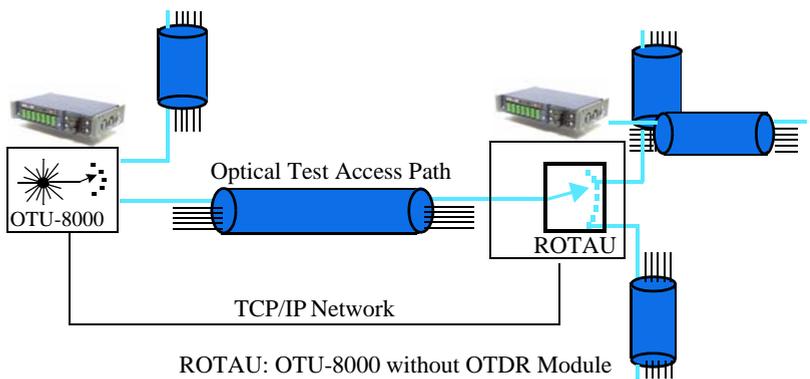
**Fig. 51**    Description Remote Optical Switch

# Adding an equipment

Each equipment intervening in the DWDM network is defined by its designation, model, make, geographical localization, and location inside the system (bay, shelf, board).

In order to facilitate equipment creation, equipment templates can be created.

**Creating an equipment template**

– Open the EQUIPMENT directory: it contains the EQUIPMENT TEMPLATE subdirectory.

– Right-click this subdirectory, and select ADD.

– Under the EQUIPMENT MODEL tab, enter the parameters for the new model.

– Under the EQUIPMENT tab, the list of network equipment and their location is displayed.
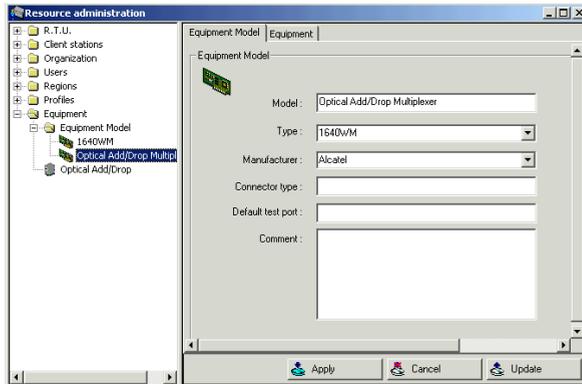


**Adding an equipment**

To add an equipment:

**1** Click on the EQUIPMENT directory. The list of equipment displays on the right.

**2** Add an equipment (see "Adding an item to a folder" page 28).

**3** Enter the equipment's designation

**4** Select the model among those created.

**5** Enter the equipment location parameters, and any comment.

**6** Click on ADD.
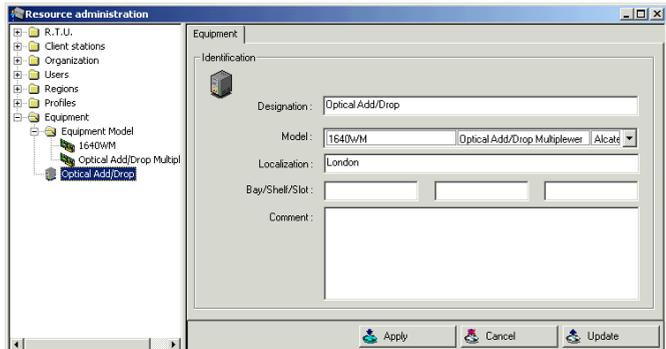
# Removing resources

**NOTE**

All the resources can be renamed or modified without using the removing phase.

**Removing procedure**

To remove a system resource in the database (RTU, ONMS client, Regions, Profiles, etc.), proceed as follows after checking that the conditions required for removing are met:

– Double-click the directory corresponding to its contents.

– Select the item to remove, and click on the ⬛ icon in the tool bar.

**Conditions required and consequences of removing**

Removing a resource is conditioned by the links it may have with other resources, as well as the alarms and links. For example, removing a profile is not possible when other users still use it.

**Removing a link**

| | |
|---|---|
| Condition | Being connected to the RTU monitoring this link. |
| Consequence | Removes all tests, alarms, plots and results associated with this link. |

**Removing an RTU**

| | |
|---|---|
| Condition | None. |
| Consequence | Removes all the links and tests from the RTU and database. |

### Removing a region

Conditions                This region must contain no RTUs nor Client
                          Stations. No users must have a profile linked with
                          this region.

When a region is renamed and not removed, the new name replaces the
old name everywhere, including for events prior the name change.

### Removing an ONMS Client Station

Conditions                None

Consequence               The station is removed only on screen. It remains
                          stored in the database. All alarms related with
                          this station are maintained.

When a Client Station is renamed and not removed, the new name
replaces the old name everywhere, including for events prior to the name
change.

### Removing a profile

Condition                 No user must have this profile.

When a profile is renamed and not removed, the new name replaces the
old name everywhere, including for events prior to the name change.

# Configuring the server

The server's configuration is accessible only to the system's general
administrator.

The CONFIGURATION / SERVER menu enables:

– entering the server's IP address

– checking the version of the ONMS software and the database.

– configuring a periodical autotest

– entering the telephone numbers of any modems linking the server to
   RTUs.

– entering the parameters related with standby (option): modem tele-
   phone number, fallback time, public holidays, etc. See
   Chapter 11 'On-duty management option".

– entering the ONMS key upon the first use of the system or after
   each configuration modification (see "Entering the ONMS Key"
   page 33).

▬▬▬▬ | **NOTE**
--- | ---
 | The **Geographical coordinates** tab allows to configure the geographical data as well as the unit to be used for the GPS: select either a display in «Degrees, minutes, seconds» or a display in «Degrees» (decimal). This choice will have repercussions in the display screens (ex: the alarm report, in an alarm descrption table etc.).

**Entering the server's properties**

In the CONFIGURATION menu, select the SERVER option. The SERVER CONFIGURATION dialog box opens on the PROPERTIES page.
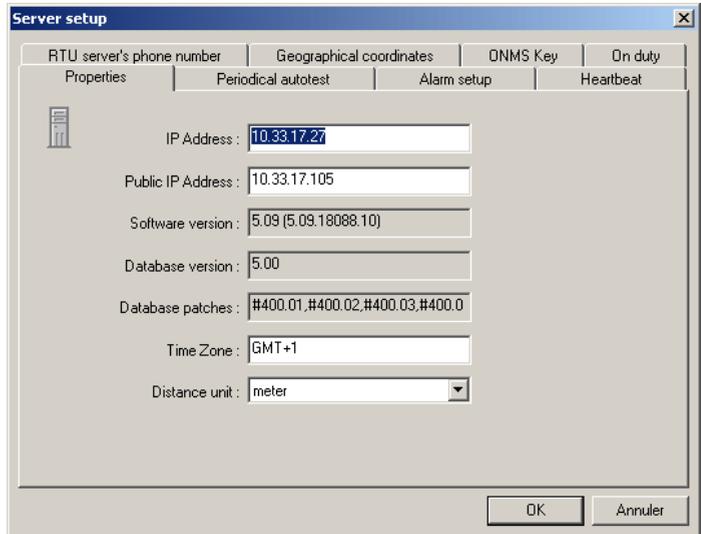


**Fig. 52**     Configuring the server

It will be possible to:

– view and/or modify the server's IP address.

– view and/or modify the Public IP Adress (used to establish communication via Internet)

– viewing the versions of the ONMS application and database, as well as the Patches applied to the database.

– entering the time zone (e.g. GMT, EDT, etc.). The latter is reminded at the bottom of the ONMS window, in the status bar, and whenever the date is printed (e.g. in the alarm report).

– entering the distance unit proposed by default by the server: meter, mile, or foot. Each client station can then choose a unit in the CONFIGURATION / CLIENT / UNIT menu.

**Entering the telephone number of the modems**

**NOTE**

This function is valid only when the RRAS service is installed on the server. If not, call your Customer Service JDSU.

In the CONFIGURATION / SERVER menu, click on the RTU SERVER'S PHONE NUMBER tab to add or modify the telephone numbers of the server's modems. Each number is transmitted to the appropriate RTU, which uses it to escalate an alarm by modem.
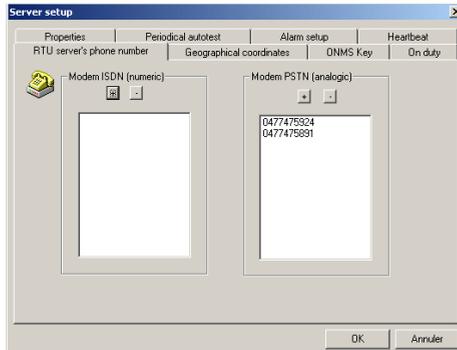


**Fig. 53**     Configuring the Phone Numbers

In the ISDN DIGITAL MODEM and RTC ANALOG MODEM columns, enter the appropriate telephone numbers.

**To enter a number**

**1**   Click on the + in the column selected.



**Fig. 54**     Adding a telephone number

**2**   Enter the number and click on ADD.

**To delete a number**

**1**   Select the number.

**2**   Click on **-**.

**On duty management option**   See chapter "Standard configuration of calendar" page 226.

# Checking the status of the system and programming a periodical Autotest

During the autotest for each RTU, the OTDR modules and the optical switches are tested on server request by the primary communication means.

It is also possible to test the backup communication means between the server and the RTU, and the communication means between the server and the Client Station.

In the CONFIGURATION menu, select SERVER then click on the PERIODICAL AUTOTEST tab.
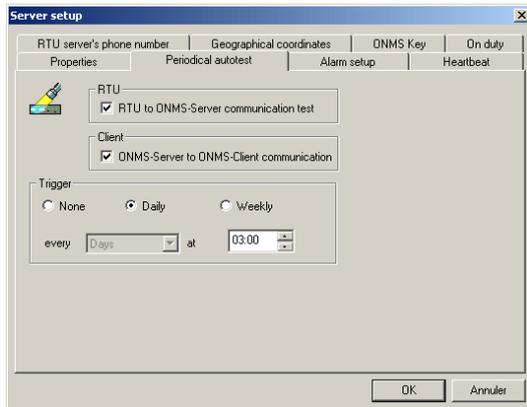


**Fig. 55**       Configuring a Periodic Autotest

**Test selection**    Check the box corresponding to the communication test required:

1. **Testing the RTU to Server communication.**

   The following operations are performed during the autotest:

   – the server connects to the RTU through its primary means,

   – if the box is checked: the RTU calls back the server using the primary communication means, then the backup communication means (if it exists).

   – the server calls back the RTU using the backup means (if it exists)

2. **Testing the Server to Client communication.**

   Here, the autotest simulates an alarm, and sends it to all AVAILABLE client stations, except to any nomad stations.

**Choosing**    – for a DAILY autotest, check the appropriate box, and enter the trigger
**periodicity**       time.

   – for a WEEKLY autotest, check the appropriate box, and enter the trigger day and time.

**Checking**    You may see in the RTU directory on the right panel, when the last auto-
**regularly the**    test actually occured.
**results of all**
**auto-tests**
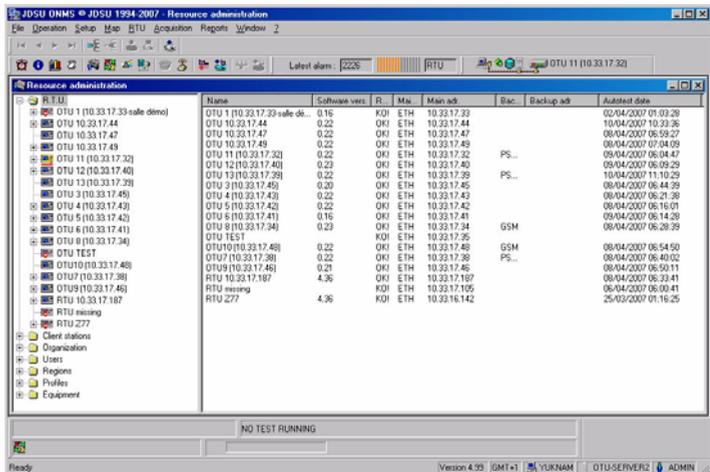


**Fig. 56**    Checking all auto-test results in one blink

You may also see in the Workstation directory the date when occured the last auto-test for each of them.

**Automatic unreachability detection (for workstations)** During all auto-tests, the server will test the connections to all client stations declared reachable. If one station can not be reached, the server will consider it as temporarily non reachable. It will test it again in the next auto-test, but shall not send it alarms if alarms shall occur. Nevertheless, the unavailable client station will receive a message once connected again, and will shortly be automatically recognized as reachable again.

# Activating the «Heart Beat» test

The Heartbeat test is used for periodical checking that the RTU can be joined. It operates only using the Ethernet primary communication means.

To activate the function, in the CONFIGURATION / SERVER menu, select the HEARTBEAT tab, then:

– Click the ENABLED button.

– Select the testing interval, among: 5, 10, 20 or 30 minutes, 1, 2, 4, 6, 8 or 12 hours according to the number of RTUs in the ONMS system.

– Enter the number of no replies tolerated before triggering an alarm: from 1 to 5.
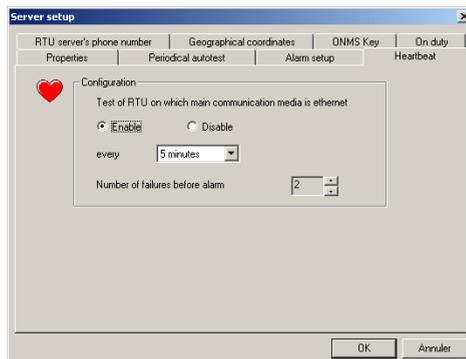


**Fig. 57**     Configuration of a Heartbeat test

# Modifying alarms management

The transfer of the cleared or acknowledged and cleared alarms to the log can be automated.

First, select in the Alarm setup menu, whether exclusively the **cleared** alarms or the **cleared and acknowledged** alarms will be moved to History.
Then, select the frequency for automatic transfer of alarms to the log in the **Trigger timestamp** parameter.

In order to limit the overloading of the database and improve its legibility, you can:

– move to history the alarms older than x days (7 days in the example)
– delete historized alarms older than x days (0 if historized alarms must not be deleted)
– program the erasing of events with the «Information» severity (log-on, test modification, etc..): delete them when they are older than x months (3 months in the example)
– number of trace to be kept per predictive test (0 if none must be stored, as in the example).
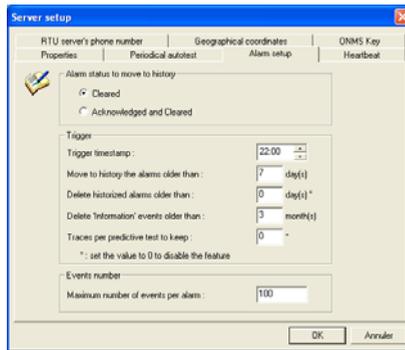


**Fig. 58**     Alarm management

In the example provided, a cleared alarm is archived automatically after 7 days, at 10:00 pm, and the system information is erased every 3 months.

In addition, in the case of the OSA measurement, an alarm can contain a number of events, and hence define the maximum number of events to be stored.

Ensure you do not program transfer to the log while saving the database.

# Configuring reports

For the OTDR measurements, the user can configure two types of reports:

– the PERFORMANCE REPORT of the service summarizing, over a given period, the number of alarms, their distribution according to severity, and their persistence. It concerns alarms transferred to the log, and not active alarms.
– the OTDR ALARM HISTORY EXPORT related with a given alarm, and describing this alarm (location, channels, spectrum, etc.): See .

These reports can be printed immediately, or savedl.

The user must configure these reports, by selecting the section and the logo(s) to be included, and their position in the report.

### Report sections selection method

To configure a report, select it in the REPORT > SETUP menu, among:

– OTDR ALARM REPORT
– OTDR PERFORMANCE REPORT

The configuration window displayed includes a left field containing the sections possible, and a right field to which the sections to add to the report must be dragged and dropped.

The user has two methods for moving sections to the right-hand field:

– Either select the sections or subsections in the left field, then click the ⟩ button.
– Or transfer all the sections, by clicking on the ⟩⟩ button and if required remove the sections selected on the right by clicking on the ⟨ button.

To remove all the sections selected, click on the ⟨⟨ key.

To save the report configuration and close the configuration window, click on the ✔ button. To close the window without saving the configuration, click on ✘.

**Logos** The default logo (on the left-hand side) is the JDSU logo. To replace it with your own logo, the network administrator must place it in the «Atlas client» folder using the filename "LOGO.GIF" .

**Report header**    The header (general information) is included to all alarm and performance reports, and no user can modify it.

### In the OTDRor OA alarm report, the header includes :

– identification of the user having launched the report

– report generation date (date, time, time zone).

– alarm date.

– alarm duration

– name of the RTU the alarm was issued by

– link concerned by the alarm

– alarm description (e.g. Injection failed)

– alarm severity (major, minor, ...)

– alarm status (acknowledged or not)

### In the OTDR performance report, the header includes :

– identification of the user having launched the report

– report generation date.

– region and list of links included (OTDR report)

– list of regions, taps, organizations and equipments included

– the observation period

– number of alarms excluded, notably those with a number of events exceeding the threshold set in alarm management (see "Modifying alarms management" page 73).

**Selecting the sections in the OTDR alarm report**

**Document**

– LOGO: By default, the JDSU logo is located on the left. To replace it with your own logo, locate it in the «Atlas CLIENT» directory, under the name LOGO.GIF.

Partial transfer    Total transfer of the sections



Close the window and save the configuration    Cancel the configuration modification
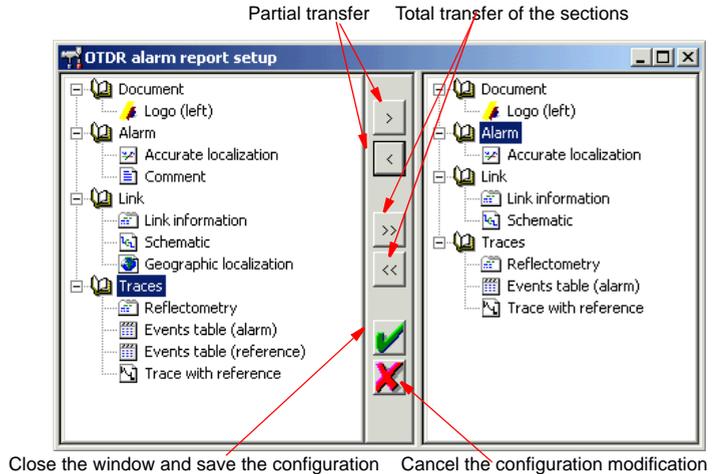
**Fig. 59**     Example of configuration of an OTDR alarm report

**Alarm**
– ACCURATE LOCALIZATION: drawing representing the fibre, and
  providing the default location in respect to the fibre beginning (with
  attenuation due to the fault) and the upstream and downstream
  markers.
– COMMENT: comment associated with the alarm:

**Link**
– LINK INFORMATION: status (monitoring or maintenance).
– SCHEMATIC
– GEOGRAPHICAL LOCALIZATION

**Traces**
– REFLECTOMETRY: date, RTU, module, wavelength, pulse, acquisition
  time, range, resolution, index.
– EVENTS TABLE (ALARM TRACE): table specifying the characteristics of
  each event detected during the acquisition having triggered the
  alarm.
– EVENTS TABLE (REFERENCE): table specifying the characteristics of
  each event detected during reference acquisition.
– TRACE WITH REFERENCE

**Selecting the sections in the OTDR performance report**
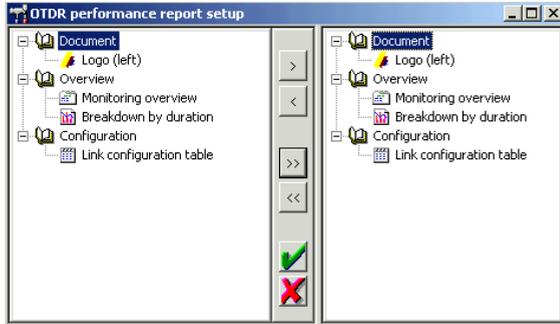


**Fig. 60**    Example of configuration of an OTDR performance report

**Document**

– LOGO: by default, the JDSU logo is located on the left. To replace it with your own logo, locate it in the «Atlas CLIENT» directory, under the name LOGO.GIF.

**Overview**

– MONITORING OVERVIEW: on the regions or links selected, and over the period chosen:
  - Number alarms per severity.
  - Alarm time per severity (percentage).

– BREAKDOWN DURATION: percentage alarms with a duration within 0 - 1h, 1- 2h, 2-4h, 4-8h, 8-12h, 12-24h, 24-48h, >48h:
  - for all alarms
  - for each severity level.

**Configuration**

– Link configuration table.

# Displaying the Events viewer (admin privilege)

All the operations performed by the system such as: logging on, adding, modifying or removing items (RTU, Client stations, Regions, etc.) as well as the events associated to the alarms are saved in the database.

To view the list of these operations, select the OPERATION > EVENTS menu or click on the 🛈 icon.

> ⚠️ **Only the "General Admin" user is allowed to view the list event list.**



Description of the event selected

**Fig. 61**      Example of events list

If an event is selected, its description displays at the bottom of the window.

The 🛠 button refreshes the display.

### Filter system events

It is often best not to display all the operations saved in the log, by applying filters to events. In order to do this, click on the 🖌 button and select events to display according to:

– original resource (RTU, Client, server).
– type of resource (OTDR, OSA)
– minimum severity,
– region of origin.
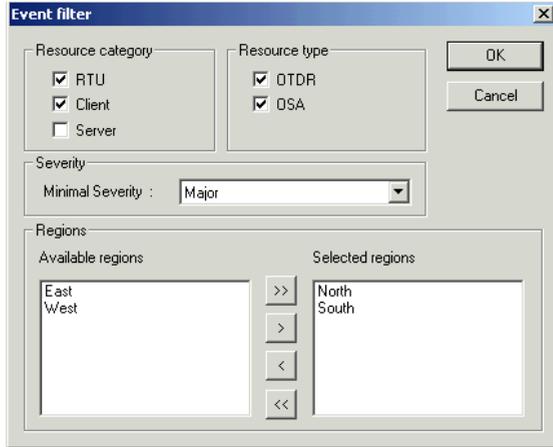
**Fig. 62** Filtering events

Events are erased automatically when an alarm is archived or when the number of events related with an alarm exceeds the threshold set in alarms management (see "Modifying alarms management" page 73).

### Detail of an OTDR and OSA alarm

The events display window includes all the events related with alarms.

To open the OTDR or OSA alarm window for an event, select it in the list, then click on the 🖼 button.

# Setting a sound for an alarm

Different sounds can be configured, according to the nature of the alarm.

In order to do this, in the CONFIGURATION menu, select the CLIENT OPTION: THE following dialogue box enables activating sounds for optical, photonic and system alarms (using the same method in all cases).
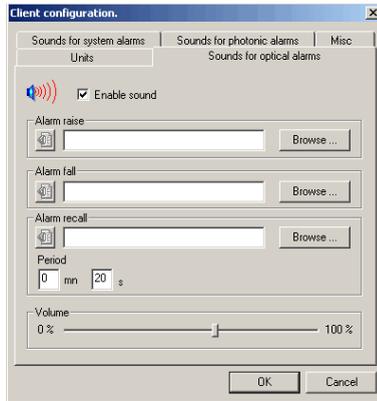
**Fig. 63** Configuring sounds

A first checkbox enables ACTIVATING THE SOUND.

When sound is enabled, it is then possible to select the 3 files matching the sounds which will be played respectively:

– for alarm escalation
– upon return to normal, when the cause of the alarm is resolved.
– periodically upon alarm, until it is acknowledged.

For each of these sounds:

– a BROWSE button opens a dialogue box enabling easy selection of a sound file in the WAV format.
– a button located to the left of the name of each sound file enables testing the sound selected.
– for the periodical beep, select the period.

A cursor enables setting the sound volume.

# Selecting the units

The ONMS can manage the following length and distance units: meter, mile, and foot.

For the OSA measurement, the user can select a different unit for wavelength and coupler compensation.

The length unit selected in the server configuration is imposed to any new client (see "Configuring the server" page 68).

However, the user can change it, in the CONFIGURATION > CLIENT menu, under the UNITS tab. This new unit is maintained, even when the unit is modified later on the server level. The unit is linked to the client station, and not to the user profile.

The SETUP > CLIENT menu also enables the user to select, in the case of OSA measurements, the wavelength and wavelength deviation units.
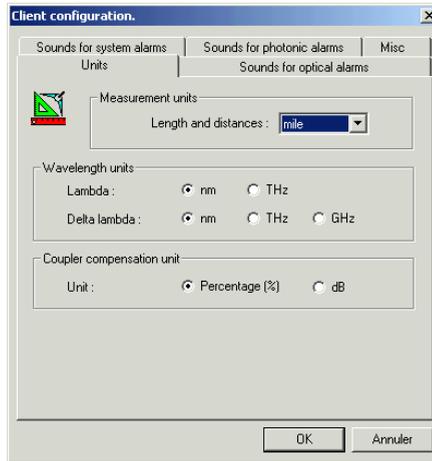


**Fig. 64**      Configuring the units

# Selecting automatic log off

The SETUP > CLIENT > MISCELLANEOUS menu also enables the user to

– modify the duration of automatic log-off.
– program automatic opening of the photonic view upon starting.

**Fig. 65**      Configuring automatic log off

# Creating a schematic

# 6

This chapter describes the tools used to modify and create the schematic of a link. It also explains how to simulate a network from the schematic.

The functions described are:

> **NOTE**
> To realize the schematics of links the advised methods are given in the section "Creating a new link" page 126.

# Description of the schematic worksheet window

When a new schematic file is selected, the worksheet window is displayed as follows:
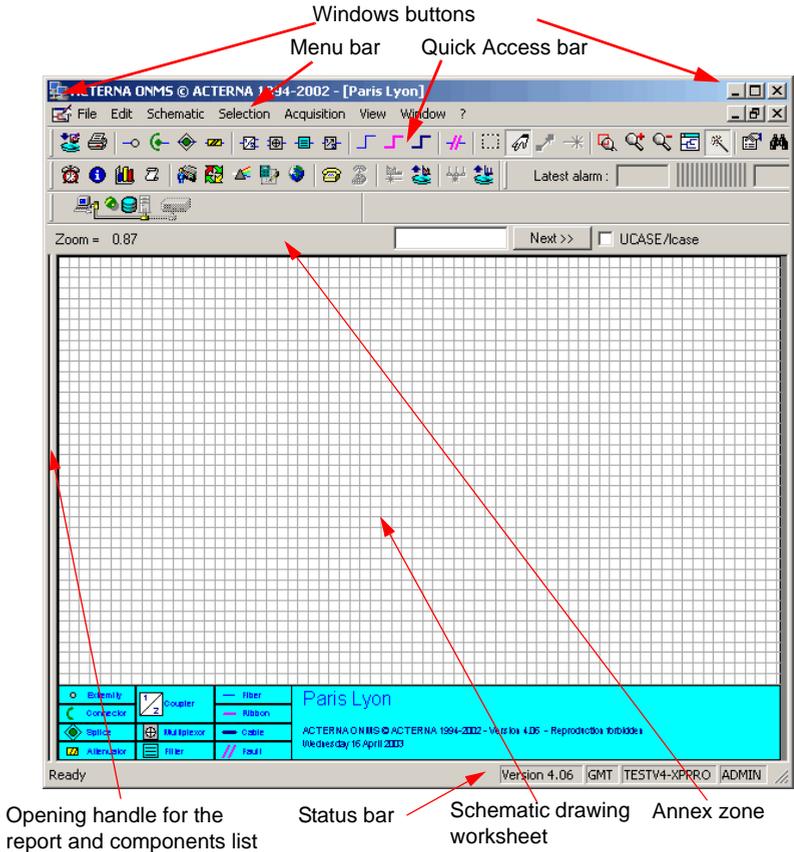


**Fig. 66** Empty schematic worksheet

The QuickAccess tool bar located below the menu bar provides shortcuts to various commands of the menus, allowing quick access to the most frequently used tools.

For information concerning the Windows buttons, refer to your Windows User Guide.

### Opening handle for the status report and component list

Click on and drag the handle to reveal the report and list of components, their names,...(see the example page 87). The **SPLIT WINDOW** option from the **VIEW** menu or the F5 key also allows you to control the handle.

### Schematic drawing worksheet

The schematic drawing worksheet is the area where the schematic is drawn. The components are placed on this sheet. A zoom factor can be applied to the sheet to facilitate reading the details, or, conversely to show all the layout. What is displayed is identical to the printout that can be made and contains an identification section showing the legend of the schematic and the main components (see the example page 87).

### Status Bar

The status bar gives the information concerning the:

– current action,
– characteristics of the component selected on the schematic,
– software version,
– name of client station,
– user name of connected user.

### Annex Zone

Three functions are accessible from this zone:

– The current zoom factor (greater than 1 for zooming in, and less than 1 when zooming out) The **ZOOM** buttons (in the right of the tool bar) control this factor directly.
– The wavelength used for the simulation. The characteristics of the different elements will be entered for this wavelength.
– To quickly access the components of the schematic: a text window for entering the name of the component, a button for moving to the next component, and a check box for choosing between upper case and lower case in the name of the component (see "Quick access using the annex zone" page 95).

## Components of an optical network

A complete theoretical optical network contains three components:

– The drawing (schematic).

– The characteristics of the components.

– The optical report and the table of measurement points.

Table of measurement points                                        Schematic drawing
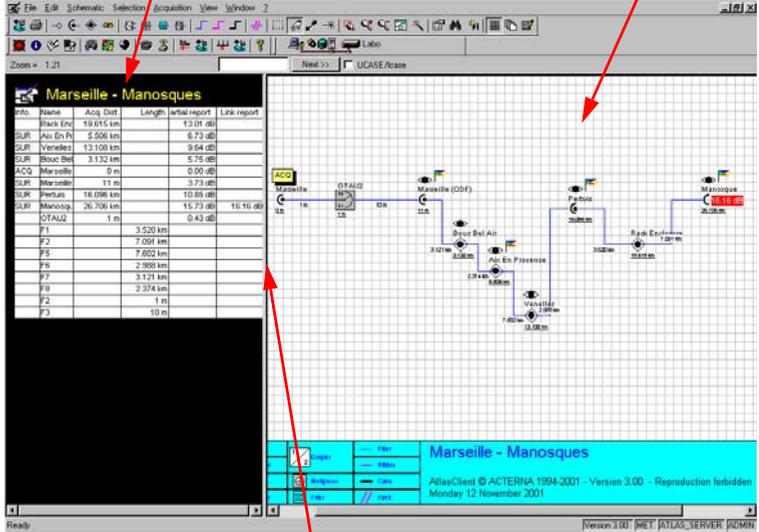


Opening the table of measurement points
(click on the left border of the schematic and slide to the right or choose the «Split window» option from the View menu, or F5)

**Fig. 67**       Example of schematic drawing and optical report

The F6 key allows the cursor to be moved from the schematic drawing to the table measurement point and vice versa.

# Creating a schematic

To create a schematic for a link, press the  button: select or create a link (see "Creating a new link" page 126), then click on the Schematic button. The schematic creation page is displayed.

You can now create a new schematic drawing worksheet or call an existing schematic which you can modify. To call an existing schematic:

– either open the schematic of the link to be used and select it (**SELECT ALL** option from **SCHEMATIC** menu) then return to the empty schematic page and paste it (**PASTE** option from **EDIT** menu).

– or open the schematic of the link to be used and choose (**SCHEMATIC**/Select all) then return to the blank schematic page and paste (**FILE** menu / Paste).

– or, from the empty schematic page, import a schematic (**IMPORT** option from **FILE** menu).

> **NOTE**
>
> To modify an existing schematic, see "Accessing the characteristics of several components" page  102)

## Recommendations for drawing a schematic

### Components definition

In the following text:

– The splices, couplers, multiplexors, attenuators, connectors, filters, faults and extremities are called elements.

– The fibers, cables and ribbons are called links.

### Error messages

Certain rules must be observed to avoid error messages being displayed (if one of these messages is displayed, click on the OK button)**:**

– Create the schematic from left to right.

– Start by positioning the elements, then connect them together using links.

– For all the components, the inputs are found on the left, and the outputs on the right.

– It is not possible to make a fiber, cable or ribbon come backwards (the output of the fiber would be at the left of its input).

## Moving around the schematic

If no mouse is available, the different menus are accessed using the <Alt + underlined character of the menu name >, then in the menu, use the cursor control keys <↑>, or <↓>, and validate by pressing <Return> on the selected option.

Once in the schematic area, the cursor can be moved using the <←>, <↑>, <→> or <↓> keys. The cursor will first move slowly and then more rapidly.

## Undo command

All the schematic creation actions can be cancelled using the **UNDO** command. This command only allows you to cancel the last edition action made. These edition actions include positioning a component and

edition of a component characteristic. However, this command does not allow you to cancel a zoom action, recover the previous value of a simulation parameter, and so on.

To cancel the last command made:

1. In the **EDIT** menu, select **UNDO**, or type <Ctrl + Z>

   The last command is cancelled and you are able to continue editing the schematic.

2. Selecting the **UNDO** command a second time returns to the initial state, in other words it cancels the first **UNDO** operation.

## Positioning a component

**1** Click on the component to be positioned in the toolbar (or use the **ADD AN ELEMENT** command from the **SCHEMATIC** menu).

   The button of the component to be positioned is shown pressed in.

**2** Move the cursor on the schematic sheet.

   The cursor is shown in the form of the component selected.

**3** Click at the required worksheet location where you want to position the component. In the case of insertion of an element on a link, a window is displayed allowing confirmation of the requested insertion or modification of the position of the component on the link:
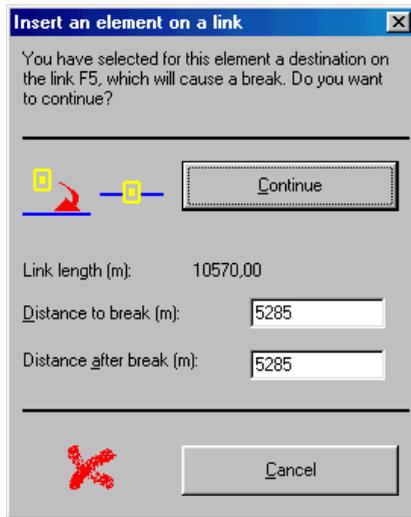


After the component has been positioned, the cursor does not change its form.

**4** Move the cursor to the required position then click to position other components of the same type.

**5** When you have finished positioning components of the same type, click on the button of another type of component or any other button.

A default name is applied to each component, along with its default characteristics. This information can be modified at any time by the person creating the network in order to give meaningful names (for a description, see page 103). To move a component see "Manipulating the components" page 94.

| | |
|---|---|
| Extremity button...................................... | ⊸ |
| Connector button ..................................... | ⟨• |
| Splice button........................................... | ◈ |
| Attenuator button.................................... | ▱ |
| Coupler button ........................................ | ⊠ |
| Multiplexor button .................................. | ⊕ |
| Filter button ........................................... | ▤ |
| Optical switch button............................... | ⊠ |

The position of these components on the schematic isn't necessary.

| | |
|---|---|
| Spare Cable button.......................... | ◯◯ |
| Fiber, Ribbon, Cable button .............. | ⌐ blue,  ⌐ purple,  ⌐ dark blue |
| Fault button .................................... | ⫽ |

### Positioning a link

Positioning a link implies that the elements have already been positioned so that they can be connected.

#### Positioning a fiber

**1** Click on the Fiber button  ⌐

**2** Move the cursor onto the schematic worksheet.

The cursor takes the form of a pen ✏️ .

**3** Point to the source element.

The element is displayed red.

**4**  Click on the element while it is red without releasing the mouse button.

**5**  Move to the destination element situated to the right of the source element.

The fiber is represented as a dotted line that follows the cursor.

**6**  Point to the destination element and when it is shown red release the mouse button.

The two elements are now connected with a fiber.

**7**  Repeat steps 3 to 6 until all the other elements to the right have been connected.

### Positioning a cable or a ribbon

Cable button ..........................................    (dark blue)

Ribbon button .......................................    (purple)

On the toolbar and on the schematic, the cable is shown dark blue and the ribbon purple.

**1**  Click on the cable or ribbon button.

**2**  Move the cursor onto the schematic worksheet.

The cursor takes the form of a pen   .

**3**  Click and hold down the mouse button at the position you want the cable or ribbon to start. This position must not be on an element.

**4**  Move the cursor, towards the right, to the position where you want the cable or ribbon to finish.

The cable or the ribbon is represented as a dotted line that follows the cursor.

**5**  Release the mouse button.

**6**  Define the number of fibers of the cable or ribbon (in its characteristics box) and connect the extremities of each fiber of a cable (or of a ribbon) to an element with a fiber.

The cables and ribbons are by default made up of 10 fibers. This number can be modified in their respective dialog boxes.

Repeat this operation for each extremity of each of the fibers of the cable or ribbon.

During the positioning of the fibers at the extremities of a cable (or ribbon), before releasing the mouse button at the end of the cable, make sure that the cable is shown red.

**NOTE**

ONMS has a routing function that avoids links intersecting during the creation of a cable, ribbon or fiber.

**Creating a group**

Group button ........................................ [⬚]

**NOTE**

The groups are not saved. They are used for editing.

A group can be placed with the **SCHEMATIC** menu **ADD A GROUP** command. This does not mean positioning an optical component but drawing a frame around several components. To add a group:

**1** Select Add a group.

**2** Click and drag on the part of the schematic that contains the components you want to group.

A rectangular dashed frame follows the movement of the cursor.

**3** Release the mouse button.

The group is bounded with a dashed rectangular frame with its name shown above. By default the name is **GROUP** followed by a creation order number.

**Changing the name of the group**

**1** Click on the selection tool [✎].

**2** Move to the frame of the group until it is shown as a red continuous line and double-click.

The Group characteristics dialog box is displayed to allow you to change the name and add additional comments after the creation date of the group information.

**Selecting the frame of a group**

**1** Click on the selection tool [✎].

**2** Move to the frame of the group until it is shown as a red continuous line and click.

The frame is shown has a continuous red line that flashes.

**Selecting a group and its contents**

Proceed as above but right-click with the mouse.

**Redefining the size of a group**

1   Select the group as described above.

2   Click on the move tool  or press the Ctrl key.

3   Move the cursor  to the bottom right of the group until it changes to the form of  .

4   Click and drag on the part of the schematic that contains the components you want to add to the group.

5   Release the mouse button.

# Manipulating the components

The outline of a component can be modified in order to facilitate its identification during the modification of the real optical network

Five functions are provided for the manipulation of the schematic components. Select, Move, Grid, Snap to grid and Display the names.

**Selecting a single component**

**With the Selection button**

1   Choose the selection tool  or right-click with the mouse.

2   The cursor is displayed in the form of a hand  .

3   Carefully position the cursor on the component to be moved until it is displayed red.

4   Once it is displayed red, click on the component.

The component flashes, indicating that it is possible to choose another movement tool, such as Move  , Snap to grid, etc.

**Quick access using the annex zone**

Text zone

Check box for upper case/lower case characters (Consider Case)

Next >> | □ UCASE/lcase

Button to access next component
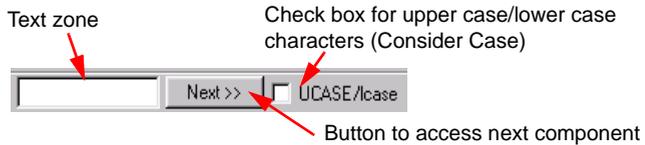
**1** Click on the text zone

**2** Enter the name of the component to find (or only the first character(s)).

**3** If necessary click on the Consider Case box.

If the box is checked, ONMS distinguishes between Upper case and lower case in the text.

**4** Click on the Next >> button to start search.

The first component found with a corresponding name is highlighted.

**5** Click on the button again to select the next component (and those that follow if you have only entered the first character(s)) until you have reached the component you require.

> When selecting a cable or ribbon, this method moves all of its fibers, even those that are not connected on the schematic (but which are displayed in the table).

**Moving a single element**

There are two methods of moving the selected component:

**Using the tool** 

**1** Click on the tool .

**2** Move the cursor onto the worksheet

The cursor takes the form of a hand with an addition sign .

Because the element is already selected, it is not necessary to position the cursor accurately on it.

**3** Click and drag the element to the position required

Use the cursor position indicator to accurately position the element.

**4** Release the mouse button.

If the element being moved is already connected to fibers (whether or not belonging to a cable or ribbon), these always follow the element during its movement.

**With the Control key**

1   When the cursor is shown as a hand  , press and hold the <Ctrl> key.

    This changes the **SELECTION** tool to the **MOVE**  tool.

2   Click and drag the element to the required position.

**Selecting several components**

1   Choose the selection tool  or right-click with the mouse.

    The cursor takes the form of a hand  .

2   Click and drag in order to form a border around the component to select.

    A rectangular border follows the movement of the cursor.

3   Release the mouse button.

    The components flashing indicate that it is possible to choose another movement tool, such as Move, Snap to grid, etc.

**Selecting the components of a group**

1   Choose the selection tool  or right-click with the mouse.

    The cursor takes the form of a hand  .

2   Move the cursor to the area of the group to select.

3   Click on the group for which you want to select the components.

4   Right click on the group.

    All the components of the group selected are displayed red and flash.

The components flashing indicate that it is possible to choose another movement tool, such as Move (using the tool  or the **Ctrl** key), Snap to grid, etc.

## Moving a link

*Moving a fiber*  For a fiber, only the joint of the fiber can be moved.

To move a joint

1   Choose the selection tool  or right-click with the mouse.

2   Move the cursor to the fiber.

3   Click when the fiber is displayed red.

4   Choose the Move tool or press and hold the <Ctrl> key.

The cursor takes the form of ➡ ,

**5** Click and move in the direction required.

**6** Release the mouse button when you have reached the required position.

***Moving a cable or*** For a cable or ribbon, it is possible to move either the entire cable (or
***ribbon*** ribbon), or the extremities, or the joint.

If the fibers that make up the cable or ribbon are already connected to the elements, the fibers automatically follow this movement.

To move a cable (or ribbon):

**1** Choose the selection tool 🖉 or right-click with the mouse.

**2** Move the cursor to the cable and click when it is displayed red.

**3** Choose the Move tool and position the cursor on the extremity of the cable or ribbon to move, or in the middle in the case of the entire cable.
The cursor takes the form of ⤵ ,

**4** Click and move the extremity in the direction required.

**5** Release the mouse button when you have reached the required position.

**NOTE**

If the cable is already linked to an element (by a fiber), it is not possible to move the extremity of the cable beyond this element (because the elements would have their inputs to the right of their outputs).

***Moving the joint of*** It is possible to modify the position of the joint of a ribbon or cable:
***a ribbon or cable***
**1** Choose the selection tool 🖉 or right-click with the mouse.

**2** Move the cursor to the link.

**3** Click when it is displayed red.

**4** Choose the Move tool 🖋 and press the Shift key,
The cursor takes the form of a double-end arrow ➡ .

**5** Click and move the joint in the direction required.

**6** Release the mouse button when you have reached the required position.

**Deleting the active selection**
When one or more components are selected, it is possible to delete them.

**1** Use the <Del> key or choose **DELETE** from the **SELECTION** menu.

The components are deleted from the schematic and are no longer shown in the optical report table.

**To deselect**
This command can be used to deselect one or several components previously selected (they are shown in red). Use any of the following to do this:

– either press the <Escape> key.
– or choose the Selection tool [image] and left-click in an empty area of the window.
– or choose **DESELECT** from the **SCHEMATIC** menu.

**To select all**
This command is used to select all the components of the schematic.

– Type <Ctrl + T> (or choose **SELECT ALL** from the **SCHEMATIC** menu). The components are selected and are highlighted.

The person creating the network can edit the schematic. This command is useful for moving all the components of the schematic on the work-sheet or for deleting all the components.

**Cut function**
This function deletes the selected item or items of the active schematic. These items are stored in the clipboard and can be pasted into the same schematic or another schematic worksheet.

**1** Select the component or components you want to cut.

**2** Select **CUT** in the **EDIT** menu.

**Copy function**
This function copies the selected item or items of the active schematic. These items are stored in the clipboard and can be pasted into the same schematic or another document.

**1** Select the component or components you want to copy.

**2** Select **COPY** in the **EDIT** menu.

**Paste function**  This function allows components previously copied or cut to be pasted into the active schematic or into another Windows document (Word, Excel, etc.).

Whenever components are pasted, by default they take the numbering of the initial name and increment it at each paste.

The other characteristics of the components pasted are not modified (wavelength, attenuation, reflectance, number of inputs or outputs, etc.)

**1** Select the active schematic that will receive the pasted or copied components.

**2** Select **PASTE** in the **EDIT** menu

The components are added to the active schematic and are displayed in red: they are therefore selected.

**NOTE**

The paste function looks for an empty space on the schematic worksheet in order to avoid any overlapping with previous components.

**Center function**  This function centers all the components of the active schematic on the worksheet.

– When the schematic is active, choose **CENTER** in the **EDIT** menu.

**Grid function**  The grid function allows you to display or not display the grid.

– When the ⊞ button is pressed down, a grey grid is displayed: the components already positioned will not be moved but all new components will be automatically positioned on a grid line.

– When the ⊞ button is released, the worksheet is white and the components can be positioned at any point.

**NOTE**

The ⊞ button is not displayed if the window is too small. If this is the case use the **GRID** command of the **VIEW** menu

**Snap to grid function**  This function positions all the selected components on the grid lines.

**1** When the grid is active, select **SNAP TO GRID** in the **SELECTION** menu or type Ctrl+A.

The selected components are moved to the closest grid lines.

**Creating a new schematic to current schematic function**

This command in the **EDIT** menu carries out a copy of the active schematic. The "New "schematic is called Synop followed by an order number that will be incremented for each new schematic created.

# Changing the fiber parameters

In the **SCHEMATIC** menu, the **PARAMETERS** command calls the following dialog box to modify certain parameters of the fiber (backscattering index and coefficient).
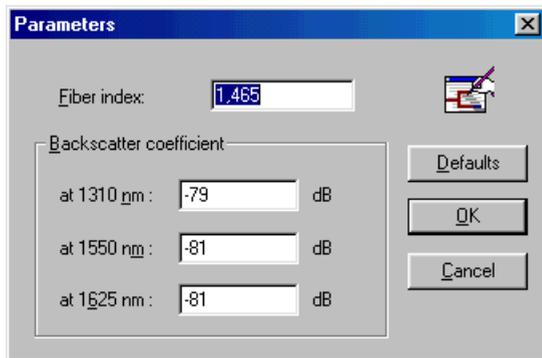


**Fig. 68**     Parameters dialog box

**NOTE**

These parameters should only be changed by qualified staff.

The **DEFAULT** button assigns default values to these parameters. Click on the **OK** button to validate the coefficients or press **CANCEL** to quit the dialog box without making any changes.

# Editing the characteristics of the components

Each component of the schematic has its own characteristics that can be modified at any time using dialog boxes. Whenever you add a new component to a schematic, a name and default values are applied to it.

**Displaying the characteristics**

**Display in the status bar**

The characteristics of the component are shown in the status bar at the bottom of the screen whenever you move the cursor onto the component.

**Display in the Tooltip help**

If the **SHOW TOOLTIPS** function (Ctrl+I) has been validated in the **EDIT** menu, the components characteristics are shown on the schematic in a yellow rectangle when the cursor is on the component.
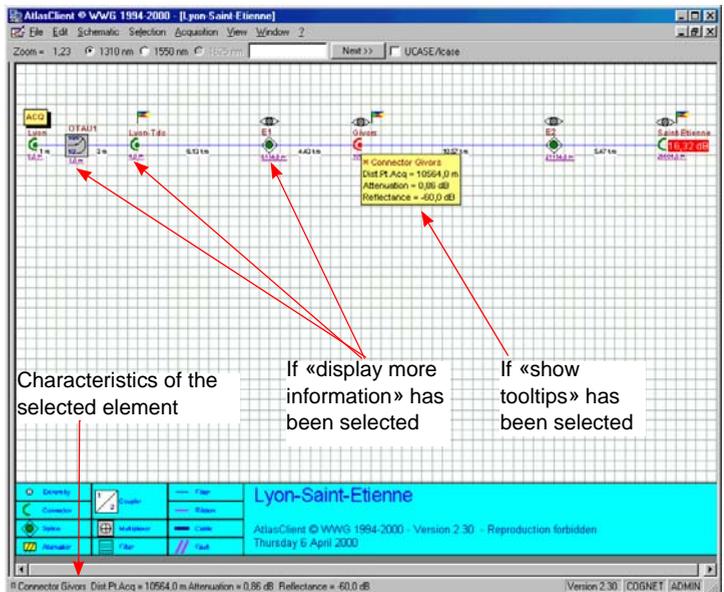


**Fig. 69**    Displaying characteristics

**Accessing the characteristics of the components**

The characteristics of the components of the schematic can be accessed either from the schematic or from the measurement points table.

**1**   from the schematic, double-click on the component: the characteristics dialog box of the component opens.

**2**   from the measurement points table (obtained by clicking and dragging on the left border of the schematic): double-click on the row for the component of which you want to modify the characteristics. The corresponding characteristics dialog box is displayed.
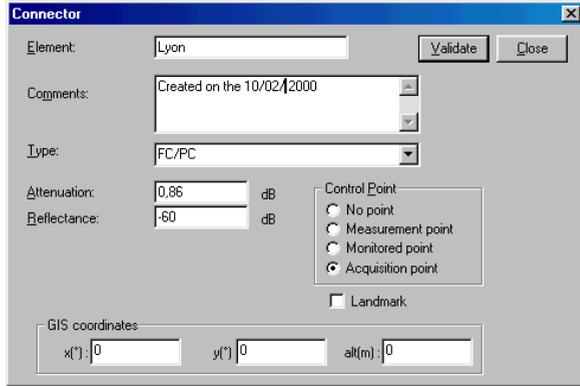
**Fig. 70**    Example of characteristics dialog box

The component selected by a double-click is zoomed-in on the schematic. If you don't want this zoom on the component, select the **FULL SCREEN AUTO MODE** in the **VIEW** menu or using the [icon] button.

**Accessing the characteristics of several components**

–    Select the components by drawing a frame around them (see "Selecting several components" page 96), or use the command Ctrl+T to select all the components of the schematic.

–    Call the characteristics definition dialog box for the first component using the command Ctrl+W (**SELECTION** menu, **CHARACTERISTICS** command).



Calls the successive
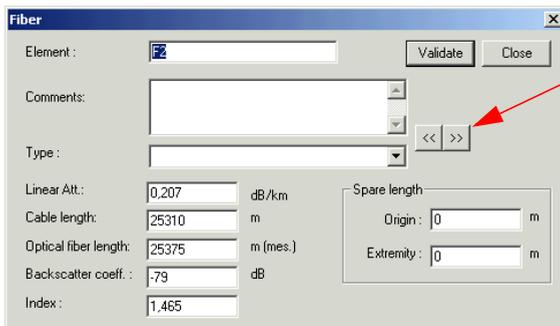
**Fig. 71**    Example of successive selections in characteristics box

– To move to boxes of the next components and links, click on the $\boxed{>>}$ button.
– To return to the characteristics of the previous elements, click on $\boxed{<<}$.

**Component characteristics**

The characteristics to enter are the following (some are specific to certain components):

For most of the components (connectors, splices, attenuators, faults, multiplexors, couplers, fibers, filters, etc.):

– **ELEMENT**: name of element.
– **COMMENTS**:any amount of text.
– **TYPE**: enter it (max. 10 characters) or select an option in the popup list.
– **ATTENUATION**: in dB.
– **REFLECTANCE**: in dB (e.g.: -35).
– **CONTROL POINT**: Choose from: **NO POINT/MEASUREMENT POINT/ MONITORED POINT/ ACQUISITION POINT**). See the section "Control points" page 105).

**For all the elements except links:**
– **LANDMARK**:Check this option if the component should be highlighted on the schematic view of the network. The landmarks serve as reference points for localizing faults.
– **GPS COORDINATES** GPS coordinates of the component: altitude in the chosen distance unit.
  Latitude and longitude must be entered in decimal degrees. For example, 10 and a half degrees, enter 10.5 and not 10.30.

**For a fiber:**
– **LINEAR ATTENUATION**in dB/km
– **CIVIL WORKS LENGTH**in the chosen distance unit
– **OPTICAL LENGTH**measured value displayed automatically after acquisition; estimated value (construction length) displayed before acquisition.
– **BACKSCATTER COEFFICIENT**
– **INDEX**
– **COILED LENGTH**length of the coiled fiber at the origin and end of the fiber (in the chosen distance unit).

### For a cable or ribbon:

Choose the number of fibres and enter the information for each fiber: click on **FIBERS**... in the dialog box. The characteristics box for the first fiber is displayed. The fiber is identified by its name; fill in the fields.

The Apply Globally button applies the same attenuation to all the fibers of the cable or ribbon.

Validate each modification so that it is taken into account.

### For a multiplexor:

– **ATTENUATION AND REFLECTANCE** for each channel.

The Apply Globally button applies the same attenuation and reflectance to all the channels.

Validate each modification so that it is taken into account.

### For a coupler:

– Number of inputs or outputs: click on the + or - button.
– Attenuation and coupling rate for each branch.

The **APPLY GLOBALLY** button applies the same attenuation and coupling rate to each branch.

Validate each modification so that it is taken into account.

### For an optical switch:

– Attenuation.
– Reflectance
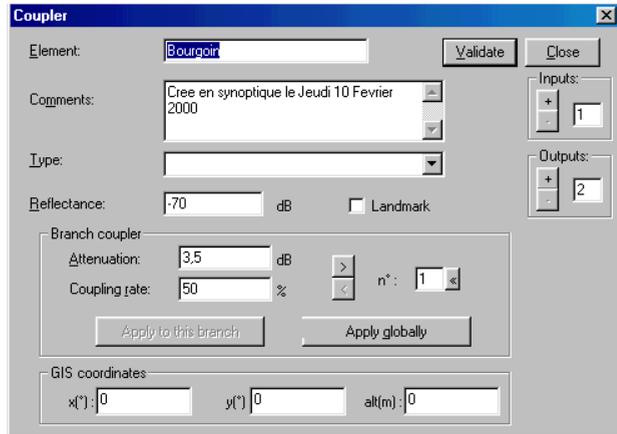
**Fig. 72**      Example of component dialog box

**Control points**  Applying a control point characteristic to an element makes it special on the schematic.

### No point

The status of this element will not be controlled during the use of the network.

### Measurement point

In the schematic  the element is accompanied by the attenuation of its branch and the $\triangleleft$  symbol.

During processing, the optical characteristics at this point will simply be recorded.

### Monitored point

This element will be monitored and the optical characteristics compared to reference values. Too big a difference on this element causes an alarm.

On the schematic the element is accompanied by the $\langle\bullet\rangle$  symbol.

### Acquisition point

Indicates that an OTDR is connected at this point

On the schematic the element is accompanied by the 【ACQ】 symbol.

The <Ctrl-E> key combination deletes this acquisition point.

On the other hand, the <Ctrl-Q> key combination defines the selected point as an acquisition point.

Click on **VALIDATE** or use the **ENTER** key so that modifications are taken into account.

Click on **CLOSE** to exit without modifying the component.

# Display tools

In addition to the display tools described below, the network designer also has the functions provided by Windows™, i.e. the display of different views, and the cascade and tiles displays.

During the use of zoom, the cursor changes into a 🔍 when it is on the worksheet.

Whenever the maximum or minimum zoom values are reached, the corresponding buttons are displayed in grey indicating that the zoom limit has been reached.

**Zoom in** The 🔍 button or **ZOOM IN** command of the **VIEW** menu or <+> key.

This tool makes a zoom centered on the displayed section of the schematic.

**Zoom out** The 🔍 button or **ZOOM OUT** command in the **VIEW** menu or <-> key.

This tool zooms out centered on the displayed section of the schematic and displays a larger area.

**Customized zoom** The 🔍 button or **CUSTOMIZED ZOOM** command in the **VIEW** menu.

This tool makes a zoom on the selected area of the schematic.

1 Select the customized zoom tool.

2 Position the cursor on the schematic worksheet.

The cursor takes the form of a magnifying glass.

**3** Click and drag over the area of the schematic you wish to zoom in on.

A rectangular border follows the movement of the cursor.

**4** Release the mouse button.

The area contained within the rectangle zooms to the maximum size for the area provided by the display.

## Zoom unit

**ZOOM UNIT** command in the **VIEW** menu or using the <0> key (on numeric keypad only).

This option cancels the zoom and returns its value to 1.

## Full screen auto mode

The ![button] button or the **FULL SCREEN AUTO MODE** command in the **VIEW** menu or <F2 > key.

In this mode:

– the schematic worksheet (with the legend) will be automatically resized to occupy the available area of the schematic window when it is modified.

– the components will not be zoomed-in when selected by a double-click.

## Full screen page

The **FULL SCREEN PAGE** command in the **VIEW** menu or the <F3> key.

The schematic worksheet (with the legend) occupies all the present window.

## Full screen schematic

The ![button] button or **FULL SCREEN SCHEMATIC** command in the **VIEW** menu or the <F4> key.

The schematic occupies all the schematic window: the legend can be removed.

## Split window

The **SPLIT WINDOW** command in the **VIEW** menu or <F5> key.

This option grabs the access handle of the report (when the computer is not equipped with a mouse).

In this case use the following methods to access the report:

1 either select the **SPLIT WINDOW** command, use the <←> and <→> keys to move the handle to the required position, and validate using the <Enter> key,

2 or click and drag the left edge of the schematic window until the required position is reached.

**Next view** The **NEXT VIEW** command in the **VIEW** menu or <F6> key.

This option activates either the schematic area or the optical report area (when the mouse cannot be used). If the optical report area is active, all the toolbar buttons are unavailable.

To access each of the components of the report.

1 Select the **NEXT VIEW** option.

2 Use the <↑> and <↓> keys to reach the required component.

3 When the component is reached, press the space bar to select the required component.

The row corresponding to this component is shown in red, the component on the schematic corresponding to the selection is also shown in red and flashes.

**Slider** This command in the **VIEW** menu allows scrolling through the schematic in the window if only a part of the schematic is displayed after a zoom action. To do this:

1 Select the **SLIDER** command in the **VIEW** menu.

2 Press the <Alt> key then one of <↑>, <↓>, <←> or <→> keys.

**Center** Select the **CENTER** command in the **EDIT** menu or <Ctrl Y>.

This command centers the schematic in the worksheet.

**Toolbar** This option displays or hides the tool bar (below the bar menu).

**Status bar** This option displays or hides the status bar (at the bottom of the window).

**Preferences...** The **PREFERENCES...** command in the **EDIT** menu allows the selection of the various displays possible for the active schematic (currently …) as well as for new schematics that will be created (INI file):

– display Tooltip help for the elements
– display more information
– display the positioning grid
– display the simulation execution progress

In the **SCHEMATIC PREFERENCES** menu (shown below), check the boxes corresponding to the required displays.
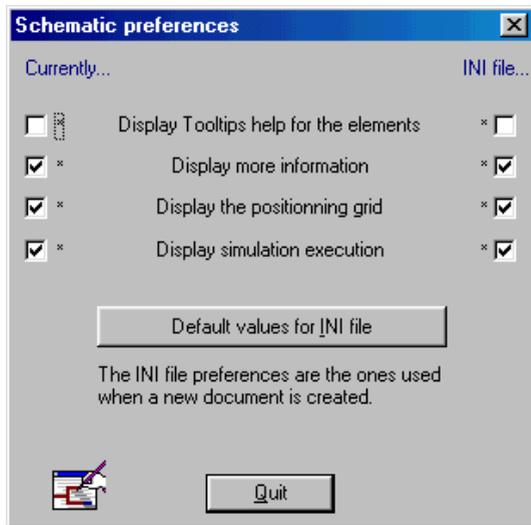


**Fig. 73** Choice of schematic display preferences

**Multipage...** The **MULTIPAGE**... command in the **SCHEMATIC** menu or <Ctrl M> allows the schematic to occupy the maximum number of pages (50 horizontal pages and 5 vertical pages).

Reducing the number of pages is not possible if the size of the schematic is incompatible with the number of pages requested.

To choose the number of pages required:

**1** Select the **MULTIPAGE** option
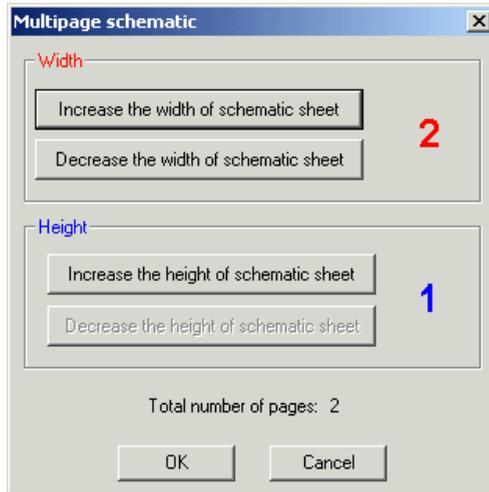
The following menu is displayed:



**Fig. 74**        Choice of number of pages for the schematic

**2**  Use the buttons to increase or reduce the number of pages from 1 to 3 in horizontal or vertical.

**3**  Validate with the OK button.

The schematic worksheet is split into the number of pages selected.

It is preferable to choose the **FULL SCREEN AUTO MODE** option (Display menu) during the transition to Multipage mode.

# Printing the schematic

**Print preview**   The **PRINT PREVIEW** command in the **FILE** menu displays the active schematic sheet as it will be printed. It could be on several pages if the Multipage… command of the **SCHEMATIC** menu has been used. Depending on the choice made in the **PRINT SETUP** menu, the schematic will be printed using either a Portrait or Landscape format.

On the worksheet the cursor is replaced by a magnifying glass     : Clicking allows you to zoom in on the schematic. Click several times to zoom to the resolution required.
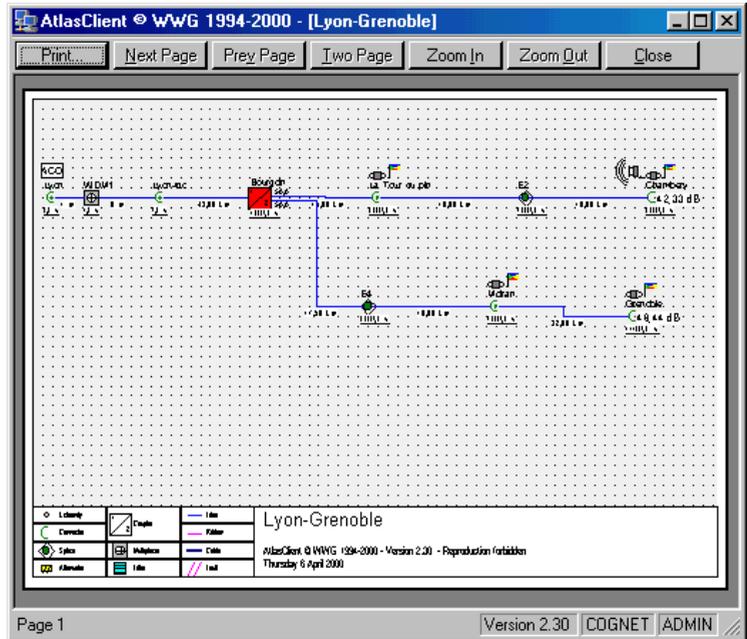
**Fig. 75**     Print preview (1 page)

The following specific buttons are available to the user.

| | |
|---|---|
| Print | Starts the printout on the printer selected using the options provided in the Print dialog box. |
| Next Page | Displays the next page if the schematic covers several pages. |
| Prev Page | Displays the previous page if the schematic covers several pages. |
| Two Page | Displays the schematic on two pages. |
| Zoom In | Zoom in on the previewed display: only possible twice. |
| Zoom Out | Zoom out on the previewed display: only possible twice. |
| Close | Returns to the schematic worksheet. |

**Example of Print preview in Multipage mode**



**Fig. 76**     Print preview (Multipage)
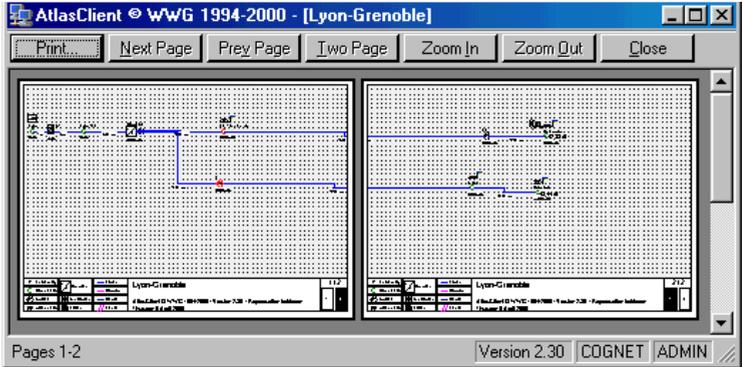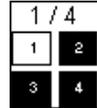
The different pages can be displayed. In the bottom right corner the pages are represented as numbered squares and the page being displayed is shown as a white square. Example: a schematic on four pages with the first page displayed:



**Printing**   **1**   Select **PRINT** in the **FILE** menu.
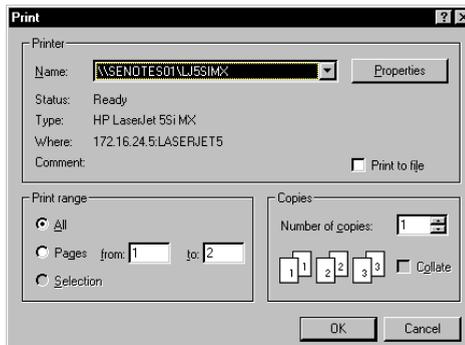**schematic**        The Print dialog box is displayed.



**Fig. 77**     Print dialog box

**2** Select the pages of the schematic you want to print if the schematic occupies several pages as well as the format of the paper in the printer:
-either all the pages (**ALL**).
- or consecutive pages identified in the menu (**PAGES**).

**3** Select the number of copies you want to print in the **COPIES** box.

**4** If the worksheet occupies several pages, select **COLLATE COPIES** to print all the pages in the correct order, page 1, then 2, …

**5** Click **OK** to confirm or **CANCEL** to exit without printing.

## Optical network report

When the network designer creates the schematic and defines the characteristics of the components, ONMS calculates the attenuation of each branch of the theoretical network.

**Accessing the optical report**
To access the report click and drag on the left border of the schematic (the cursor takes the form of ◀▶ ). The schematic window is reduced and the schematic is automatically resized to fit in the window.

A displayed table is automatically updated after each addition, deletion or modification of a characteristic of an element. Its title given in the first line is the same as the legend of the schematic.

When the table or schematic are not completely displayed, sliders allow scrolling through the non displayed areas: click on the arrow buttons of the sliders or click and drag on their position buttons.

When you click on a line of the report table:

– The line is displayed in red and the corresponding component on the schematic flashes red.

– The schematic element tools of the toolbar cannot be accessed: to access them click in the schematic zone (or press F6).

**Contents of the optical report**
The table gives the following data for each component: attenuation and distances as shown below defined by the network designer as well as the network report

– **INFO**: Type of control point.
**MES**:     Measurement point.
**ACQ**:     Acquisition point where the fictive OTDR is connected.

SURV: Monitored point.

Fiber of a cable or ribbon.

– **NAME**: Default name of the component or name give by the network designer.

– **ACQ DIST:** Distance from the OTDR to the extremity of the link.

– **LENGTH**: Total length of the link.

– **PARTIAL REPORT**:Attenuation in dB between the acquisition point (the OTDR) and the component input.

– **LINK REPORT**: Link report (attenuation) in dB between the acquisition point and the extremity of the branch of the network.

> **NOTE**
>
> Connectors of start and end of fiber are considered as «half-connectors»: to calculate the link report, the attenuation taken into account is their actual attenuation divided by two.

Optical network report                    Schematic of network



Access handle of report table

**Example**  The following simple example indicates the link between the schematic, the dialog box and the network report.

The schematic comprises of the following elements (from left to right):

| Elements | Attenuation | |
|---|---|---|
| Connector C1 | 0,5 dB[a] | |
| Fiber F1 | 0.28 dB/Km 2028 m | |
| Connector C2 | 1 dB | |
| Fiber F2 | 0.22 dB/Km 3339 m | |
| Splice E1 | 0.3 dB | |
| Fiber F3 | 0.26 dB/Km 4715 m | |
| Coupler CP1 Branch 1 (Tx of cpl=50%) | 3.36 dB | |
| Coupler CP1 Branch 2 (Tx of cpl=50%) | | 3.36 dB |
| Fiber F4 | 0.1 dB/Km 155 m | |
| Connector C3 | 0,5 dB* | |
| Fiber F5 | | 0.28 dB/Km 1730 m |
| Connector C4 | | 0,5 dB* |
| Total attenuation of branch 1 | 8.20 dB | |
| Total attenuation of branch 2 | | 8.67 dB |

a. half-connectors: their actual attenuation is divided by two (see ).

The simulated optical network report is immediately available.

The total attenuation is given on the element at the end of the optical network (use a zoom factor to easily read this information). This example shows that the result of the optical report is the sum of the attenuation of each of the components.

Schematic in Full screen mode
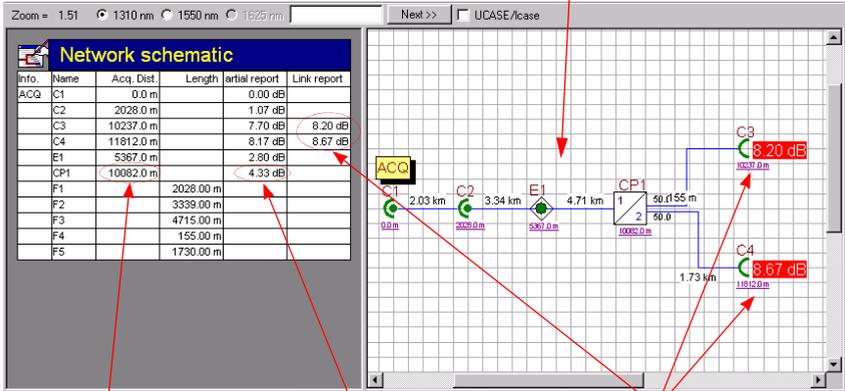


Distance between the acquisition point and the splice E1

Attenuation between the acquisition point and the input of CP1

Total attenuation of each branch in dB

# Simulation

Once the schematic is created, the simulation must reproduce the real processing conditions of the optical network.

You must therefore define certain parameters from the **ACQUISITION**/ **SIMULATION** menu.

**Configuration of the simulation**

**CONFIGURE THE SIMULATION** command, 🖳 button or <F9> key.

This function calls a dialog box allowing you to choose the acquisition conditions and the type of simulated OTDR.

**NOTE**

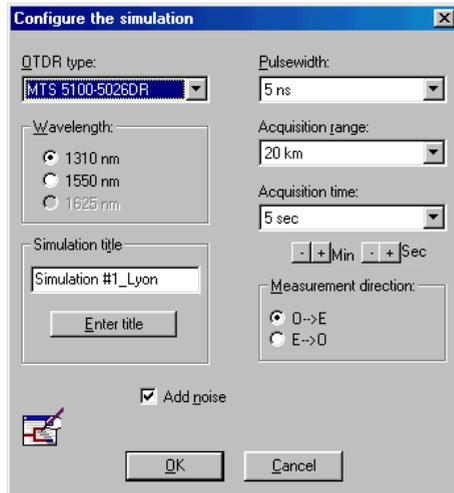If you skip this step, ONMS will not let you carry out the simulation.

**Fig. 78**　　　　Simulation configuration

Select the following options in the list boxes:

**TYPE OF OTDR**　　　　Select the required OTDR for the simulation. .

Pulsewidth range, acquisition and averaging range depend on the selected OTDR.

**WAVELENGTH**　　　　Click on the appropriate wavelength button.

**SIMULATION TITLE**　　　　A title by default is given when the operator clicks on the "Enter title" button. The network designer can choose a different title in the corresponding text box. This title will be the same as the one displayed in the title bar of the simulation result worksheet.

**PULSEWIDTH**　　　　Select the appropriate pulsewidth.

**ACQUISITION RANGE**　　　　Select the appropriate acquisition range. Make sure that this is equal to or greater than the longest schematic branch (in distance).

**ACQUISITION OR AVERAGING TIME**

　　　　The name of the box list and the range depend on the OTDR selected to carry out the acquisition. Select the appropriate acquisition time or the number of averages to be completed; or use the buttons ▁ and ▁ .

**MEASUREMENT DIRECTION** Choose the acquisition direction by clicking on the corresponding box:
- From the Origin to the Extremity (O → E).
- From the Extremity to the Origin (E ← O

**ADD NOISE** Check this box if you wish the simulation to take measurement noise into account.

**Automatic definition**

Before really installing the optical network and the measurement instruments, the **AUTOMATIC SETUP...** command of the **ACQUISITION** / **SIMULATION** menu (<F10> key) defines the most suitable OTDR for the network to be monitored. It will also define the acquisition parameters to help the network designer choose the correct instrument.
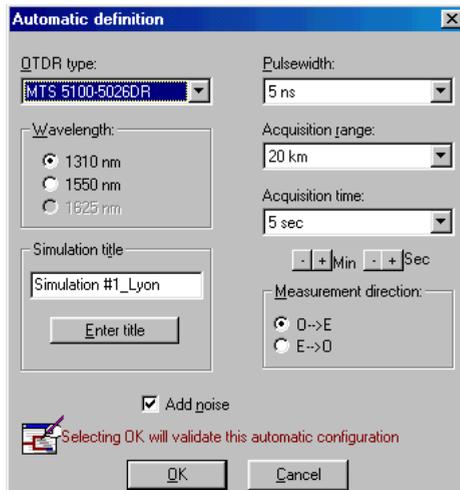


**Fig. 79** Automatic definition of correct measurement instrument

**Verification of schematic coherence**

**VERIFY SCHEMATIC COHERENCE** command of the **ACQUISITION** / **SIMULATION** menu, 🔍 icon or **F11**.

Before even starting the simulation, the network designer can verify the coherence of the network being designed.

The ONMS will then verify if the lengths and the attenuations are not equal to zero, if all the inputs and outputs of the elements are connected to a link, whether the total distribution rate is 100%, and so on.

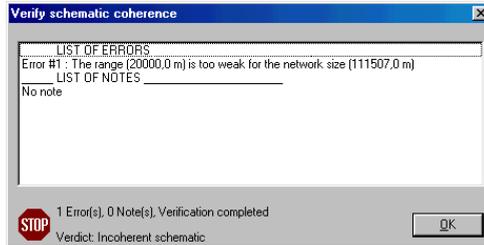At the end of the verification the following result box is displayed.



**Fig. 80**     Schematic coherence result box

If the schematic is complete, the error and notes lists are empty.

The optical network is coherent and it is then possible to start a simulation.

**Displaying the simulation progress**

Check the **DISPLAY THE EXECUTION** command in the **ACQUISITION** /**SIMULATION** menu (Ctrl D). A window showing the progress and the different steps during the simulation is then displayed to the left of the schematic.

**Starting the simulation**

The **START A SIMULATION** command in the **ACQUISITION** / **SIMULATION** menu, the [icon] icon or the <F12> key.

Once the simulation is completed, the ONMS displays a **SIMULATION RESULT** box.
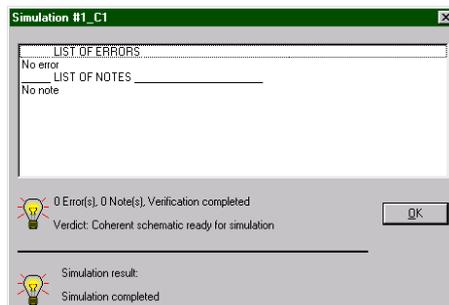


**Fig. 81**     Example of simulation result box

Click on the **OK** button to display the simulated OTDR trace and start to analyze it.

# Monitoring a link



7

This chapter describes the basic operating procedures to monitor a link using the Physical layer of the ONMS system.

Topics discussed in this chapter are as follows:

# Description of link management and operation menu

This menu is called by the SET UP > PHYSICAL LAYER command  .
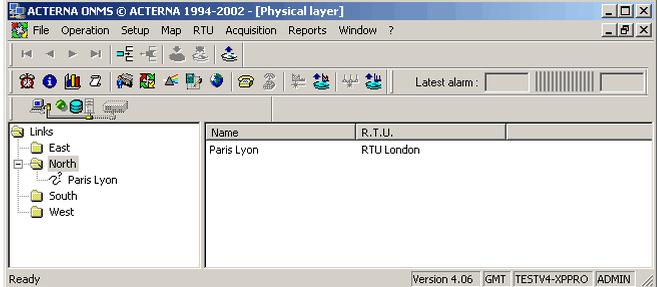


**Fig. 82** Defining a link

The left side of the window lists the folders corresponding to the regions monitored by the system.

Each folder contains the links created previously for the corresponding region. Each link name is preceded by an icon indicating the status of the link.

### Link status

The different status situations possible are shown by the following icons:

| Icon | Color | Meaning |
|------|-------|---------|
|  | Green | Normal status. The monitored link exhibits no difference with respect to its reference. |
|  | Gray | Link not monitored. |
|  | | Sequencer stopped. |
|  | Red | Intervention, construction or fault on the link. |
|  | Yellow | Monitored link in alarm state: Minor alarm (Minor threshold < current level - reference level < Major threshold |
|  | Red | Monitored link in alarm state: Major alarm (Result > major threshold). |
|  | Green | Both-end measurement. |

When a new link is created or when an existing link is selected, the right-hand side of the Operation window provides access, through a system of tabs, to the configuration pages for:

– link (name, schematic, KML configuration),
– connected material (RTU/Module, Commuter),
– monitoring tests,
– maintenance programs (OTDR measurements only),
– tests triggered by a change in status of a TTL signal (on option, OTDR measurements only).
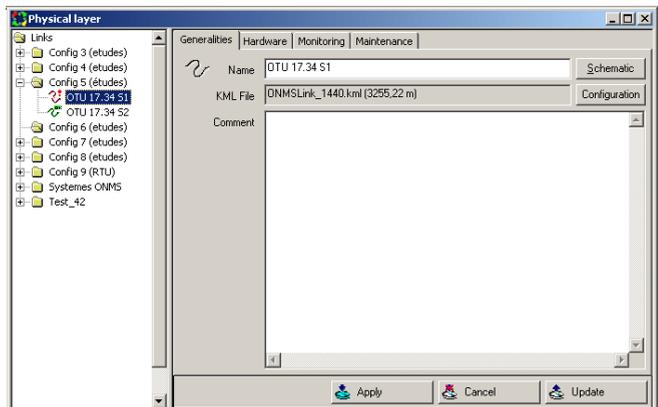


**Fig. 83**    Configuration of a link for an OTDR measurement

# Conducting an OTDR measurement on demand

A measurement on demand may be made:

– either on a clearly defined link stored in the database (known as a "declared link")
– or on a link not stored in the database ("undeclared"), by specifying the OTAU port(s) to which it is connected. This type of request makes it possible to know if a link is connected to the port(s) in question.

### 1. Connecting to the RTU

– Click on the RTU/Connection icon [icon] in the toolbar.
– Select the RTU in the RTU drop-down menu.
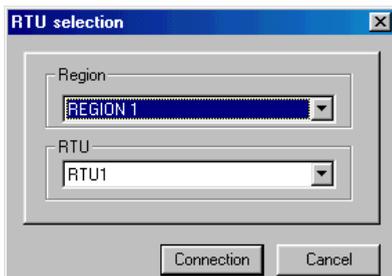
**Fig. 84**    Choice of RTU

The connection is established: In the Network Activity bar, the RTU link flashes.



**2. Choosing a link**

– Click on the icon  or select MEASUREMENT ON DEMAND in the ACQUISITION menu.

The LINK SELECTION dialog box displays the list of declared links.
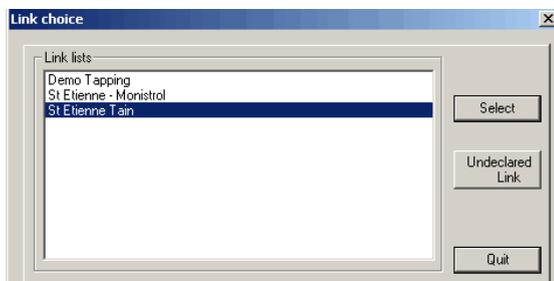


**Fig. 85**    Link selection

**To choose a declared link**

– Select a declared link then click on SELECT.

– In the ACQUISITION PARAMETERS field, select the function, the wavelength, pulsewidth, range, acquisition time and resolution (manual or automatic choice).
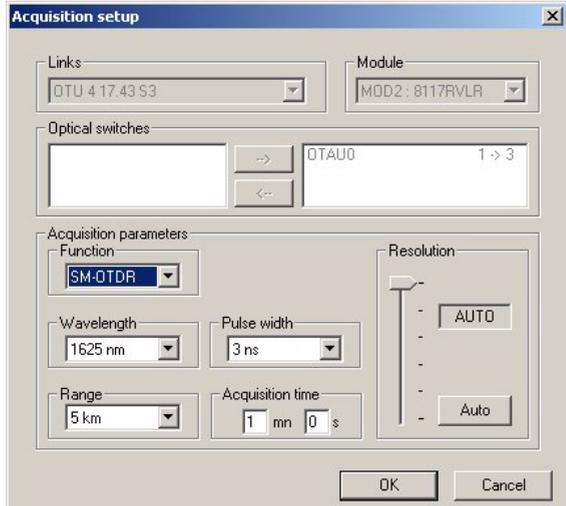
**Fig. 86** Acquisition setup for a declared link

**To choose an undeclared link**

– Click on the UNDECLARED LINK button.



**Fig. 87** Acquisition setup of an OTDR trace

– Choose the port of the RTU to which the link is connected.

– In the ACQUISITION PARAMETERS field, select the function, the wave-length, pulsewidth, range, acquisition time and resolution (manual or automatic choice).

**3. Acquisition startup**

– Click on OK. A bar graph shows the status of acquisition progress.

The Stop button stops the acquisition in progress.

When acquisition is complete, the OTDR trace is displayed.
For information about analyzing this trace, refer to Chapter 10.

# Creating a new link

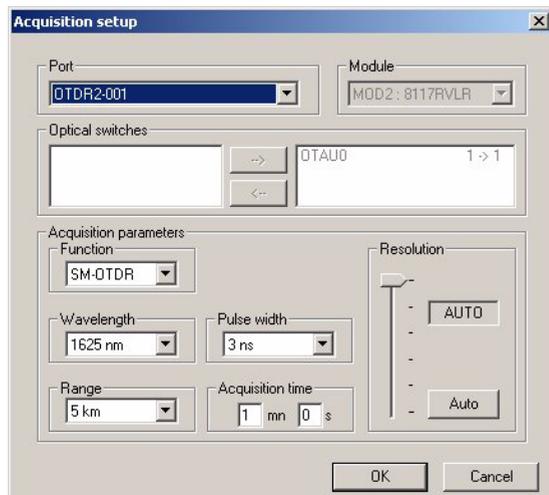Depending on the user's data type, the following procedures are possible:

– creating a new link without any data about the network,

– creating a new link from existing optical distances,

– creating a new link from existing cable distances.

You can also create a schematic using a manual method: to do so, report to chapter .

**Creating a new link without any data about the network**

If there isn't any available data about the network, it is possible to auto-matically create the schematic from the OTDR trace obtained from an acquisition. To do this, do the following steps:

**1. Press the RTU > connection icon , then:**

– Select the RTU in the RTU menu

– Click on CONNECTION.

**2. In the Operation window called by the Configuration >**

**Physical layer command  :**

– Click on the LINKS folder,

– Click on the Add icon  .

**NOTE**

Don't create the link directly in the REGION directory, but in the LINK directory. It will then be automatically moved in the region monitored by the RTU.

**3. Under the Generalities tab:**

– Enter a name for the link in the NAME field,

– You can attach a KML file, created via Google Earth[TM], and repre-
senting the localization of the link:

  – Click on **Configuration**

    The KML Association dialog box opens
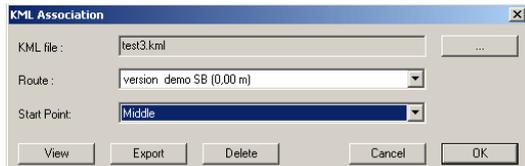


**Fig. 88** KML Association dialog box

  – Click on ⬚... to select the KML File

  – Click on Open

  – If several routes are available in the KML file, select the one in
    the list Route

  – According to the schematic, select the start point in the list

  – Once validated, the file name is displayed in the KML File field

> ⚠️ **When drawing the cable route in Google Earth[TM], draw it in the
> same sense as the fiber. For example, if the origin is at St
> Etienne and the extremity at Lyon, draw the route from St
> Etienne to Lyon (and not from Lyon to St Etienne).**

  – In the KML Association dialog box, you can Export a KML file,
    clicking on **Export** button, and saving the file on a directory (with
    kml extension)

  – Click on **Delete** to remove the kml file associated to the link

  – Click on **Cancel** to cancel the association, and return to the link
    page.

– Enter any comments about this link in the COMMENTS field.
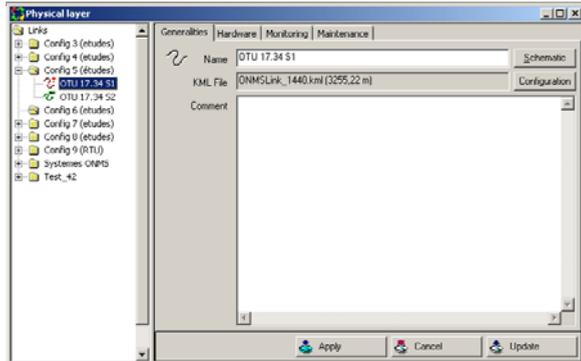
**Fig. 89** Creating a new link

**4. Under the Hardware tab, choose:**

– the RTU/OTU to monitor the link,

– the RTU/OTU port on which the link is connected.

– if an OTDR module is selected, the time slot for the RTU/OTU:

   - if the link is monitored by a single RTU/OTU, choose the FREE position: the RTU/OTU can then access the fiber as required.
   - if the link is monitored by two RTUs/OTUs placed at both extremities, then two links must be defined (for example, Saint-Etienne-Lyon and Lyon-Saint Etienne) and select a time zone that is different to both RTUs. See "Both end measurement function" page 217.

– Select / deselect the parameter **Disable SNMP traps for that link** if the generation of SNMP traps for each link must be deactivated / activated.
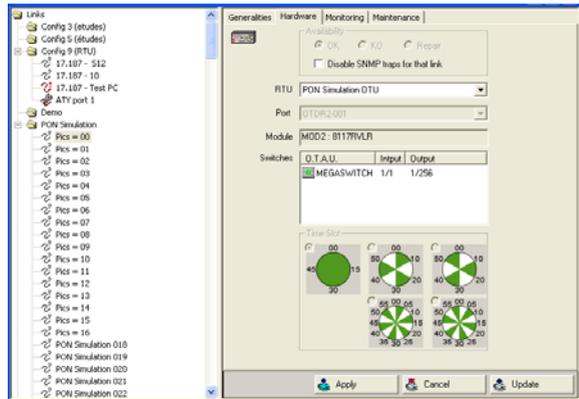
**Fig. 90** Configuring the RTU

| For OTU & RTU | | For OTU only | | |
|---|---|---|---|---|
|  | Free |  | 0 -> 4 min.<br>10 -> 14 min.<br>20 -> 24 min. | 30 -> 34 min.<br>40 -> 44 min.<br>50 -> 54 min. |
|  | 0 -> 9 min.<br>20 -> 29 min.<br>40 -> 49 min. |  | 05 -> 09 min.<br>15 -> 19 min.<br>25 -> 29 min. | 35 -> 39 min.<br>45 -> 49 min.<br>55 -> 59 min. |
|  | 10 -> 19 min.<br>30 -> 39 min.<br>50 -> 59 min. | | | |

– Click on the APPLY icon  to save information in the database. The new link is displayed in the directory of the selected RTU.

**5. To make an Acquisition on the link, do the following:**

– Click on the Acquisition icon  or select MEASUREMENT ON DEMAND in the ACQUISITION menu,

– Select the link just created from the displayed list, and then click on the Select button,

– It is recommended that the acquisition be configured as follows:

Acquisition time: 1 minute.

Range: more than twice the length of the fiber.

Resolution and pulsewidth as low as possible while, at the same time, maintaining more than 6 dB above the noise level at the end of the fiber, as indicated in the section "Reference trace" page 139.

– Click on the **OK** button to start the acquisition. A bar graph will be shown displaying the progress of the acquisition. When the acquisition is completed, the trace is displayed including markers but without any measurement results,

– Click on the icon to include the measurement results in the trace,

– Click on the icon (or select EVENTS TO MARKERS in the CALCULATION menu) to transform the events detected by the auto mode measurement in markers)

– If required, add manually other markers (refer to the Fiber Trace manual).
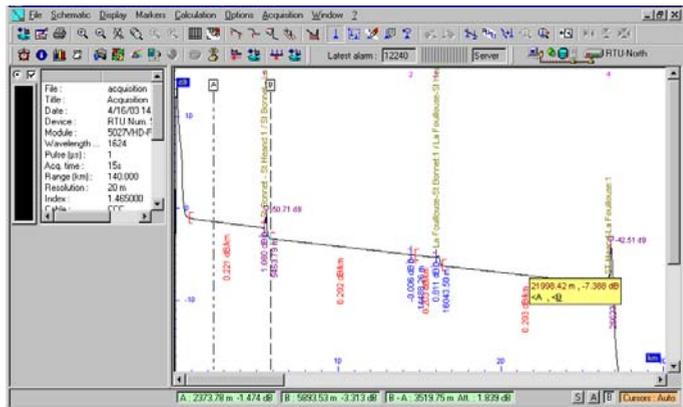


**Fig. 91**      Auto mode results transformed to markers

**6. Create the schematic, by doing the following:**

– Select the SCHEMATIC > OPEN SCHEMATIC option or click on the button,

– From the trace window, select from the Schematic option EXPORT MARKERS TO THE SCHEMATIC or click the button,

– Return to the schematic window: the schematic of the link is displayed. From here, it is possible to modify the schematic (see "Modifying the schematic for the link" page 135) and modify the trace (see "Setting up a reference trace of a link" page 138 or see Chapter 10).

**Creating a new link from existing optical distances**

If the optical distances of the network are available in a text format file, it is possible to copy them to the trace in order to automatically place the markers. To do this:

**1. Press the RTU connection icon 📇 , then:**

– Select the RTU in the RTU drop-down menu and click on CONNEC-TION.

**2. To call the Physical layer window, use the command 🔀:**

– Click on the LINKS folder,

– Click on the icon ⊒⊏ (or right click then ADD).

> **NOTE**
>
> Do not attempt to create a link directly in a region folder.
> The created link in the LINKS folder will be automatically placed in the monitored region by the RTU that it is attached to.

**3. In the Operation window, under the Generalities tab:**

– Enter a name for the link in the LINK field,

– If necessary, attach a KML file (see page 127)

– Enter any comments about this link in the COMMENTS field.

**4. Under the Hardware tab, from the drop-down menus, choose:**

– The RTU to monitor the link,

– The RTU port,

– Choose the desired Time Slot of the RTU.
  - if it is monitored by a single RTU, choose the FREE position: the RTU can then access the fiber as required.
  - if the link is monitored by two RTUs placed at both extremities, then two links must be defined (for example, Saint-Etienne-Lyon and Lyon-Saint Etienne) and select a time zone that is different to both RTUs (see "Both end measurement function" page 217).

– Click on the APPLY icon 🔽 to save the information in the database. The new link will then appear in the selected RTU folder.

**5. To make a trace, do the following:**

– Click on the icon 🔆 or select OTDR MEASUREMENT ON DEMAND in the ACQUISITION menu,

– Select the link just created from the displayed list, and then click on the Select button,

– It is recommended that the acquisition be configured as follows:

Acquisition time: 1 minute.
Range: more than twice the length of the fiber.
Resolution and pulsewidth as low as possible while, at the same time, maintaining more than 6 dB above the noise level at the end of the fiber, as indicated in the section "Reference trace" page 139.

– Click on the OK button to start the acquisition. A bar graph will be shown displaying the progress of the acquisition. When the acquisition is completed, the trace is displayed including markers but without any measurement results,

– Open the text file containing the information about the optical distances of the network.

Information about the network is contained in a table format with the following structure:

- 1st column: Optical distances from origin in meters (mandatory)

- 2nd column: Marker names (mandatory)

- 3rd column: Longitude of marker (optional)[1]

- 4th column: Latitude of marker (optional)[1]

– Select the information about the network (lines and columns) that you wish to use in the schematic and copy it to the clipboard (Ctrl+C keys or Edit / Copy menu),

– Return to the window displaying the trace,

– Right-click on the trace and select PASTE MARKER SELECTION FROM CLIPBOARD (or use the Ctrl+V keys),

| | |
|---|---|
| Cancel marker selection | |
| Copy selected markers | Ctrl+C |
| Cut selected markers | Ctrl+X |
| Paste marker selection | Ctrl+V |
| Paste and invert marker selection | Shift+Ctrl+V |
| Copy trace | |
| Copy results | |
| Copy print preview | |

The markers are placed on the trace.

– There might be a need to adjust the markers if they are not positioned correctly with the detected events on the trace.

---

1.Must be entered in decimal degrees (for example, enter 10,5 degree and not 10 degrees 30 min.).

**6. Create the schematic, by doing the following:**

– Select from the SCHEMATIC menu, the OPEN SCHEMATIC option or
click on the 🖼 button,

– From the displayed trace window, select from the Schematic option
EXPORT MARKERS TO THE SCHEMATIC or click the 🖼 button,

– Return to the schematic window: the schematic of the network is
displayed. From here, it is possible to modify the schematic (see
"Modifying the schematic for the link" page 135) and modify the
trace.

## Creating a new link from existing cable distances

The schematic of the link can be populated from a file containing the
cable distances from a table format. This file must have the following
structure:

1st column: Cable distances from origin in meters (mandatory)

2nd column: Marker names (mandatory)

3rd column: Longitude of marker (optional)[2]

4th column: Latitude of marker (optional)[1]

If this is the case, then you create a link with the following steps:

**1. Press the RTU connection icon 📞 , then:**

– Select the RTU in the RTU drop-down menu and click on CONNEC-
TION.

**2. Use the Configuration > Operation command 🔲 to call
the Physical layer window:**

– Click on the LINKS folder,

– Click on the icon 🔲 (or right click then ADD)

**NOTE**

Do not attempt to create a link directly in a region folder.
The created link in the LINKS folder will be automatically placed in the
monitored region by the RTU that it is attached to.

**3. Under the Generalities tab:**

– Enter a name for the link in the LINK field,

– If necessary, attach a KML file (see page 127)

– Enter any comments about this link in the COMMENTS field.

2.Must be entered in decimal degrees (for example, enter 10,5 degree and
not 10 degrees 30 min.).

**4. Under the Hardware tab choose:**

– the RTU,

– the RTU port on which is connected the link

– the desired TIME SLOT of the RTU.
  if it is monitored by a single RTU, choose the FREE position: the RTU can then access the fiber as required.
  if the link is monitored by two RTUs placed at both extremities, then two links must be defined (for example, Saint-Etienne-Lyon and Lyon-Saint Etienne) and select a time zone that is different to both RTUs. See "Both end measurement function" page 217.

– Click on the APPLY icon  to save the information in the database. The new link will then appear in the selected RTU folder,

– Click on the ADD button,

– Select the commuter from the displayed list and click OK

– Select which input and output switch channels the link is connected and click OK,

– Click on the APPLY icon  to save information in the database.

**5. To copy the cable distances to the schematic, do the following:**

– Under the GENERALITIES tab, click on the SCHEMATIC button to open the SCHEMATIC window,

– Use WINDOWS EXPLORER to search for the file containing the cable distances. When found, open this file,

– Select the information about the network (lines and columns) that you wish to use in the schematic and copy them to the clipboard (Ctrl+C keys or Edit / Copy menu),

– From the SCHEMATIC window, in the EDIT menu, select the PASTE MARKERS. option.

**6. To make a reference trace, do the following:**

– Click on the Acquisition icon . The link selection dialog box is then displayed.

– Select the link just created and click on the SELECT button,

– It is recommended that the acquisition be configured as follows:

  ACQUISITION TIME: 1 minute.
  RANGE: more than twice the length of the fiber.
  RESOLUTION and PULSEWIDTH as low as possible while, at the same time, maintaining more than 6 dB above the noise level at the end of the fiber, as indicated in the section "Reference trace" page 139,

– Click on the OK button. A bar graph will show the progress of the acquisition. When the acquisition is completed, the trace is displayed including markers but without any measurement results,

– In the SCHEMATIC menu, select the command IMPORT MARKERS FROM SCHEMATIC to import markers in the trace, or click the [button] button,

– The markers are placed on the trace: there might be a need to adjust the markers if they are not positioned correctly with the detected events on the trace (refer to "Setting up a reference trace of a link" page 138),

– In the SCHEMATIC menu, select the command EXPORT MARKERS TO SCHEMATIC to update optical distances in the schematic.

## Creating manually a new schematic

To create manually a new schematic entering physical and optical distances, report to Chapter 6 "Creating a schematic" on page 85.

## Modifying the schematic for the link

A simple procedure for modifying the schematic is described below. For more detailed information, refer to chapter .

**1. Position the components**

– Locate the connectors  .

– Position the splices  .

– Position the optical switch  .

– Connect the elements of the fiber. To do this, select the Fiber component  . Position the cursor on the first element to be joined: when it is displayed red, click and, without releasing the mouse button, slide to the second element. Release the mouse button when this second element becomes red.

Components
Extremity, Connector, Splice, Attenuator, Coupler, Multiplexor, Filter, Switch, Spare Fiber
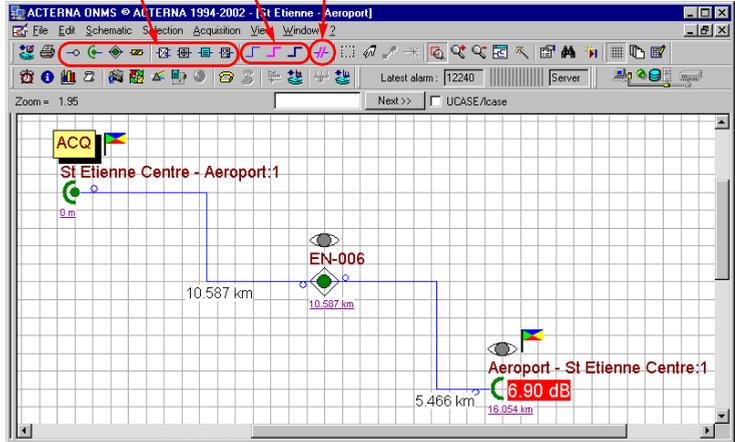
Links:
Fiber, Ribbon, Cable
Fault



**Fig. 92**      Example of a schematic

### 2. Choose the characteristics of each connector

Double-click on each connector to display its definition box.

In each case, enter the following information:

– The name of the ELEMENT corresponding to its location.

– Any COMMENTS concerning, for example, its location (exact address, floor, room n°, etc.). These comments may be displayed when an alarm is activated. By default, the date when the element was created is displayed.
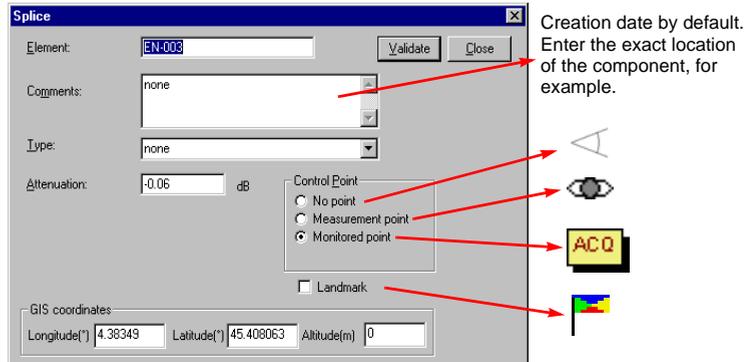
– The TYPE of element.

**Fig. 93**    Definition of a component

– CONTROL POINT (refer to "Component characteristics" page 103).

The first connector must be the acquisition point ![ACQ], that is the point where the OTDR is connected.

Since the connector which follows the switch is in the dead zone, it cannot be monitored. It must be designated as a LANDMARK ![landmark] (flagged on the map view and serving as reference for fault detection).

The last connector of the schematic must be a MONITORED POINT ![monitored], that is a point for which the characteristics are compared to a reference which may be used to trigger an alarm.

The other connectors and splices will be landmarks ![landmark] if they are easily identifiable and are monitored points ![monitored].

– Click on VALIDATE after defining each connector in the schematic.

**3. Enter the distances**

– Double-click on each link between the components of the schematic to enter its name, comments (default value is date of creation) and type.
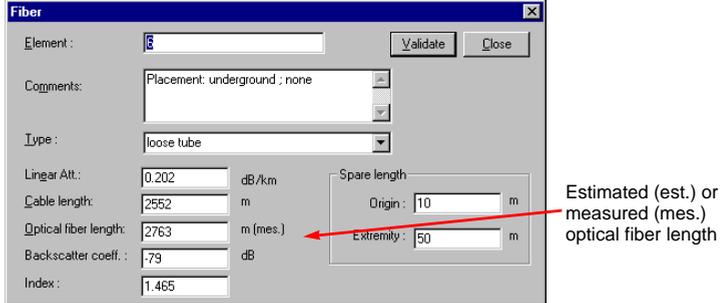
| | |
|---|---|
| **Fiber** | |

Element : [6]  Validate  Close

Comments: Placement: underground ; none

Type : loose tube

Linear Att. : 0.202  dB/km
Cable length: 2552  m
Optical fiber length: 2763  m (mes.)
Backscatter coeff. : -79  dB
Index : 1.465

Spare length
Origin : 10  m
Extremity : 50  m

Estimated (est.) or measured (mes.) optical fiber length

**Fig. 94**      Defining a link

– Enter its construction length, provided by the cable installers, or enter the optical length directly if this is known.
If the optical length is not given at validation, the construction length is taken as an estimated value and in this case is followed by the suffix (EST.). The real optical length will be automatically recorded following acquisition of the reference trace. It will then be followed by the suffix (MES.).

– Click on VALIDATE after the definition of each link.

**4. Record the schematic by using the Apply command , then close it.**

# Setting up a reference trace of a link

### 1. Making an on demand measurement

To obtain the reference trace for a link, a measurement must be made on the link, as indicated in the section "Configuration of a link for an OTDR measurement" page 123:

– Click on the Acquisition icon .
– Choose the name of the link.

It is recommended that the acquisition be configured as follows. The first reference being used for the localization, you must choose parameters giving the best possible accuracy.

Acquisition time: 1 minute.

Range: More than twice the length of the fiber.

Resolution and pulsewidth as low as possible while, at the same time, maintaining more than 6 dB above the noise level at the end of the fiber, as indicated in the section "Reference trace" page 139.

– Start the acquisition by clicking on OK.

– Modify the acquisition parameters, if necessary, and return the measurements until a satisfactory trace is obtained.

### 2. Positioning markers on the trace

The markers are positioned on the trace which has just been acquired at the theoretical distances entered on the schematic.

– Move the markers to the exact position of each event on the trace.

For a reflective event: at foot of Fresnel peak

For a non-reflective event: at peak of the event, as near as possible to decreasing front
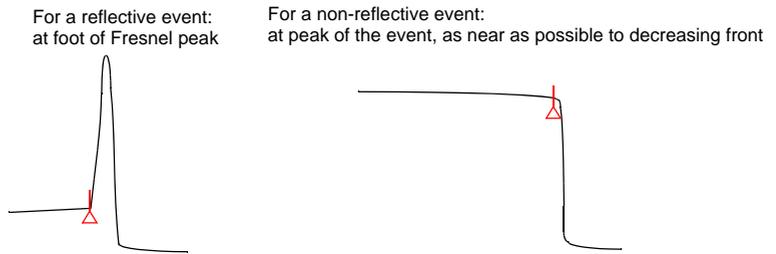


**Fig. 95**       Position of markers
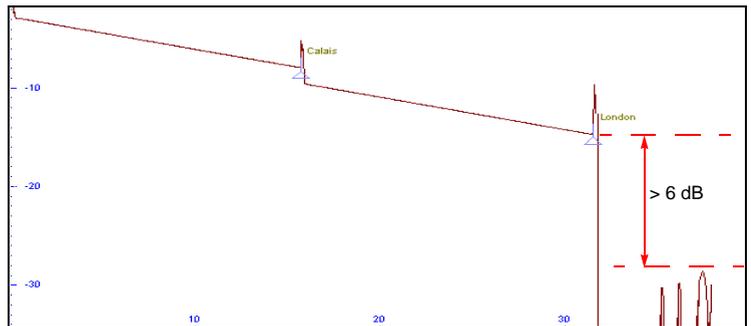


**Fig. 96**       Reference trace

– Save the reference trace (  ) without closing it.

### 3. Copy the position of the markers in the schematic

– Open the schematic (SCHEMATIC/ OPEN menu).

– Display the reflectograph by selecting its name in WINDOW menu.

– In the SCHEMATIC menu, choose the command Copy Marker to schematic.

On the schematic and in the FIBER dialog boxes, the optical lengths are then indicated with the suffix «mes.»

**4. Fixing the injection level manually**

The ONMS automatically compensates for fluctuations in injection level. This level is defined as the intersection with the vertical axis of the slope of the trace, as soon as it becomes linear, not including the dead zone, connector and splice (see example below).
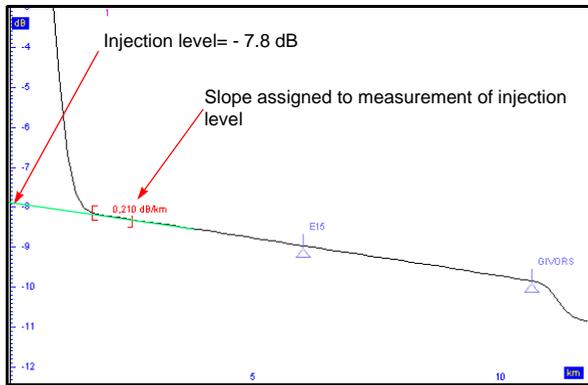


**Fig. 97**     Correction of injection level

The slope must be measured on a short zone since, if a fault appeared on this, the level of injection would no longer be correct. Its length must be chosen according to the total length of the fiber. To define this zone:

– Using the cursors, at the beginning of the trace delimit a section whose slope appears to respect the definition of the level of injection.

– Click on the slope button [⌐⌐]. The section then appears in red and its slope is displayed. It will always be used for measuring the injection level.

▬▬▬▬ | **NOTE**

If no slope was measured at the beginning of the trace, the slope will be defined by the automatic mode of trace analysis. The result may then be disturbed by optical events following the dead zone. It is therefore recommended that you use the manual mode described above.

> ⚠️ The first slope measurement on a reference trace is always automatically assigned to measuring the injection level.

# Monitoring a link

**Principle** Two reference traces are used to set up monitoring for a link. One is used for detecting anomalies, the other for locating them.

–   Reference trace for detecting anomalies

During the current acquisition, the level at the points located by the markers is compared to the level of this reference trace to detect any possible anomaly. Since this comparison is not intended to locate the anomaly, a good level of resolution is not necessary. However, given the number of fibers to be monitored, the acquisition time for each measurement must be minimum. The recommended acquisition time for producing this reference trace is 10 seconds.

–   Reference trace for locating anomalies

If comparison with the detection reference trace reveals a difference which is greater than the fixed threshold, a second acquisition is run using the parameters of the reference trace used to locate the fault. In this case, the level of precision of the location of the fault depends on the resolution. The reference trace recorded when the link was created is then used.

Several monitoring tests may be assigned to the same link.

**Defining a link monitoring test**

**1. To obtain a reference trace for detecting anomalies**

–   Make an on demand measurement on the link (see "Configuration of a link for an OTDR measurement" page 123).

–   Choose the following parameters to configure the acquisition:

Acquisition time: 10 seconds

Wavelength and range: same values as those used to acquire the reference trace.

Resolution: Auto

Pulsewidth: such that the end of the link is 6 dB at least above the noise level.

–   Start the acquisition.

– Define the calculation slope of the level of injection (refer to "Fixing the injection level manually" page 140)

– Save the reference trace thus defined by clicking on the ![icon] icon.

**2.  Defining a link monitoring cycle**

– Select the required link in the LINKS folder of the PHYSICAL LAYER window.

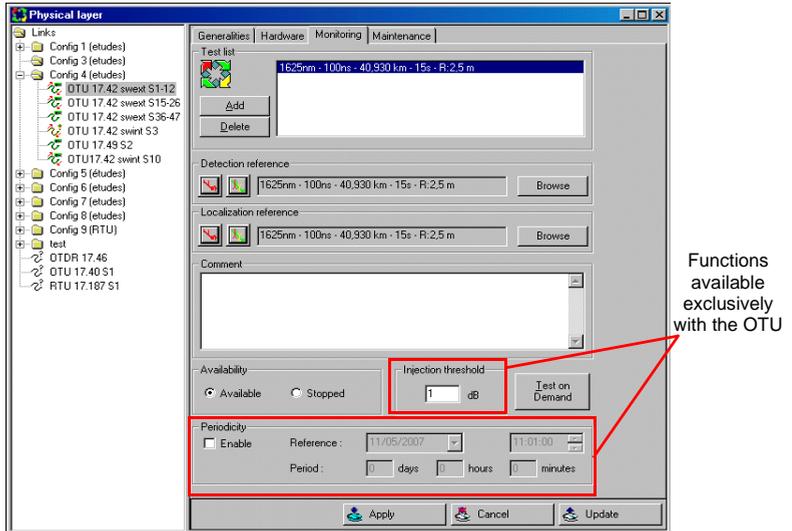– On the right of the screen, click on the MONITORING tab.



Functions
available
exclusively
with the OTU

**Fig. 98**      Monitoring setup

– Click on the ADD button.

– In the DETECTION REFERENCE field, click on the BROWSE button, and choose the detection reference trace from those which are proposed. The ![button] button (red) allows you to visualize the chosen trace.

**NOTE**

The detection trace must have a low acquisition time.

– Click on the ![button] button to fix the monitoring thresholds beyond which there will be a minor or major alarm.

- When a monitoring test is added, the next box is used to make the global choice of the thresholds for all the markers. Except for very special cases, the default thresholds of ±1dB for the minor alarms and ±3 dB for the major alarms may be kept.
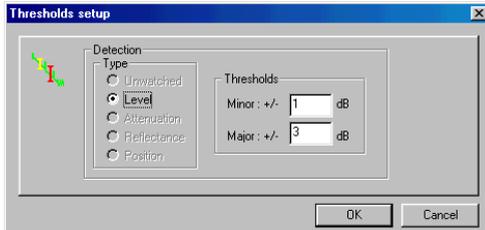


**Fig. 99**        Global choice of monitoring thresholds

- When the configuration of an existing monitoring test is modified, pressing the [icon] button calls the next box which can be used to individually choose the threshold assigned to each marker:
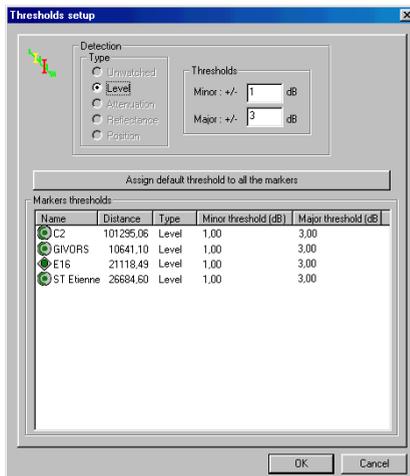


**Fig. 100**       Choice of monitoring thresholds per marker

Double-click on the marker in the list to access the choice of thresholds. Several thresholds may be modified simultaneously. To do that, select the markers (pressing down on the CTRL key), then right-click the mouse button and, in the displayed menu, select MODIFY THE THRESHOLDS.

– In the LOCALIZATION REFERENCE field, click on the BROWSE button and choose the location reference trace from those proposed. The ⬟ button (red) allows you to visualize the trace you have chosen. The ⬟ button is used to fix the monitoring thresholds.

– In the AVAILABILITY field, the AVAILABLE button must be selected. The STOPPED button can be used to interrupt the test. AVAILABLE must then be selected again to restart the test.

– With an OTU-8000, enter the injection threshold, in db.

– With the OTU-8000 exclusively, a periodical monitoring can be defined. In the field **Periodicity**:

  – Notch **Enable** in order to fill the different parameters.

  – Select the **first day** of the test by clicking on the arrow ▼ and select the **date** in the calendar displayed ▦.

  – Select the **starting hour** by using the arrows ▲▼

  – Enter the **period** during which the test will be performed (days / hours / minutes).

– Record the monitoring test thus defined in the database by clicking on the Apply ⬆ button.

– Display the RTU activity bar to visualize the name of the link tested, the test in progress, the state of acquisition progress, the acquisition parameters. To do this, select the RTU ACTIVITY option in the contextual menu.



– To check the configuration, it is advisable to run a test by pressing the TEST ON DEMAND button (see next section). At the end of the test, the message NO ANOMALY DETECTED should appear. If not, check the reference traces, the position of the markers, etc.

– Acknowledge then close the alarm message.

# Conducting an on demand test

To test a link for which a monitoring test has been defined (refer to "Defining a link monitoring test" page 141):

**1** Call the OPERATION window using the ▦ icon.

**2** Select the link in the LINKS folder on the left of the screen.

**3** Choose the MONITORING tab on the right of the screen.

**4** Click on the ON DEMAND TEST button.

**5** Wait for colored scrolling on the alarm bar. This color shows the severity of the result (see page 186).



Double click on this bar to open the alarm window listing the events occurred. From this window you can display the localization and detection traces and start an alarm report (see "Generating an alarm report" page 204).

# Implementing a maintenance program

A maintenance program is used to acquire results (attenuation of a splice, reflectance of a connector, etc.) at regular intervals so that any gradual degradation of the link can be detected.

To configure this program, proceed as follows:

**1** Call the OPERATION window using the  icon.

**2** Select the link in the LINKS folder on the left of the screen.

**3** Choose the MAINTENANCE tab on the right of the screen.



Function available
exclusively with
the **RTU**
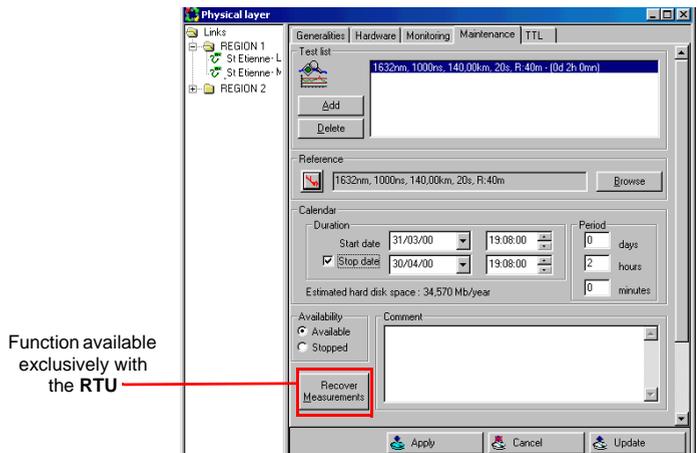
**Fig. 101** Setting up a maintenance cycle

**4** Click on the ADD button.

**5** In the REFERENCE field, click on the BROWSE button. Select the location reference trace in the dialog box OPEN A TRACE.

**6** In the CALENDAR box, choose the program start date, then the end date, as required.

**7** Choose the frequency of the test: a daily test is generally sufficient.

---

**NOTE**

The number of results stored, and thus the storage capacity required, depends on the frequency chosen. An estimate of storage needs for the chosen calendar is displayed in Mb/year. Check that storage capacity meets these needs and, if it does not, reduce the frequency of the tests.

---

**8** It is also possible to enter comments for maintenance tests.

**9** Save the configuration for the maintenance program in the database by pressing the icon  .

---

**NOTE**

The test created then appears in the test list.

---

### Action possible during the maintenance program

– The results will be stored in the RTU. They will be sent to the server at each autotest or each time the RECOVER MEASUREMENTS key is pressed (<u>with the RTU exclusively</u>).

– The STOPPED and AVAILABLE boxes can be used to interrupt the maintenance program and to revalidate it.

# Using a TTL signal to start the test on a link

When the TTL option is present on the RTU, it is possible to request that a link be tested when a TTL signal arrives on the corresponding input of the TTL connector. This input was defined on the Administration page (see <span style="color:blue">"To program the test of a link when the TTL signal status changes on an input:" page 55</span>).

To assign a test to a link:

**1** Select the link in the LINKS folder of the PHYSICAL LAYER page.

**2** Click on the **TTL** tab.

**3** Click on the ADD button.

**4** It is then possible to choose the different traces for detecting a fault and locating it. To choose these traces, click on the corresponding BROWSE button and find the reference trace of the link among those which are proposed. The ⬐ icon (red) is used to visualize the chosen trace. The 🗴 icon is used to define the monitoring thresholds.
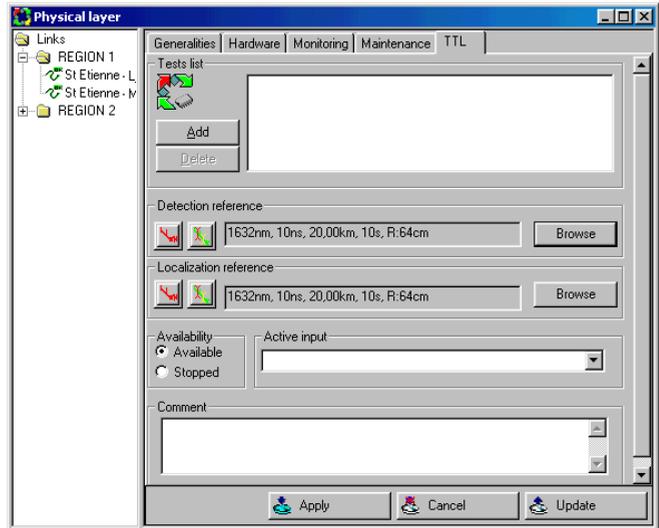


**Fig. 102** Assigning a test to a link

**5** In the ACTIVE INPUT drop-down menu, choose the input of the TTL connector.

**6** The STOPPED and AVAILABLE boxes can be used to interrupt the test and to validate it again.

– It is also possible to enter comments for maintenance tests.

– Store the information in the database by clicking on the Apply button in the toolbar.

**NOTE**

The test created then appears in the test list.

# Analyzing the results of a maintenance program

The measurement results from a maintenance cycle can be visualized in graph form and then transferred as required to a spreadsheet and/or printer.

**1** To call the corresponding function, click on the [icon] icon in the toolbar or choose the MEASUREMENTS option in OPERATION menu. The following display is then obtained:
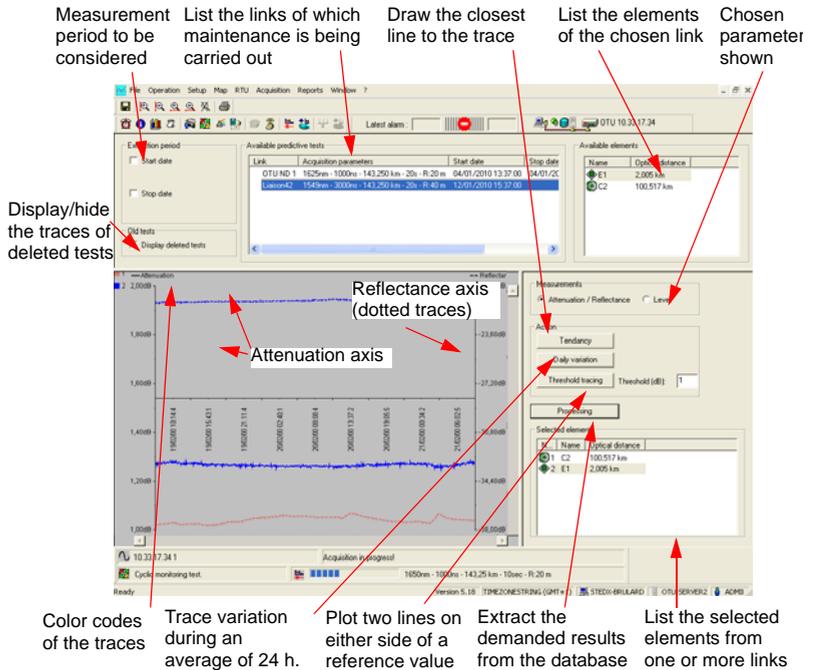


**Fig. 103** Measurement results

**2** In the field AVAILABLE PREDICTIVE TESTS, choose the link on which the element(s) you want to visualize are located. The elements of this link are then displayed in the neighbouring AVAILABLE ELEMENTS field (together with their characteristics).

**3** Select one or several of these elements by dragging them to the SELECTED ELEMENTS field in the bottom right hand corner (or double-click on an element to select it).

> **NOTE**
>
> To remove an element from this list, select it and then use the DELETE key.

A number is assigned to each element selected.

**4**  If necessary, proceed in the same way to select the elements of other links being tested.

**5**  If you wish, you may choose the start and end dates for the period of the maintenance cycle that you want to visualize. If no choice is entered in the EXTRACTION PERIOD field, all available measurements will be displayed.

**6**  Click on the PROCESSING button to call the measurement results corresponding to your selection

## Displaying results

### Color

The selected elements of the link are identified by a number.

The corresponding traces are displayed with a color code (identified by the element number) and located in the upper left corner of the trace display window.

Double-click on an element (in the SELECTED ELEMENTS field) to change the color of its trace, as required.

### Measurements

The selection made in the MEASURES field can be used to obtain:

– either the attenuation and reflectance results simultaneously with: a scale for one of the parameters on the right and a scale for the other on the left, with <u>attenuation displayed in the form of a solid line and reflectance as a dotted line</u>.

– or the results concerning the level.

> **NOTE**
>
> If a result is not available for a selected element, the corresponding trace will not be displayed.

### Zoom and framing

Zoom and framing functions are available for use with traces, as required. These same functions are accessible from the toolbar or keyboard (see table below).

| icon | Keyboard shortcuts | Function |
|------|-------------------|----------|
|  | z | Vertical zoom in |
|  | u | Vertical zoom out |
|  | Z | Horizontal zoom in |
|  | U | Horizontal zoom out |
|  | | Remove zoom |
| | f | Horizontal framing of all the results in the window |
| | F | Vertical framing of all the results in the window |
| | Arrows | Shifting of traces in 4 directions |

### Tendancy

If the trace indicates a lot of noise, it is sometimes difficult to determine its slope and, thus, any degradation.

In this case, click on the TENDANCY button to display, for each trace, the straight line closest to its different points. This straight line is defined using the least-squares method.
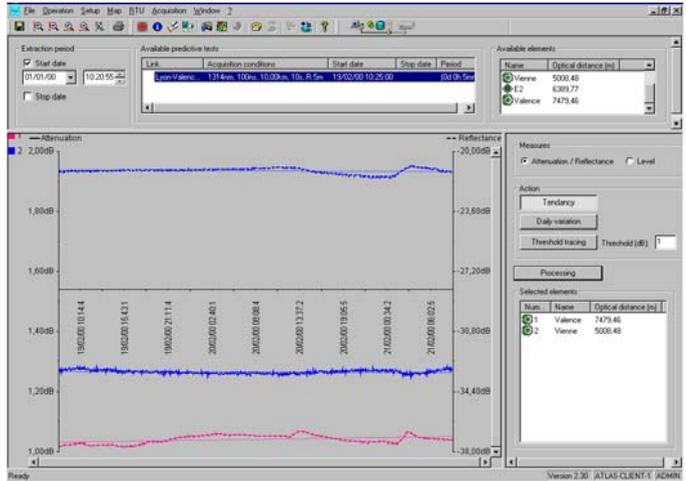
**Fig. 104**     Displaying the trend of traces

To delete the straight lines, click on the TENDANCY button again

### Daily Variations

To identify phenomena appearing at certain times of the day, such as
those linked to temperature, the ONMS calculates the mean of the
measurement results for the period visualized, according to the time of
the day.

To display this trace:

**1**   Click on the DAILY VARIATIONS button

**2**   Click on the trace concerned: the daily variations table appears.
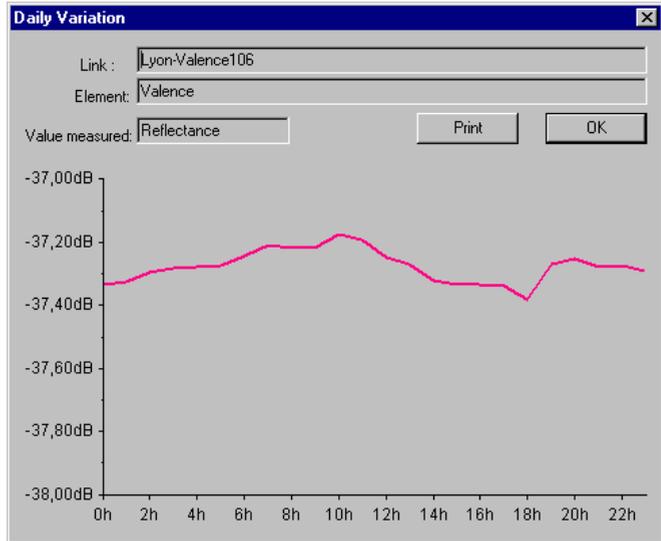
**Fig. 105**     Daily variation of results

**3**   To print this table, click on the PRINT button.

**4**   To close it, click on OK.

### Threshold tracing

The THRESHOLD TRACING key is used to plot two lines on either side of a reference value. This reference value is calculated from the reference trace used for the maintenance cycle. To display these lines, click on the trace concerned and adjust the zoom level, as required.

The difference between each line and the reference value must be selected in the THRESHOLD field, in dB.

Threshold

Click the Threshold tracing
button, then click on the trace

Threshold
selection
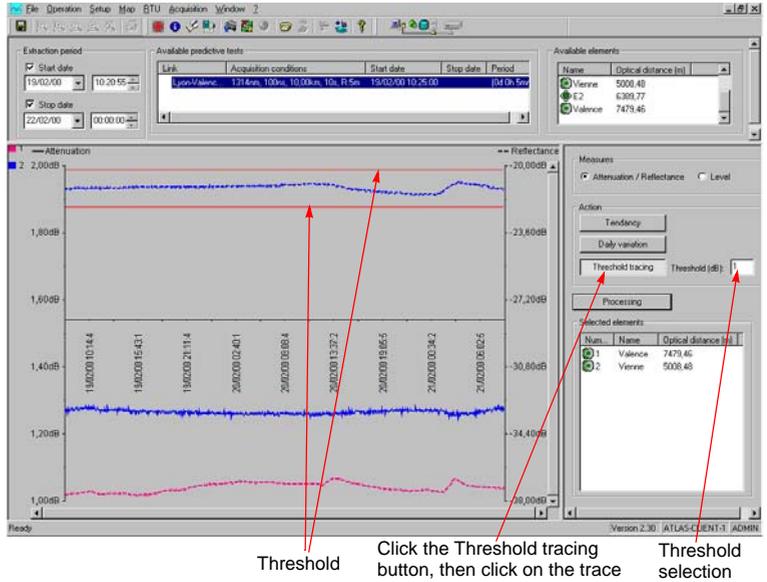
**Fig. 106** Display of the tendency and threshold tracing on a
trace

**Backing up displayed results**
Click on the ![disk icon] icon or choose SAVE AS... in the FILE menu to give the name and destination of the backup file. The results are backed up in the form of a tabulated text which can be imported into a spreadsheet.

**Printing displayed results**
Before requesting a print of the results displayed by the FILE/PRINT menu, the desired page format can be defined through the FILE/PAGE SETUP menu. A dialog box is used to enter headers and footers and define the desired margins:
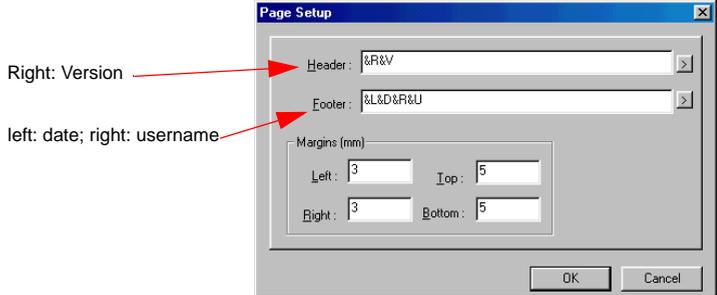
**Fig. 107**     Page layout

– For the Header and Footer click the associated button  >  , then enter the required information. The Page Setup menu contains the following values:

| &D | Current date | &L | Align to the left |
| &T | Current hour | &C | Align in the center |
| &V | Version | &R | Align to the right |
| &U | Username | | |

Enter the alignment value before entering the information code as shown in the above example.

– Enter the desired margins in mm.

# Link between schematic and OTDR trace

**Simultaneous display of schematic and trace**

A schematic and the corresponding trace file can be displayed simultaneously on the screen. To do this:

**1** Open the trace (see "Opening an OTDR trace" page 214).

**2** Open the corresponding schematic using the OPEN SCHEMATIC tab from the SCHEMATIC menu.

**3** Choose TILE from the WINDOW menu.

As the schematic and trace are being displayed simultaneously on the same screen, to access the SCHEMATIC menus click on the schematic window, and to access the trace menus click on the trace window.

**Transfer of markers between schematic and trace**

The link between the schematic and the trace allows markers associated to schematic elements to be transferred to the trace.

To do this:

– With an active trace, choose the IMPORT MARKERS FROM SCHEMATIC option from the SCHEMATIC menu.The markers associated to each element appear on the trace at the theoretic position given by the fiber lengths of the schematic,

– If the theoretic positions do not correspond to the effective positions given by the trace, you have the possibility of adjusting the fiber lengths with respect to a real acquisition made on the fiber,

– On the trace, position the markers at the base of the corresponding optical event,

– Transfer the new positions with the EXPORT MARKERS TO SCHEMATIC option of the SCHEMATIC menu,

The lengths of the optical fibers of the schematic are automatically readjusted, the legend "MES" (measured) indicates that the optical length is no longer estimated "EST", but measured. On the other hand the physical lengths remain unchanged.
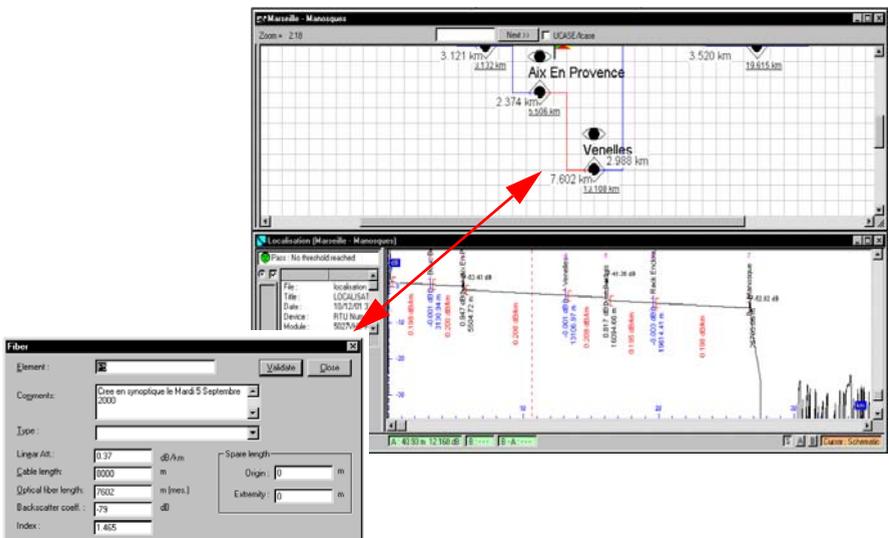


This readjustment allows the optical elements of a link to be localized precisely. When a fault is detected, the system can determine the intervention location, from a geographic position, due to the fact that it knows the optical and physical lengths of the fibers and cables.

**Dynamic link between schematic and trace**

It is possible to select an element on the schematic and to automatically indicate its corresponding event position on the trace. Inversely, it is also possible to point to an event on the trace and select the corresponding event on the schematic.

### Schematic-trace link

Select an element on the schematic with the right mouse button. The element is shown in red on the schematic and a dotted vertical red cursor is positioned before the event on the trace (refer to the figure "Schematic-Trace Link" page 156).

It is also possible to make a normal selection of the schematic by using the left mouse button, and then either:

– pressing the <F8> key,
– or choose, in the SCHEMATIC menu, the LINK TO TRACES command.



Selected element (right mouse button) in red

Red cursor indicates the corresponding element of the schematic
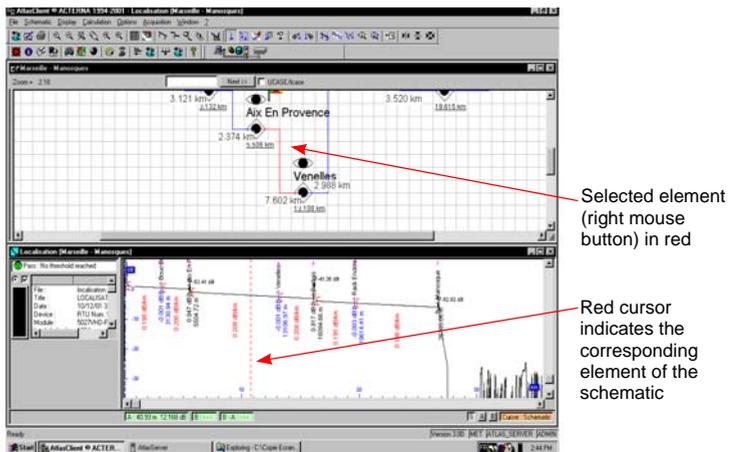
**Fig. 108**    Schematic-Trace Link

### Trace-Schematic Link

On the status bar on the right-hand side, click on the S button, to display the S cursor in the associated window. The S cursor (red) is then present in the trace window.

Move the cursor to mark a point on the trace[3]:

---

3. If you are in Marker mode, press the Ctrl key to move the cursor.

– The element on the schematic corresponding to this point on the trace, starts flashing.

– A window indicates, beside the Schematic cursor, distances between
  - the cursor and the previous marker
  - the cursor and the closest previous landmark.
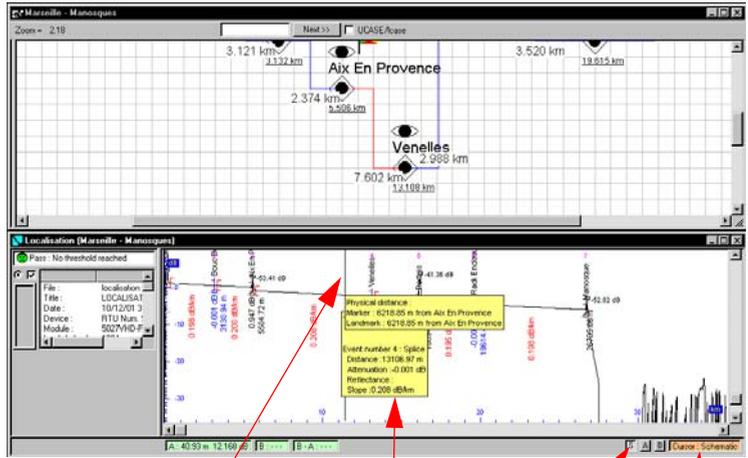


| Schematic Cursor | Schematic Cursor Distance of marker and located precedent topology | Select of Schematic cursor |

**Fig. 109**    Trace-Schematic Link

# Photonic layer of the system

8

This chapter describes the basic procedures for using the system's photonic layer.

Topics discussed in this chapter are as follows:

# Description of the system's photonic layer

The photonic layer of the ONMS system concerns the monitoring of the wavelengths (or channels) transmitting the information over a network.

This monitoring is ensured by JDSU RTUs (fitted with WDM modules) acquiring spectrums, analyzing them, comparing them with a grid, detecting results out of standards, and issuing alarms.
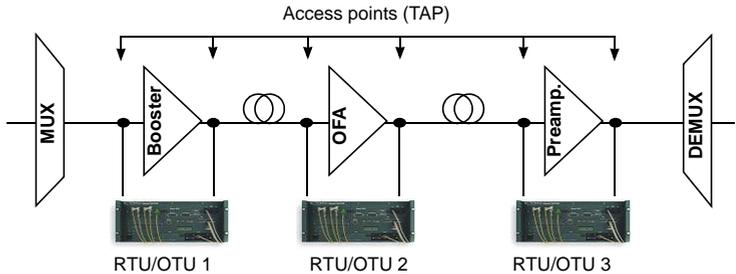


**Fig. 110**      Example of network with equipment (MUX, DEMUX, Amp, etc.) and monitoring RTUs connected to TAPs.

### Access point definition

The access points or TAPs (Test Access Points) are points at the input and output of equipment which can be connected to an RTU to monitor them or acquire a spectrum.

### Spectrum characteristics monitored

– Presence, wavelength, power and Signal/Noise ratio of the channels
– composite power, gain Tilt and gain Slope.

Each result in the grid tolerances corresponds to an event sent to the server. Therefore, a photonic alarm usually consists of several events.

### Graphic interface

The photonic layer uses a graphic interface: the RTUs ensuring monitoring can be displayed on a background image representing the network. When a measurement result is out of standards on an RTU, the latter is outlined with a circle, with a color indicating the alarm level.

Three fields enable easy display of the lists of regions monitored, RTUs used, access points defined in the equipment, channels or owner organizations or channel leasers, with a reminder of the characteristics of the element selected.

### Main actions possible from the Photonic layer window

– Connecting an equipment access point to a port available on an RTU.

– Configuring monitoring for a spectrum.

– Allocating a channel to an organization

– Displaying the optical path of a channel, i.e. its path in an optical network (list of TAPs it passes through).

## Photonic Layer display window

This menu is called using the  button or the Setup > Photonic layer command.
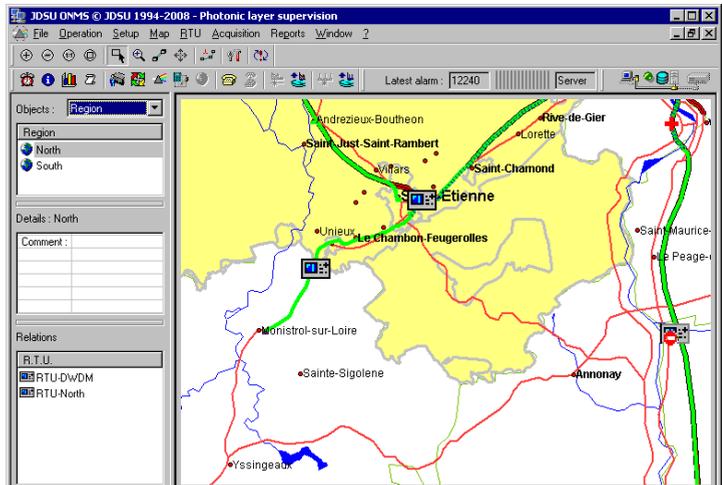


**Fig. 111** Defining a link

The right part of the screen displays the background image selected (see "Selecting a background image" page 165) containing the RTUs ensuring network monitoring (see "Placing the RTUs on the map" page 167). The alarm status of each RTU is automatically updated by the system.

The left-hand part of the screen includes three fields:

1.  the Object field displays, according to the selection using the
    scrolling menu:
    - either the list of Regions monitored by the system,
    - or the RTUs used by the system for monitoring,
    - or the Access Points of the system,
    - or the Organizations defined by the system administrator.
2.  the Detail field reminds the characteristics of the object selected.
3.  the Relations field displays the list of items related directly with the
    object.

| Object selected | Details and Relationships displayed |
|---|---|
| Regions | Comment entered by the administrator in the definition box of the region selected. List of RTUs monitoring this region. |
| RTU | Characteristics of the RTU selected: Software version, Accessibility, Main media, Main address, Secondary media, Backup address. List of TAPs for this RTU. List of ports available in this RTU (not connected to TAPs). |
| Access Point | Name of the RTU connected, RTU port used, TAP monitored or not, equipment identification, monitoring periodicity, number of channels managed, and number of channels monitored. List of managed channels. For each channel, whether it is monitored or not. |
| Organization | Address, tel and fax # of the organization, Comments. List of TAPs and channels related with the organization. |

**Status of the RTUs**  The various statuses possible for an RTU and TAP are represented by the following icons:

| Icon | Meaning |
|---|---|
|  | (grey RTU) RTU using an OTDR module available |
|  | (yellow RTU) RTU connected to the server |
|  | (red RTU) A probelm occured on a module or on the OTAU |

| Icon | Meaning |
|------|---------|
|  | RTU unavailable |
|  | RTU selected |
|  | RTU having detected at least one optical, photonic or system alarm. The color of the circle blinking corresponds to the highest severity when several alarms are detected by this RTU. Right click on it, select Alarm, then select an alarm to have all details about this alarm (see Chapter 9 "Alarms management"). |

**Status of the TAPs**  The various statuses possible for a TAP are represented by the following icons:

| Icon | Meaning |
|------|---------|
|  | Access point available |
|  | Access Point unavailable |
|  | Port of the RTU not connected with an equipment |

# Commands related with objects

Right-click an object (Region, RTU, TAP, or Channel) to display the list of commands available for this object. These commands can be different, according to whether the object is in the upper Object field or the lower Relations field.

### View Region

A right-click on an RTU included in the Object field followed by the Visualize Region command moves the regions in the Object field and moves the RTUs to the Relations field.

### View RTU

A double-click on an RTU in the Relations field (or a right-click followed with the Visualize RTU command), moves the RTU up in the Object field to display its Details and Relationship.

A right-click on a TAP in the Object field followed with the command Visualize RTU, moves the RTU up in the Object field to display its Details and Relationship.

### View TAP

A double-click on a TAP in the Relations field (or a right-click followed with the Visualize TAP command), moves the TAP up in the Object field to display its Details and Relationship.

A double-click on a TAP in the Object field opens the TAP configuration box.

### Connection/disconnection

From an RTU, a TAP or a port not connected to a TAP, the Connection (or Disconnection) command enables connecting the RTU to the TAP (or disconnecting it).

### Removing the TAP

**NOTE**

To be able to delete a TAP, you must connect to the RTU.
In addition, no active alarm must be connected to it. If there are any, they must be acknowledged and resolved, then transferred to the Log.

To remove a TAP, right-click the TAP name in the Object or Relations field, then select the Delete the TAP command. The port to which the TAP was connected re-displays on the list of ports available.

### Adding a TAP to a port

To be able to add a TAP to an equipment's port, the latter must be connected to the RTU.

Right-click on this port in the Relations / TAP field, then select the Add a TAP command. The TAP configuration box displays (see "Setting TAP under monitoring" page 169).

### Display a name for a channel's owner and client

With the TAPs in the Object field, right-click on a channel in the Relation-ship field to display the name of its owner and client.

### Test on demand, Suspend and Restart monitoring

To be able to launch a test on request on a TAP, this TAP must be connected to the RTU.

Click on the TAP name located in the Object or Relations field, then select the Test on request command. A test is launched on the TAP. The Suspend and Restart monitoring commands stop and restart this monitoring.

### Editing optical paths

Right-clicking on an Organization calls the Optical paths management command, which displays the Optical paths management box.

The ⬛ button calls this same box directly.

See "Editing an optical path" page 177.

---

# Selecting a background image

The background image is a map or a schematic or logical description of the network.

The files representing the background images must be stored in the database of the ONMS.

To store images in the database, update them, remove them, or select a background image: click on the ⬛ button. The Photonic Layer Config-uration windows displays.

### Format of the image files

bmp, png or jpeg.

### Selecting the background image

If the image is in the database, select it using the Background image scrolling menu, then click on the OK button. The selected map is displayed.

> **NOTE**
>
> If the map is not displayed, use the ⬛ icon to center it on the screen.

### Adding images to the database

**1**   Click on the Add button.

The Open dialog box enables searching files to add and selecting them.

**2**   Click on OK: they are added to the database.
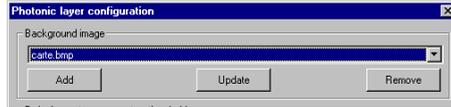


**Fig. 112**   Selecting the background image

### Removing images in the database

**1**   Using the Background image scrolling menu, select the image to remove.

**2**   Click on the Remove button. The image is deleted in the database.

### Updating images in the database

The image to update selected in the Background image scroll menu:

**1**   Click on the Update button.

The Open dialog box enables searching the new version of the image.

**2**   Click on OK to replace it in the database.

**Zoom and map moving functions**

Several buttons are available to ensure the zoom in, zoom out, reduction and move function on the background image:

⊕   Pressing this button several times enables displaying the various levels of zoom in.

⊖   Pressing this button several times enables displaying the various levels of zoom out.

(1:1)   Exits zooming. The absence of zoom is recommended to have the highest display performance.

⬚   Displays the full image.

⊕   Click this key and outline the zone which will display in the entire window.

⬌   Click on this button, then click on the image, and while maintaining the mouse button, drag the cursor on the image to move it in the direction required.

# Placing the RTUs on the map

Select (in the Object or Relations field) the RTU to place and drag it on the map up to the point required. Release the mouse button.

> ▬▬▬▬ | **NOTE**
> To place the RTU precisely, zoom in on its placement zone.

**Information bubble related with the RTU**

When the cursor is located on the RTU, an information bubble displays:

– the RTU name
– the list of alarms and their severity, if in alarm:
   - system alarm status
   - link status
   - TAP status

**Tracing an optical path linking the RTUs**

This line is only an indication on the map. It is not associated with data.

To place a line between two RTUs:

**1** Click on the [button icon] button.

**2** Click on one of the RTUs.

**3** Drag the mouse without releasing the button, up to the other RTU, and release the mouse button.

**Removing a line between two RTUs**

– After a right-click on the line, in the list displayed, select Delete from map.

**Moving, deleting, viewing an RTU**

When a RTU is selected it has the following aspect on the map: [icon].

To select a RTU:

– either, if the button [icon] is activated (RTU selection mode), click on the RTU on the map.
– or click on the RTU name in the Objects or Relations list .

To move a selected RTU, maintain the mouse button depressed, and drag up to the new location required.

To delete a RTU, right-click on it and select the Delete from the map command.

To view the RTU relations on the left side, right-click on it on the map or in the Objects list, then select the Visualize command.

**Viewing a RTU alarm**

The RTU having detected at least one alarm is circled. To view details on a RTU alarm :

**1** Right click on the circled RTU,

**2** Select the Alarm command: alarms relative to this RTU are listed.

**3** Select the alarm then click on the button Display Alarm.

The box giving all details on this alarm displays: its description is given in chapter .

**Choosing the default optical spectrum monitoring thresholds**

**1** Click on the ⚙️ button. The following window appears.
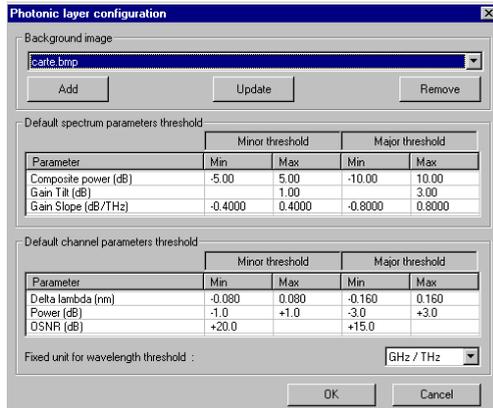


**Fig. 113** Selecting the default thresholds

**2** Enter the deviations authorized in respect of spectrum values measured.

▬▬ **NOTE**

These deviations are used only to configure TAPs from a spectrum.

Each parameter is associated with minor and major thresholds:
- exceeding a minor threshold triggers a minor alarm;
- exceeding a major threshold triggers a major alarm.

**3** Select the unit used for wavelengths: GHz /THz or nm.

**4** Click on OK.

# Setting TAP under monitoring

To monitor a TAP, it must be identified, then associated with an RTU port and the optical equipment to monitor.

### To create a TAP, you must

– Select the RTU, connect it, and select a port available.

– After a right-click on the port selected, select the Add a TAP on a port... command
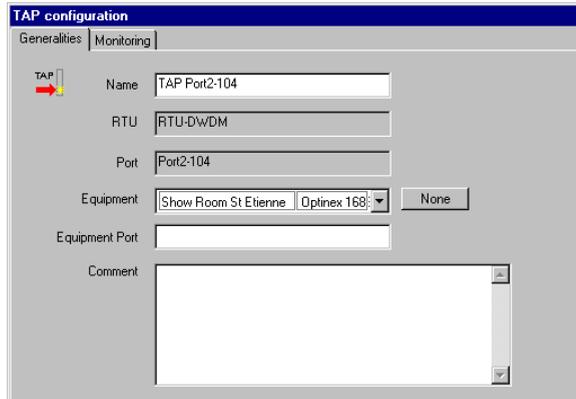


**Fig. 114** Configuring the TAP - General

– Under the Generalities tab appear the TAP's name (can be modified) and the RTU's name and its port used.

– Select the equipment to monitor and enter any comment required. The name of the equipment port connected to the TAP is reminded.

– Under the Monitoring tab, configure the measures made on the level of the TAP and define their frequency. Configuring the parameters can be performed manually (see "Manual configuration of monitoring" page 170) or by importing the parameters of an existing spectrum or a trace or text file (see "Configuration of monitoring by import" page 173)

### To modify the configuration of a declared TAP

To call the edit box for a declared TAP, two methods are possible:

– either by selecting Access Points in the Object field, and double-clicking the TAP to modify.

– or, after a right-click on the TAP (in the Object field or in the Relations field), by selecting the Edit... command.

Configuration of the TAP is then available. You can change the name of the TAP, the equipment monitored and, if the RTU is connected, the configuration of the monitoring (see below).

> If the RTU is not connected, the part under the Monitoring tab is read-only; and cannot be modified.

**Manual configuration of monitoring**

Configure the measurements which will be made on the level of the TAP, i.e. choose the parameters for the spectrum and channels to be monitored, as well as the thresholds beyond which a minor or major alarm is triggered.

For this, it is recommended to use the specifications provided by the manufacturers of the equipment to monitor.

### Spectrum configuration

– Check the box corresponding to the spectrum parameter to monitor: Composite power, Gain tilt, Gain slope.

– For each parameter, enter its value and the min and max deviations beyond which the RTU will trigger minor and major alarms. For tilt gain, as the ideal value is 0, entering a min value is useless.

### Periodicity of measurements

The Monitoring scrolling menu proposes:

– either stopping monitoring (Stopped)

– or permanent monitoring (Permanent). If several TAPs are monitored, they will be continually, one after the other.

– i.e. the measurement periodicities of 1, 2, 4, 8, or 12 hours, or 1 or 2 days.
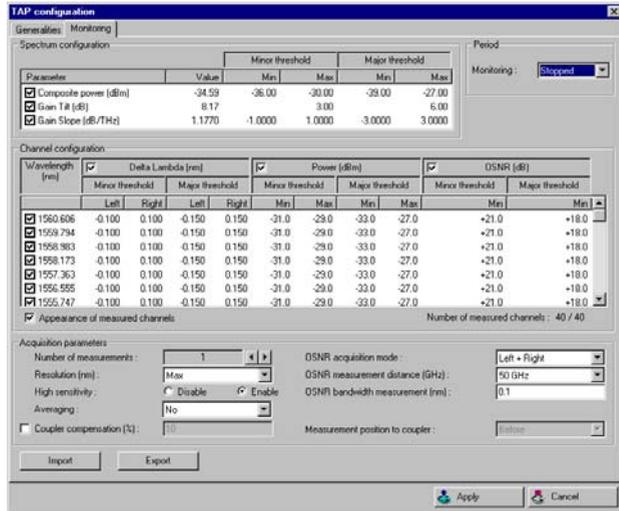
**Fig. 115** Configuring the TAP - Monitoring

### Configuring channels

To enter or modify manually the wavelengths of the channels to monitor and associated thresholds:

– After a right-click in the field reserved to the channels, select the Add a channel command. The line reserved to this channel displays.

– Enter the wavelength, then hit Enter. Enter the min and max thresholds beyond which minor and major alarms will be triggered: upon each input, the following field displays to enter the next threshold. The Signal/Noise ratio parameter (OSNR) has no max threshold, as the channel quality is as high as this ratio is.

All channels being entered, for a given monitoring, select by checking their box:

– the channels to monitor (the number of channels selected and the total number of channels are reminded under the field reserved to the channels).

– the parameters to monitor on the channels: Lambda derivation, Power,
Signal/Noise ratio (OSNR).

When a channel not selected for monitoring, is detected by the measurement, you can request an alarm to be triggered. In order to do this, check the Appearance of measured channels box.

### Configuration of the measurement

The following parameters must be selected to configure measurement acquisition:

| | |
|---|---|
| **Number of measurements** | from 1 to 60.<br>The values 2 to 60 are used to generate statistics which are not used by the ONMS system. |
| **Resolution (nm)** | - Max: maximum resolution of the filter (< 0.1 nm)<br>- 0. 1   0.2   or   0.3 nm |
| **High Sensitivity** | - Inactive: the power range varies from +10 to - 60 dBm.<br>- Active: the power range varies from -25 to -70 dBm. |
| **Averaging** | None, High or Low<br>This function enables reducing the noise level by a value reaching 5 dB. |
| **OSNR acquisition mode** | side of the peak where the reference point is taken for noise (left, right, left and right) |
| **OSNR measurement distance (GHz)** | distance between the channel peak and the reference point for noise:<br>Auto: distance according to channel spacing.<br>or at 25 GHz, 50 GHz,100 GHz from the peak. |
| **OSNR bandwidth measurement (nm)** | Reference band used for noise acquisition:<br> values included between 0.001 nm and 10 nm. |
| **Coupler compensation (%)** | If the measurement is performed through a coupler, to compensate the loss introduced by this coupler, check the box, and enter the compensation value, as a percentage of the value measured  from 1 to 99%. The thresholds indicated take compensation into consideration. |
| **Measurement position to coupler** | This feature is not available. |

When the configuration of the TAP is completed, click on the Apply button to save in the database.

## Exporting a TAP configuration

To export the configuration of a TAP to a text file, in the TAP configuration box, under the Monitoring tab, click on the Export button.

The Save As... box enables searching or creating the directory storing the file, and naming it.

This file has the txt format, enabling to read under Excel<sup>TM</sup> for example. Certain parameters can hence be modified before use.

**Configuration of monitoring by import** To import a TAP configuration from an existing TAP or spectrum, or from a trace or text file:

– under the Monitoring tab, click on the Import button.
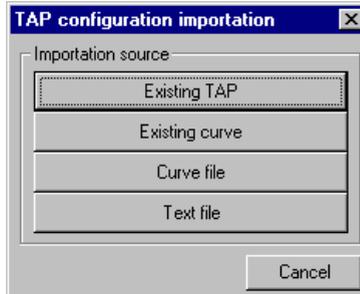


**Fig. 116** Source of the TAP configuration import

– click on the button corresponding to your choice

### Import configuration from an existing TAP

**1** Click on the Existing TAP button.
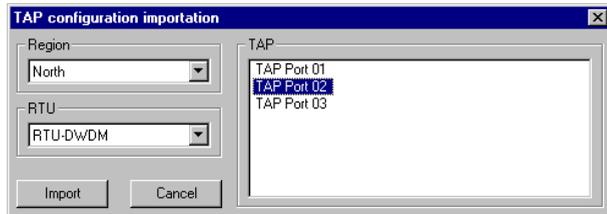


**Fig. 117** Choosing the TAP source

**2** Select the region, RTU, and TAP used as a configuration source.

**3** Click on the Import button.
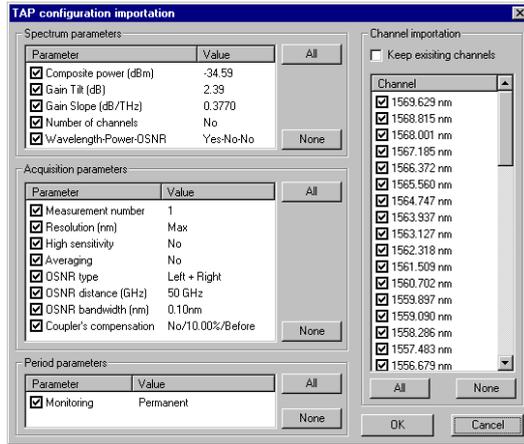
**Fig. 118** Selection of parameters to import

**4** Check the boxes corresponding to the source configuration parameters to be imported. In each section, to ensure an easier choice, the All and None buttons enable selecting all the parameters in the section or none.

**5** Check the Keep existing channels box if you wish the imported channels to be added to the existing ones, without deleting them. To ensure the imported channels replace the existing channels, do not check the box.

**6** When your choices are made, click on the OK button. The TAP configuration window then contains the imported parameters.

### Import configuration from an existing spectrum

– In the "Source of the TAP configuration import" page 173 box, click on Existing curve button.

– In the Open spectrum box, choose a TAP: the list of spectrums acquired on this TAP and their main characteristics are displayed.

– Choose the spectrum, and click on OK. the parameter to import selection box displays as in the previous case (see "Selection of parameters to import" page 174). When your choices are made, click on the OK button.

### Import configuration from a curve file

– In the box enabling to select the import source for a TAP (see page 173), click on the Curve file button.

– In the Open box, search the curve file: it must have the wdm format.

– Select this file, and click on the Open button.

The selection box for the parameters to import displays (see "Selection of parameters to import" page 174). When your choices are made, click on the OK button.

### Import configuration from a text file

In the box enabling to select the import source for a TAP (see page 173), click on the Text file button.

– In the Open box, search the text file: it must have the txt format.

– Select this file, and click on the Open button.

The selection box for the parameters to import displays (see "Selection of parameters to import" page 174). When your choices are made, click on the OK button.

**Launching a monitoring test on a TAP**

**1** In the Object field, display the RTUs. Select the RTU the TAP is connected to. The TAPs associated to the RTU are displayed in the Relations field.

**2** Connect the RTU.

**3** Select the TAP. Right-click on it and select the Test on demand command.

The test starts: a bar at the bottom of the screen indicates its progress.



| TAP Port 03 | Acquisition in progress! |
| Test on demand. | 1 Measurements, High sensitivity, R:Max, OSNR:Left + Right/50 |
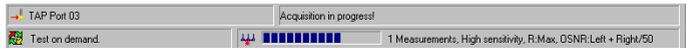
**Fig. 119**     Progress status of the Test on demand

### Test result

At the end of the test, an alarm provides the result: either TEST OK, or the detail of any anomaly detected.

### Temporary monitoring hold

While monitoring is active, right-clicking on the name of the TAP enables:

– stopping it, by selecting the Suspend monitoring command.

– starting it, by selecting the Restart monitoring command.

### Hold monitoring

Monitoring can be held from the TAP Configuration window under the Monitoring tab (see page 171):

– In the Period field, select Stopped: monitoring is held.

# Acquiring a spectrum

**1** In the left field, select the RTU ensuring measurement, and connect.

**2** Click on the ⬇️ button. The list of RTU ports and TAPs declared displays.
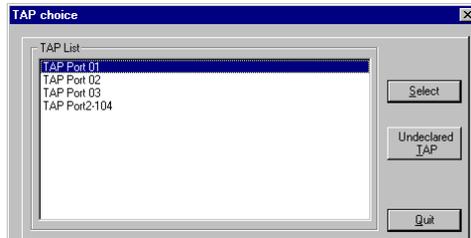


**Fig. 120** Selecting the measurement point

### Measuring a TAP declared

**1** Select the TAP, and click on the Select button.

The Acquisition setup box reminds the name of the TAP and the type of module of the RTU ensuring measurement.

**2** Enter the acquisition parameters as for TAP configuration (see "Configuration of the measurement" page 172).

**3** Click on OK.
A progress bar indicates Tuning in progress then Acquisition in progress.

The Stop button enables stopping current acquisition.

At the end of acquisition, the optical spectrum displays.
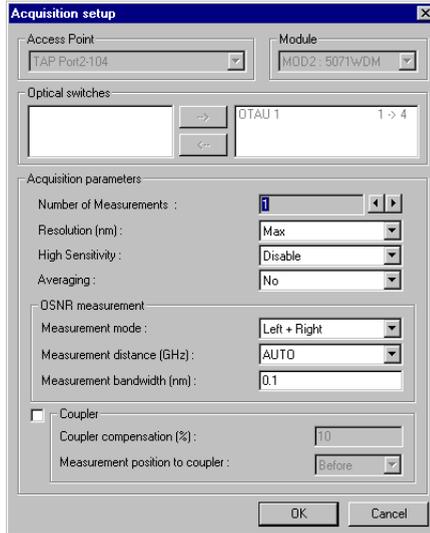
**Fig. 121** Configuring spectrum acquisition

# Allocating a client or owner to a channel

It is possible to create an optical path consolidating all TAPs a channel passes through.

To create this optical path and allocate it to a client or owner, call the Optical paths function using the button. See "Editing an optical path" page 177.

# Editing an optical path

**Defining Optical paths**
The Optical path of a channel corresponds to its path in an optical network, i.e. the list of TAPs it passes through.

It is identified by a name and includes the following information:

– channel wavelength
– path owner organization
– path client organization

– type of traffic used on this channel: STM1, STM4, ATM, etc

– the various TAPs through which the channel passes

**Defining or modifying the optical path of a channel**

Move to the photonic layer display using the  button or the Setup > Photonic Layer command.

Two methods provide access to the Optical path management box:

| | |
|---|---|
| 1st method: | Select the Organization object. Right-click an organization, and select the Optical paths management... command |
| 2nd method: | Click on the  button. |

In both cases, the dialogue box displayed displays the list of optical paths created on the system.
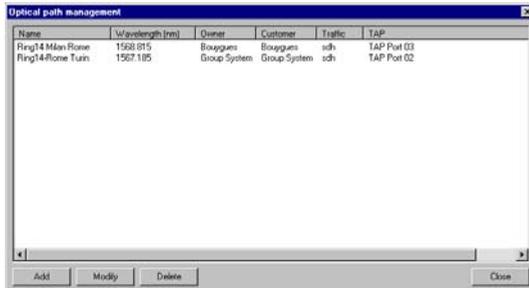


**Fig. 122**     Adding - Modifying - Deleting an optical path

**To define an optical path:**

**1** Click on the Add button.
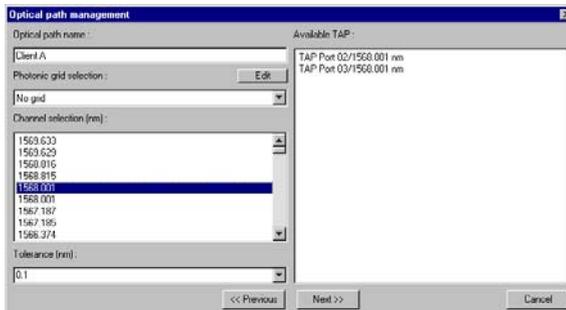


**Fig. 123**     Adding or modifying an optical path

**2** Enter a name for this path.

**3** Select a grid if required (see "Adding a photonic grid" page 180).

The values set out in standard ITU G692 are recommended (see Appendix at the end of chapter page -181).

**4** Select the channel among those defined by the system.

**5** Select the tolerance on the channel's wavelength. The TAPs monitoring this wavelength (plus or minus the tolerance selected) are then displayed in the right-hand field.

The right-hand section displays the list of TAPs created on the system and for which the wavelength is present.

**6** Select the TAPs included to the optical path.

**7** Click on the Next button.

The list of organizations created on the system is available in the Owner organizations and Client organizations fields.

**8** Select the owner and client organizations for this optical path.
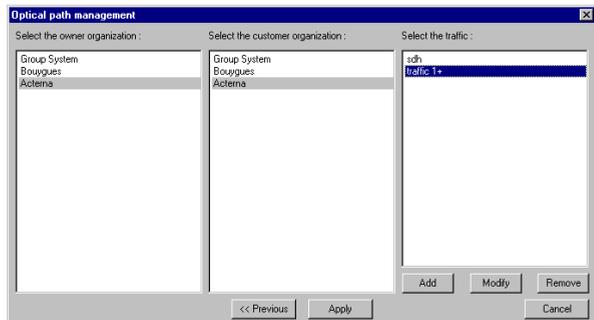


**Fig. 124** Traffic using the channel

The Select the traffic field displays the list of traffic types entered previously.

– You can add a new type of traffic. In order to do this, under the Select the traffic field, click on the Add key, enter the type of traffic in the field proposed, then click again on Add.

– You can modify an existing type of network. In order to do this, select the traffic name, click on the Modify key, then modify the traffic name, and click again on Modify.

– The type of traffic being chosen, click on the Apply button. The optical path created displays in the list.

> ⚠ The successive elements of the optical path (Owner organizations, Client organizations, Traffic) must be highlighted when pressing on the Apply button. Those which are not will not be associated with it.

**Adding a photonic grid**

To edit a new grid, in the Optical Management window (see page -178), in the Photonic grid selection field, click on the Edit button and, in the menu displayed, select Add a grid. The grid edit box displays:



**Fig. 125** Editing a photonic grid

**1** Enter the required parameters:

| | | |
|---|---|---|
| a - Name | Name of the grid | |
| b - Reference | Wavelength[1] from which the others wavelengths will be calculated. | |
| c - Step | Delta wavelength [2] | |
| d - Start and End | Wavelength limits[1] of the grid. | |
| e - Step unit: | Shows in which unit (nm or GHz/THz) the delta lambda is constant. If this unit is different from the unit selected for the Step field(c), the step value is converted in the Step Unit (e) to calculate the wavelengths of the grid. | |

---

1.Reference, Start and End wavelength expressed in nm or THz :
selection in the Setup > Client > Units > Wavelength units menu in the Lambda field (see "Configuring the units" page 82).
2.Step expressed in nm, THz or GHz :
selection in the field Delta Lambda of the menu indicated above.
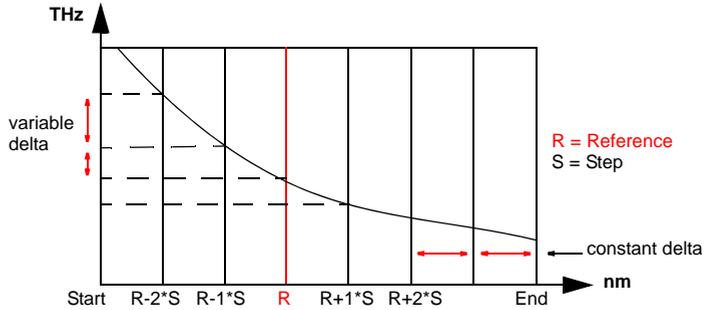
**Fig. 126** Exemple with Step Unit = nm

**2** Click on OK: the new grid appears in the Photonic grid selection scrolling menu.

### Modifying a grid

An existing grid can be modified. To do this:

**1** Select the grid in the Photonic grid selection menu.

**2** Click on the Edit button and, in the menu displayed, select Edit the grid.

**3** Make the modifications required, and click on OK.

### Deleting a grid

To delete a new grid, in the grid selection field, click on the Edit button and, in the menu displayed, select  Delete the grid.

# Appendix: ITU G.692 grid

**Table A.1/G.692 – Nominal central frequencies (THz)**

| Nominal central frequencies for spacings of 50 GHz | Nominal central frequencies for spacings of 100 GHz and above | Nominal central wavelengths (nm) |
|---|---|---|
| 194.60 | 194.60 | 1540.56 |
| 194.55 | – | 1540.95 |
| 194.50 | 194.50 | 1541.35 |

| Nominal central frequencies for spacings of 50 GHz | Nominal central frequencies for spacings of 100 GHz and above | Nominal central wavelengths (nm) |
|---|---|---|
| 194.45 | – | 1541.75 |
| 194.40 | 194.40 | 1542.14 |
| 194.35 | – | 1542.54 |
| 194.30 | 194.30 | 1542.94 |
| 194.25 | – | 1543.33 |
| 194.20 | 194.20 | 1543.73 |
| 194.15 | – | 1544.13 |
| 194.10 | 194.10 | 1544.53 |
| 194.05 | – | 1544.92 |
| 194.00 | 194.00 | 1545.32 |
| 193.95 | – | 1545.72 |
| 193.90 | 193.90 | 1546.12 |
| 193.85 | – | 1546.52 |
| 193.80 | 193.80 | 1546.92 |
| 193.75 | – | 1547.32 |
| 193.70 | 193.70 | 1547.72 |
| 193.65 | – | 1548.11 |
| 193.60 | 193.60 | 1548.51 |
| 193.55 | – | 1548.91 |
| 193.50 | 193.50 | 1549.32 |
| 193.45 | – | 1549.72 |
| 193.40 | 193.40 | 1550.12 |
| 193.35 | – | 1550.52 |
| 193.30 | 193.30 | 1550.92 |
| 193.25 | – | 1551.32 |
| 193.20 | 193.20 | 1551.72 |
| 193.15 | – | 1552.12 |
| 193.10 | 193.10 | 1552.52 |
| 193.05 | – | 1552.93 |
| 193.00 | 193.00 | 1553.33 |
| 192.95 | – | 1553.73 |
| 192.90 | 192.90 | 1554.13 |
| 192.85 | – | 1554.54 |

| Nominal central frequencies for spacings of 50 GHz | Nominal central frequencies for spacings of 100 GHz and above | Nominal central wavelengths (nm) |
|---|---|---|
| 192.80 | 192.80 | 1554.94 |
| 192.75 | – | 1555.34 |
| 192.70 | 192.70 | 1555.75 |
| 192.65 | – | 1556.15 |
| 192.60 | 192.60 | 1556.55 |
| 192.55 | – | 1556.96 |
| 192.50 | 192.50 | 1557.36 |
| 192.45 | – | 1557.77 |
| 192.40 | 192.40 | 1558.17 |
| 192.35 | – | 1558.58 |
| 192.30 | 192.30 | 1558.98 |
| 192.25 | – | 1559.39 |
| 192.20 | 192.20 | 1559.79 |
| 192.15 | – | 1560.20 |
| 192.10 | 192.10 | 1560.61 |

# Alarms management



9

This chapter describes the various types of alarms, the severities, display window and actions possible (filtering, acknowledgment, resolution, logging, etc.).

Topics discussed in this chapter are as follows:

# Definition of the alarms

During network monitoring, the first event related to a system event or a monitored element opens an alarm identified by a number which will be unique until the alarm is resolved. This alarm will evolve over time, according to the events impacting this element.

An event results from any status change of an element : fault, fault disappearance, etc.

When a user deems the alarm resolved, any new event related with this system element opens a new alarm.

**Various types of alarms**

There are three types of alarms:

– the physical alarms related with fibre optics OTDR monitoring.

– the photonic alarms related with fiber optics OSA monitoring

– the system alarms related with the various system elements ONMS (RTU, server, link,…).

An alarm can include only events of the same type. If a physical alarm is ongoing on a system element, an event occurring on the photonic level on this same element triggers the creation of a new (photonic) alarm on this element.

**Severity levels of alarms**

An alarm records all the system's status changes and allocates them a severity level. Each level corresponds to a color code facilitating the use of the alarms :

| Severity | Color |
|----------|-------|
| Critical | Red |
| Major | Orange: |
| Minor | Yellow |
| Warning | Blue |
| Information | Mauve |
| Resolved | Green |

The object in alarm can include several subsets, e.g. channels for a TAP; the modules, optical switch, etc. for an RTU. In this case, the alarm's severity is the severity for the most critical subset.

**The alarm will take the level of severity :**

– from the most severe event if a photonic alarm (usually associated with a large number of events).
– from the last event detected, if a physical or system alarm, as the status of the fibre or system is stable.

# Display of the alarms

The use of an alarm has several elements intervening :

– the alarm status bar, informing the user of the arrival of a new event,
– the table listing all the alarms active and providing their main characteristics,
– the window specific to an alarm, indicating all its characteristics, all associated events, and enabling to set the alarm as being acknowledged or resolved.

**Alarm status bar** The automatically updated alarm status bar indicates :

– the number of the last alarm for which events have been detected
– the alarm's severity : the appropriate color displays in the bars.



**Fig. 127**    Example of an alarm status bar

– the type of alarm : physical (OTDR), photonic (OSA) or system (RTU, Client or Server according to the alarm's origin).

Double-click this status bar to open the alarm window describing the alarm, listing events occurred, enabling to display the location and detection traces and to generate an alarm report.

**List of active alarms** Several actions enable displaying the list of active alarms:

– Click on the 🕭 button
– Select the Operation / Active alarms menu.
– Click the bargraph blinking in the alarm status bar.

The alarms preceded with a small flag 🚩 are acknowledged.

flag: alarm acknowledged

Red text: alarm not read

Colored case indicating the alarm's severity

**Fig. 128** Example of active alarms list

The Active alarms window displays the main parameters of the alarm :

| | |
|---|---|
| Reference | Alarm identification number |
| Date | Alarm creation date, i.e. the first event related with the alarm. |
| Severity | Alarm severity . |
| Occurrence | Number of events having occurred for this alarm. |
| Resource name | Name of the system resource having detected the alarm |
| Monitored Resource Name | Name of the link or TAP monitored in alarm |
| Resource Category | Type of the resource having detected the alarm : RTU, Server, client…. |
| Resource Type | Type of the alarm : OTDR or OSA. If nothing is indicated, it is a system alarm. |

An alarm is displayed in red until its alarm window is opened to see the detail.

⚠️ In the contrary of the alarm status bar, the alarm window is not auto-matically updated. Use the button Refresh to update the window.

### Alarm notification when server is unreachable

When the server is unreachable, if the equipment is supplied with an OTU 8000 getting a GSM Modem, the OTU will notify the user directly with a SMS.

The SMS content is as follows:`[OTU NAME]-[Alarm Type]-[Time-Stamp]-[Severity]-[LinkName]-[Distance]`
Example: **OTU PARIS – Optical – 2008/03/13 12:33 – Crit-ical – Paris Lyon – 50235m**

See for the configuration of the SMS.

### Opening an alarm's window

In the active alarms table:

– either double-click the appropriate line,

– or, after selecting the alarm, click on the [icon] button.

### Transferring a resolved alarm to the log

In the active alarms table, select the alarm, then click on the [icon] .

**NOTE**

In general, alarms should be acknowledged before they are trans-fered, but this is not a requirement.

For more information on this function, see .

### Filtering alarms

To shorten the alarms list displayed, apply a filter by clicking on [icon] .
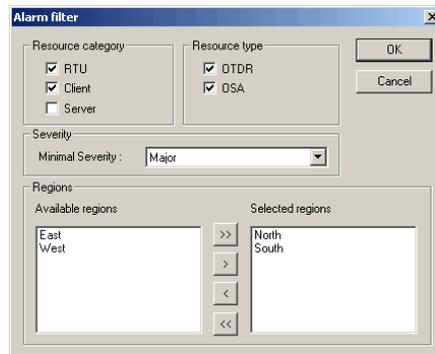


**Fig. 129**    Filtering active alarms

This dialogue box enables applying several filters:

– per type of alarm: check the boxes corresponding to the alarms to display (OSA, OTDR) and for system alarms, check the resource having issued the alarms to display (RTU, Client, Server).

– per severity: choose the minimum severity. All alarms with a higher or same severity to that chosen are displayed. For example if Major is chosen, Major and Critical alarms will be displayed.

per region: the regions available are listed on the left. Move the regions with alarms displayed to the right.

To move a region, double-click on its name or use the ⟨ or ⟩ keys. To move all regions, use the ⟨⟨ or ⟩⟩ keys.

## Window describing a physical alarm (OTDR)

A physical alarm results from the detection of a fault on a link by an OTDR measurement.

Alarm identification number

Resource at the origin of the alarm

Alarm severity

Maximum severity achieved by this alarm.

Display / Print the alarm notifications (*On duty* option)

Cause of the alarm

Optical distance from the origin

Physical distance at:
- closest upstream and downstream monitored point
- closest upstream and downstream landmarks

Window closing

Access to the alarm's physical location in Google Earth™ (option)

RTU port to which the link under alarm is connected

Alarm acknowledgment

Shift of the alarm into the «Resolved» severity

GPS Coordinates and Cable name

Attenuation in respect to the reference value, after the fault.

Initiate the Alarm report

Paste or modify an existing map (OFM or Google Earth™ option mandatory)

Display of the location or detection traces



**Fig. 130**     OTDR alarm

### General information

The upper part of the window displays the following general information:

– Alarm identification number
– RTU having detected the fault
– Link concerned by the alarm
– RTU port the link is connected to
– Current alarm severity

– Maximum severity achieved by an event of the alarm
– Alarm's date of creation, of any acknowledgment and resolution.
– By clicking on the ON DUTY REPORT button, the dialog box opened displays the user(s) to which the alarm has been sent, and allows to print the alarm notification information.

**NOTE**

The alarm takes the severity of the event with the highest severity. For an OTDR alarm, it is the last event which occurred.

### Copy / Paste a map (OFM or Google Earth™ option mandatory)

The MAP button allows to paste a map and visualize it, or to modify it if it already exists.

Paste / Modify a map:

– Go in OFM software, right click on the map and click on **Copy** in the menu

   or

– Open Google Earth™ software, adjust the view; draw the cable route and go to the **Edit** menu> **Copy** > **Copy Image**
– Go back in the ONMS software and click on the **MAP** button.
– Click on **Paste map**.

### Detailed location of the fault

The lower part of the window indicates the following information to locate the fault :

– Optical distance from the link's origin.
– Physical distance to the downstream and upstream markers, and designation of these markers.
– Physical distance to the closest upstream and downstream landmarks.
– Detection trace of the fault and associated reference, obtained by clicking on the **Detection Trace** button.
– Fault location trace and associated reference, obtained by clicking on the **Localization Trace** button.
– Click on the button **KML**, to display the alarm localization via Google Earth™, if a KML file has been associated to the link.

Localization and description of the alarm
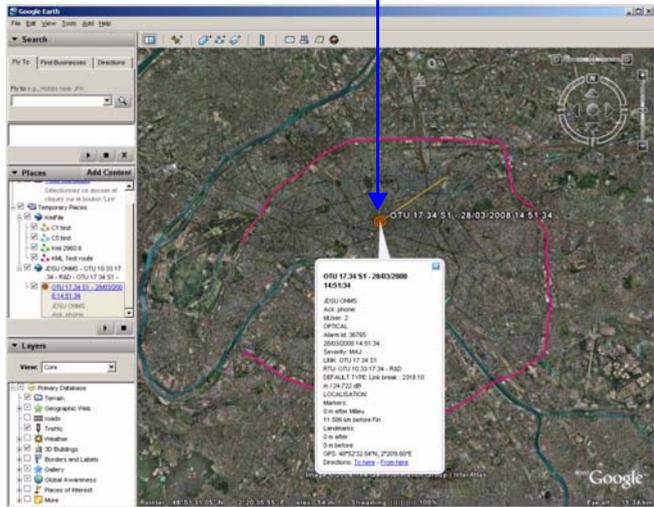


**Fig. 131**      KML link

– By clicking once on the icon representing the alarm, ◎ , the infor-
mation relative to this alarm display, among which: alarm identifica-
tion, link on which alarm occured, OTU concerned, default type,
localization...

### Events list

The last event of the alarm is described and located under the Description
tab.

Under the Event tab, the user has access to the list of events related with
the alarm, with their description and date, to follow the status of this alarm
over time. In addition to the events related to its acknowledgment or reso-
lution, the type of user having acknowledged or declared the alarm
resolved is indicated. A color square indicates the severity of events.
Transient events can appear.
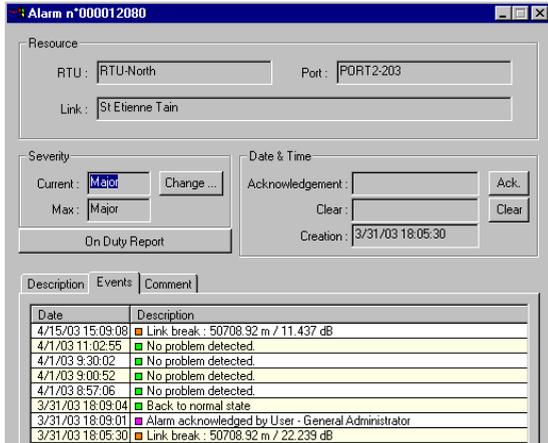
**Fig. 132**    Events list for an OTDR alarm

Double-click an event in the list to display its description, its location, and have access to the associated traces.

Click on the On Duty Report to display the users on duty to which the alarm has been sent, and / or to print the alarm notification information.
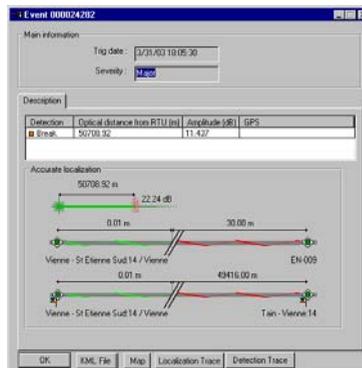


**Fig. 133**    Description of an OTDR alarm event

**Link between a fault and the events associated with this fault**

A fault can be related with several events. To know this link, select the fault in the Description window, then click the Event tab: the events related with this fault are displayed in red.

**Adding a comment to a physical alarm**

In the Comment tab, click on the Add... button and enter your comment. The entry date and name of the user having entered the comment are displayed automatically.

**Generating an OTDR alarm report**

To edit a report on an OTDR alarm, open this alarm's window from the Active alarms list and click on the Report button.

For more information and an example of OTDR report, see "Generating an alarm report" page 204.

See "Configuring reports" page 75.

**Window describing a photonic alarm (OSA).**

A photonic alarm is issued whenever an OSA measurement detects a threshold was exceeded.

**General information**

The upper part of the window displays the following general information:

– Alarm identification number
– RTU having detected the fault
– RTU port connected to the TAP in alarm
– Name of the TAP in alarm
– Equipment in alarm
– Port of the equipment in alarm
– Severity of the last event and maximum severity achieved for this alarm.
– Description of the last event.

**NOTE**

The OSA alarm takes the severity of the event with the highest severity.

See "Acknowledging and resolving an alarm" page 202.

**Description of the OSA alarm**

The following displays under the Description tab:

– The parameter currently under alarm and its last value measured.

–　The Channel Measurement field displays the results outside the OSA measurement template for each channel, their wavelengths and characteristics (the color code indicates the parameters outside the template and the level of severity of the fault). For each channel, the owner and client organizations are also indicated.

–　The Channel movement field displays: the new channels detected in the spectrum, as well as those no longer detected.

The Spectrum key enables displaying the spectrum for the current result.

### List of OSA alarm events

Under the Events tab, the user has access to the list of events linked with the alarm, with their date and description. The alarm's acknowledgment or resolution is indicated, with the type of user having performed the intervention. A color square indicates the severity of events.

To display the spectrum for an event, select this event, then click on the Spectrum button.

Click on On Duty Report to display the users on duty to which the alarm has been sent, and / or to print the alarm notification information.
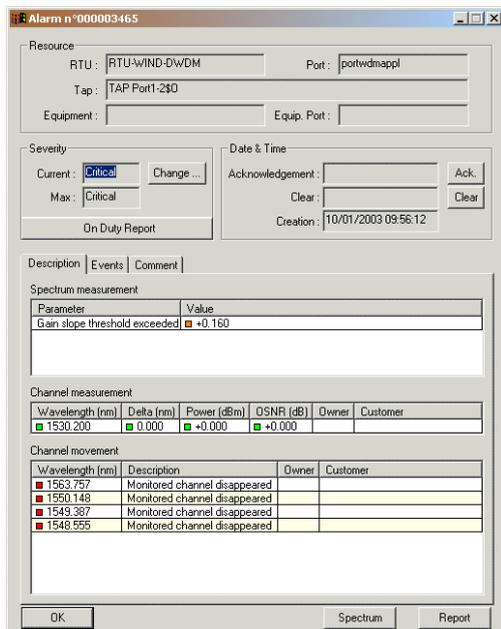


**Fig. 134**　　　OSA alarm

**Link between a faulty channel and the events associated with this fault**

A fault on a channel can be related with several events. To see this link, select the channel in the Description tab, then click on the Events tab: the events related with this channel are highlighted (red).
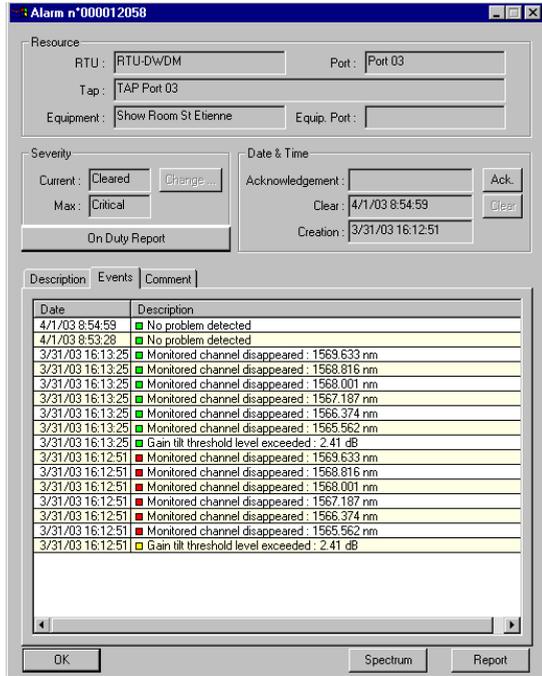


**Fig. 135**     OSA alarm - Events

**Generating an OSA alarm report**

To edit a report on an OSA alarm, open this alarm's window from the Active alarms list and click on the Report button.

For more information and an example of OSA report, see "Generating an alarm report" page 204.

**Locating an OSA alarm on a map background**

The Photonic layer window opened displays the RTUs connected to the equipment access points. Faulty RTUs are circled. To display details on the alarm, perform the following :

– Right-click the RTU under alarm
– Select the Alarms line in the menu displayed. A window displays the list of current alarms on this RTU (with the RTU name and alarm severity).
– To call the window for this alarm, select one of the alarms, and click on the Display alarm button (or double-click on the line corresponding to this alarm).

**Window describing a system alarm**

A system alarm is triggered by one or several faults on one of the system's elements: RTU, Client station or Server.

In the active alarms table, alarms with a type not indicated are system alarms. Click on an alarm to open its window.

### General information

The upper part of the window displays the following general information:

– Alarm identification number
– Name of the resource in alarm
– Category of this resource (RTU, Client station or Server)
– Current alarm severity
– Maximum severity achieved by this alarm.
– Alarm creation date and time
– Alarm acknowledgment and resolution date.
– Description of the alarm.

▬▬▬ | **NOTE**

The System alarm takes the severity of the event with the highest severity.

See "Acknowledging and resolving an alarm" page 202

### System events list

Under the Event tab, the user has access to the list of events related with the alarm, with their date and description : this provides this alarm's history. The alarm's acknowledgment or resolution is indicated, with the user having performed the intervention. A color square indicates the severity of events.

### Alarm -> Event link

A system alarm can be associated with several faults. To display the events related with a fault, under the Description tab, select this fault, then click on the Event tab: the events related with the fault are displayed in red.
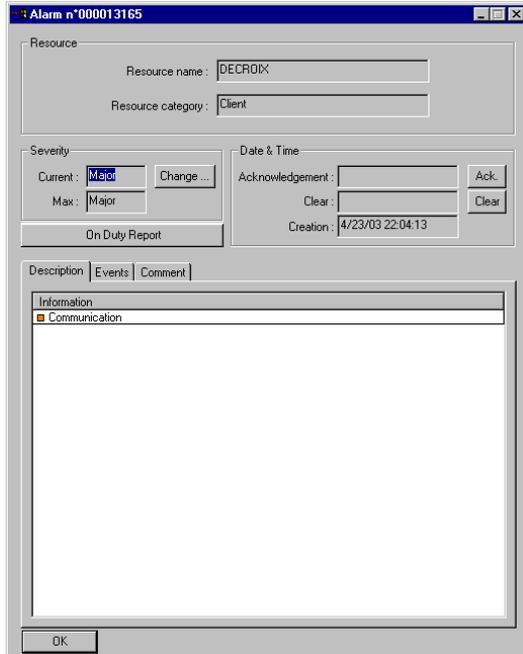


**Fig. 136**   System alarm

Click on ON DUTY REPORT to display the users on duty to which the alarm has been sent, and / or to print the alarm notification information.

**Adding a comment to an alarm**   Regardless of the type of alarm, its window includes the Comment tab. Under this tab, click on the Add... button, and enter your comment.

The entry date and name of the user having entered the comment are displayed automatically.

# Alarms history

When the alarms are resolved and acknowledged, we recommend archiving them in the HISTORY.

**Transferring alarms into the history**

Alarm archiving can be either manual or automatic. To archive resolved alarms manually:

**1**  Click on the ⏰ icon to display the ACTIVE ALARMS window.

–  Select the alarms to transfer.

–  Click on the 🖫 icon

This operation can be performed automatically, with a period defined by the system administrator in the CONFIGURATION > SERVER > ALARMS CONFIGURATION menu (see "Configuring the server" page 68).

**History description**

Archived alarms can be viewed, at any time, by clicking on the icon 🗗.

> **NOTE**
>
> If the history does not display any archived alarms, check that filtering has not hidden them from the table (see "Filtering archived alarms" page 200.



**Fig. 137**     Alarms history

In the history, the alarms are displayed with the following information :

–  Alarm identification number

–  Alarm creation date

–  Alarm duration (time elapsed between its creation and resolution)

–  Maximum alarm severity

–  Name of the resource monitored

–  Category of the resource monitored

–  Type of monitoring (OTDR, OSA, etc.)

At the bottom of the table, a message indicates whether the alarms are filtered.

All archieved alarms may be sorted using the column criteria (click on the column name to sort alarms according to the criteria).

**Filtering archived alarms**

To shorten the list of alarms displayed in the history, filter them, by clicking on the 🧹 button (see ).

**NOTE**

Unless in the active alarm list, the alarms history cannot be filtered according to the region, a ll the existing regions will be displayed.

**Displaying the detail of an archived alarm**

For more information on an archived alarm, in the Alarms log window:

– either double-click the appropriate line,

– or, after selecting the alarm, click on the 📇 button.

In addition to its identification, maximum severity and persistence, a diagram shows, using the color code, the evolution in the alarm's severity over its lifetime.

The following then displays :

– for the OTDR alarm: the description and main characteristics

– for the photonic alarm: the list of spectrum channels and alarm severity for each

– for the system alarm: the resource concerned (RTU, Client or Server), and the alarm's description.

**Displaying the traces associated to an archived OTDR alarm**

In the Alarms History window, double-click on the OTDR alarm concerned, then :

– click on the Localization trace button to view the trace used by the system to locate the fault.

– click on the Back to normal trace button to view the OTDR trace after fault resolution.

**NOTE**

In an OTDR alarm detailed, the MAP button allows to paste a map from the OFM and to visualize it, or to change a map if it already exists.
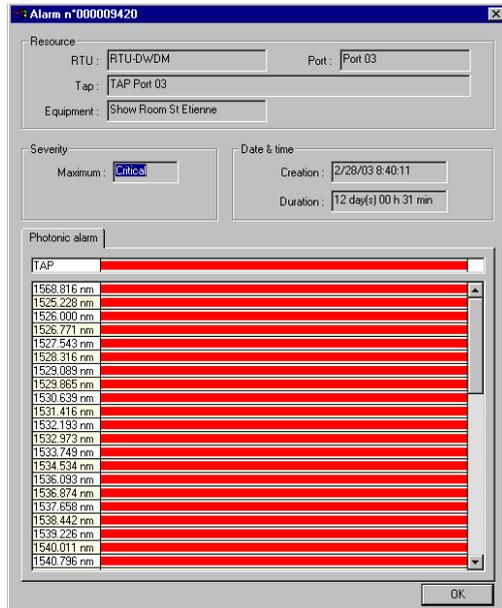
**Fig. 138** Detail of an OSA alarm archived in the history

**Deleting alarms from the History**

In the Alarm History, you can remove one or several alarms displayed, whatever the severity of the alarm.

– Select the alarm(s) to be purged.

**NOTE**

For a multi-selection, maintain the Ctrl key pushed and click on each alarm. For a list of alarm, select the first one and maintain the MAJ key pushed and click on the last one from the list.

– Click on the button  on the right of the table.
– Confirm the action in the dialog box.

**The alarms are removed from the history table and defenitively deleted from the database.**

# Acknowledging and resolving an alarm

### Acknowledgment

> **NOTE**
>
> Only users with the appropriate privilege can acknowledge an alarm.

To acknowledge an alarm, open its window, and click on the ACK button :

– in the Active alarms list, a small red flag displays next to the acknowledged alarm.

– an acknowledgment notification is sent to all client stations. The alarm can no longer be acknowledged by other users.

Each new event related to the alarm will reactivate it, and it must be acknowledged again by a user.

### Resolution

Most alarms are resolved automatically upon disappearance of the fault (after fixing or configuration modification). However, certain alarms (e.g. hardware modifications) cannot : the user must resolve them manually.

To declare an alarm resolved, click on the Res button.

When an alarm is declared resolved, it is closed for the system, and can no longer receive new events and change status. If a new fault occurs on the element which was under alarm, the system creates a new alarm.

Manual resolution of an alarm acknowledges it automatically.

# Locating RTUs in alarm

You can view the RTUs in system alarm or which have detected a physical (OTDR) or photonic (OSA) alarm.

In order to do this, click on the  button calling the Photonic Layer : a map background displays, locating the RTUs.



The RTUs in alarm are circled, the color representing the severity of the alarm detected.

To display the list of alarms detected by an RTU and their description, after a right-click on the RTU, select the Alarm menu.
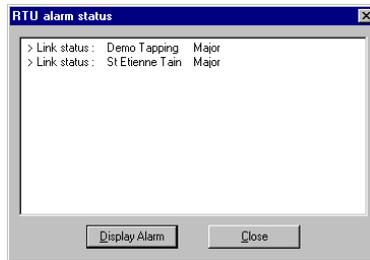


**Fig. 139** List of alarms detected by an RTU

To view the detail for an alarm, select it in the list, then click on the Display alarm button.

# Active alarms dashboard

Click on the [icon] icon to display a dashboard indicating the number of alarms active, filtered and non filtered according to their severity



**Fig. 140** Example of alarms dashboard

Reading is facilitated by the color code (severity) and a system of the bargraph type, indicating the proportion of non filtered alarms.

# Generating an alarm report

The network administrator has defined the contents of the OTDR alarm report: header, logo, information on the alarm (general or detailed, comments); see "Configuring reports" page 75.

To generate the report for an alarm from a Client station, click on the **Report** button present in its alarm window (see "OTDR alarm" page 190 and "OSA alarm" page 195).

This report can then be printed, or saved in a file in the html format:

– To print it, click on the Print command, and define the printing parameters.
– To save it as a file in the html format, click on the Save as... command, and select the destination directory and the file name.
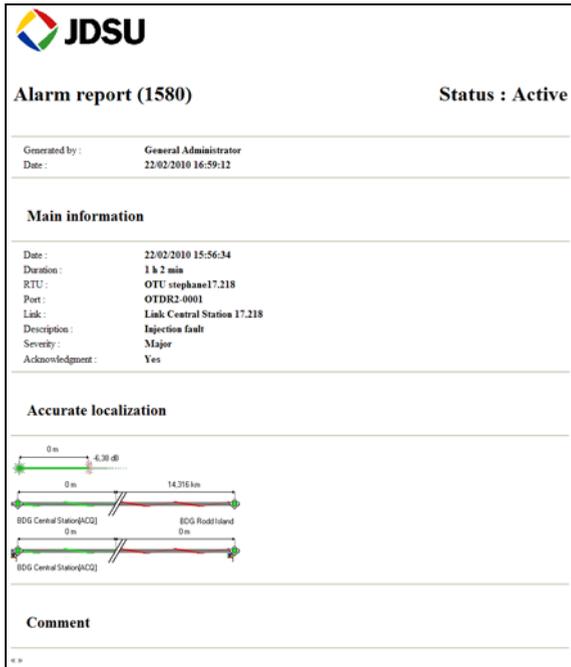


**Fig. 141**     Example of alarm report (OTDR)- part 1

**Link : Link Central Station 17.218**

| | |
|---|---|
| Port : | OTDR2-0001 |
| Status : | MONITORED |
| Schematic : | see attached file : AIO_2010-02-22_16-59-12_1580_Syn.jpg |
| Comment : | * * |

**Geographical localization (GPS: 33°52'54,98"S, 151°12'24,60"E)**

| | |
|---|---|
| Cable : | C8:11 |
| KML : | see attached file : AIO_2010-02-22_16-59-12_1580_.kml |



**Reflectometry**

| | |
|---|---|
| Reference date : | 20/01/2009 10:52:20 |
| Equipment : | OTU 8000 Num.50 |
| OTDR : | 8117 VLR Num.2897 |
| Wavelength : | 1625 nm |
| Pulse : | 100 ns |
| Acq. time : | 10 s |
| Range : | 143,25 km |
| Resolution : | 5,00 m |
| Index : | 1,465 |
| Trace with reference : | see attached file : AIO_2010-02-22_16-59-12_1580_Ref.jpg |

**Fig. 142**     Example of alarm report (OTDR)- part 2
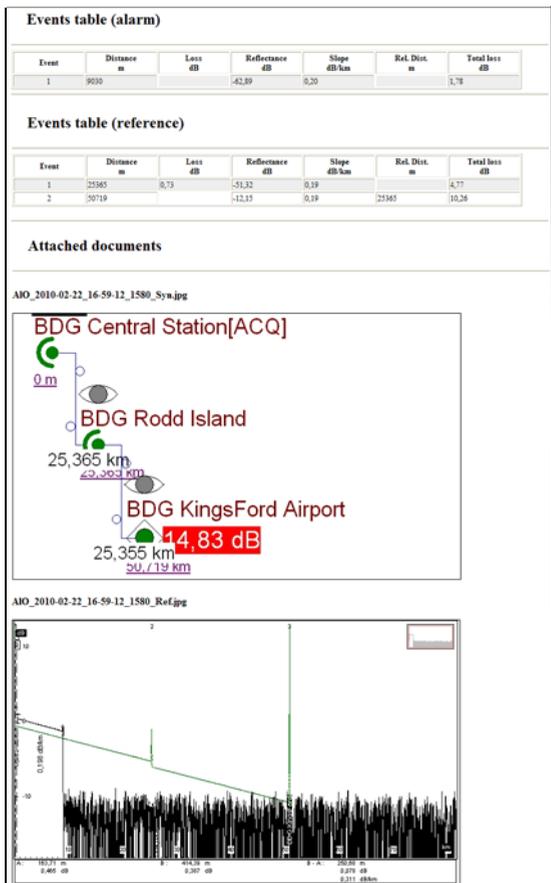
**Fig. 143** Example of alarm report (OTDR)- part 3

# Exporting the alarm history to Excel

The export of the alarm history to excel allows the customer to generate its customized report using Excel.

All the attributs of the alarms are exported in xls format.

To export the alarms history to excel:

**1** In the Report menu, click on **OTDR alarm history report**.

A dialog box opens, enabling to select the region to take into account, the link to select and the time period covered by the report.



**Fig. 144**    Example of selection for alarm history report

**2**    Click on OK once all the parameters have been selected.

**3**    In the **Save as** dialog box, select the directory in which the xls file will be saved, and enter its name.

The xls file is generated.

The file countains  by default 2 sheets: **Alarm history** and **Fault History**.

If the parameter **Export sample history**, the file will also include the sheet **Sample history**.



**Fig. 145**    OTDR Alarm History Report - *Alarm History*

**Fig. 146**       OTDR Alarm History Report - *Fault History*



**Fig. 147**       OTDR Alarm History Report - *Sample History*

**If the parameter *Export sample history* is selected, the loading of the xls file may be longer.**

# Generating a performance report

A performance report enables summarizing the alarms in the physical and photonic layers, over a given period.

> **NOTE**
>
> This report relates to the <u>alarms transferred into the Log</u>, and not to active alarms.

The network administrator has defined the contents of the OTDR and OSA performance reports: See "Configuring reports" page 75.

### Generating a performance report

**1** To generate a performance report, in the Report menu, select the type of report.

A dialogue box opens, enabling to select the time period covered by the report, and its object.



**Fig. 148** Example of selection of a performance report

– For an OTDR performance report, select: one or all regions, one or all links:

**2** After selection, click on OK.

A message specifies that the report is being generated, and the report is displayed.

**Examples of performance report**
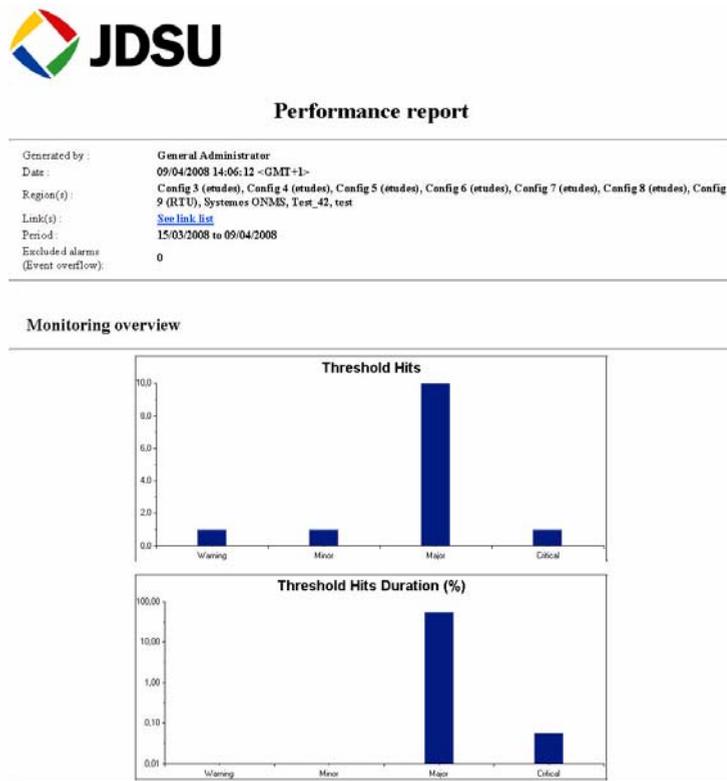


**Fig. 149** Example of OTDR report (part 1)

Breakdown of thresholds hits by duration
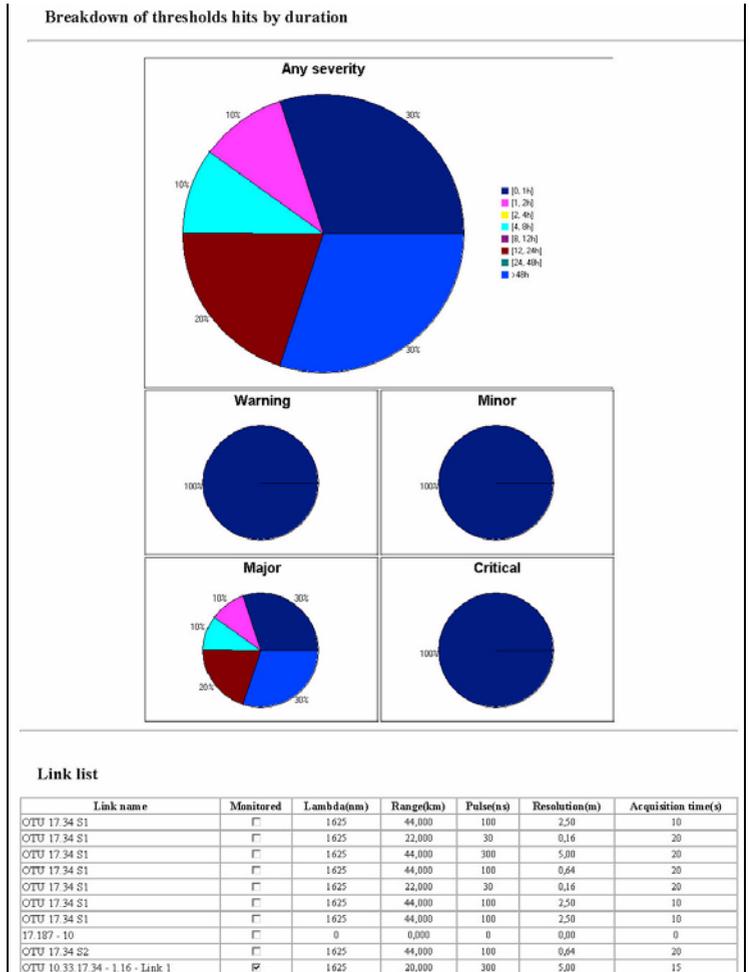


**Fig. 150**     Example of OTDR report (part 2)

### Processing the performance report

This report can then be printed, sent by E-mail, or saved in a file in the html format:

– To print it, click on the Print command, and define the printing parameters.

– To save it as a file in the html format, click on the Save as... command, and select the destination directory and the file name.

# Analyzing OTDR and WDM Traces

10

---

**NOTE**

For the analysis of theOTDR and WDM curves, refer to the user manual of the JDSU FiberTrace OFS 100 software.

Analysis of the reflectometry curves by the ONMS is similar to that performed by the Fiber Trace software, except for a few details described in this chapter, and concerning:

# Opening an OTDR trace

The OTDR trace displayed can be a trace just acquired or can be one stored in memory.

To load an OTDR trace stored in memory:

**1** Click on **Acquisition** menu, then on **Open OTDR trace...**

A new dialog box opens.
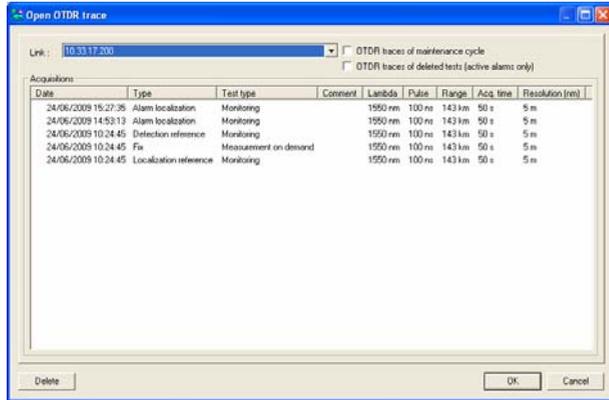


**Fig. 151** Open OTDR trace dialog box

**2** Select the parameter **OTDR traces of maintenance cycle** in order to display and open the traces of maintenance cycles.

**3** Select the parameter **OTDR traces of deleted tests (active alarm only)** in order to display and load the traces of deleted tests, for traces with an active alarm only.

**4** Select the trace to be opened.

Press **Ctrl** to select multiple traces.

**5** Press **OK** to open the OTDR trace.

If several traces have been selected, each trace is opened in a different window.

# Cursor mode S
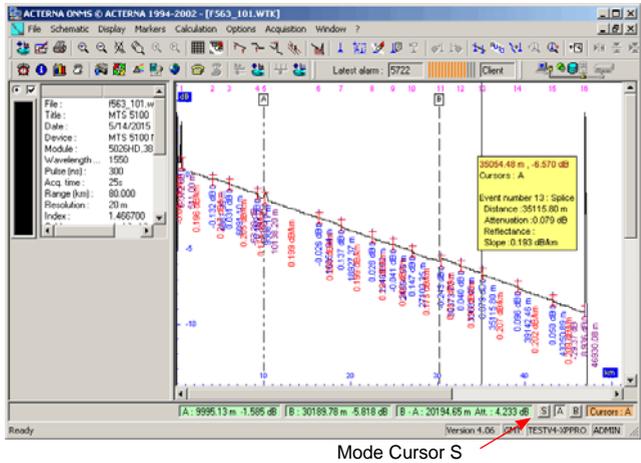


Mode Cursor S

**Fig. 152**    Trace analysis window

Cursor mode S (diagram marking) enables linking the diagram with the curve (see "Link between schematic and OTDR trace" page 154).

# Trace information

The Color and Launch cable options are not used in the ONMS.

The following information and parameters ( unused in Fiber Trace) are available with the ONMS:

| | |
|---|---|
| Link | Link identification |
| Test [1] | Type of test: Monitoring, Maintenance, TTL, Acquisition on demand, Manual (imported trace). |
| Type[1] | Type of acquisition: Detection reference, Location reference, Completion (Traces measured and stocked during the maintenance cycle), Original reference, Previous alarm, Detection alarm, Location alarm. |

---

1.only appear for the ONMS traces in the database

# Reference trace function

Use the following procedure to open a reference curve for an open curve:

1. Select Reference curve…in the Displaymenu.

   The following dialogue box enables selecting the reference curve search mode:
   - either automatically in the database.
   - or manually in the database.
   - or manually outside the database (file).



**Fig. 153**      Selecting the origin of the reference curve

2. Searching and opening the reference curve.

**NOTE**

If the reference curve selected is not compatible with the active curve displayed (different indexes, different devices, or different pulses), an alarm message displays, and you can cancel or resume the operation.

The window displays the initial curve, as well as the reference curve: the reference curve displays in green to be differentiated with the active curve.

Any cursor or marker and any calculation performed apply only to the active curve. The reference curve can be used only for comparison purposes.

It is used for Both End measurements.

# Both end measurement function

With the ONMS system the Both End measurement function uses the Reference trace function described below.

The traces obtained form the two measurements (one in each direction) are displayed with event markers. They are first displayed superimposed, then the E -> O trace is inverted (left $\leftrightarrow$ right), in order that the markers associated to the same events are on the same horizontal position.

The Both End measurement result table show the following loss types:

–   The O $\rightarrow$ E and O $\leftarrow$ E losses correspond to the measurements made in each direction. They are different because backscattering coefficients of the segments situated at either side of the event are different.

–   The last loss is equal to half the sum of the two previous losses. By thus taking the average value of both previous results we eliminate the error due to the difference of the reflectance thresholds from either side of the event. Therefore this loss corresponds to the real value of the insertion loss of the event.

Use the following procedure to make a Both End measurement:

**1**   Open the trace file of the measurements made from one of the extremities of the fiber you want to measure (preferably O->E).

**2**   Use the Reference trace ... function from the Display menu to select the trace file of the measurements made from the other extremity of the same fiber (preferably O<-E): see "Reference trace function" page 216.

**3**   If necessary select the type of inversion for the O<-E trace, in the Display / Preferences menu.

**4**   If necessary choose to insert in the analyze the launch cables in the box called by the File > Edit file information menu > Cable.

**5**   Select the Both end measurement command from the Display menu.

The O $\rightarrow$ E and O $\leftarrow$ E traces are displayed. They are automatically shifted to align the corresponding events.

# On-duty management option

# 11

The On-duty Management option (Ref. 95NT/Alarm) is used to call on-duty personnel. It exists in two versions:

– 95NT/Alarm: the user can be contacted by Pager, mobile phone, fax, or E-mail.

– Emaildisp: the user can be contacted only by E-mail

This chapter describes the following functions:

# Alarm management principle

When an alarm is detected, a call is sent to on-duty person according to a roster defined by region.

For each on-duty user, you can define the minimum severity from which he is contacted (see "Defining on duty levels" page 43).

The designated person is informed by a message on a Pager, mobile telephone, fax or only e-mail according to the version (see below). These different means of notification are defined in the Media directory of each user.

When the user has read the message, he must:

**1** log onto the ONMS system, either directly using a portable PC or by telephone. In this last case the user must call the telephone number of the server given in the alarm message (it was entered when setting up the server: see "Entering the telephone number of the modems" page 70).

**2** acknowledge the alarm using the identifier given in the message.

The same user may be called by different means of communication (maximum of four). Each of them may be defined as primary or secondary. The secondary numbers are only used if the message cannot be left with the primary numbers.

For each on-duty user, the name of a backup user can be given. This user will be called if the first user has not acknowledged the call or the alarm after a certain period defined in the server parameters (default value - 60 minutes). The backup user is also called if the message cannot be left when the first user is called. The name of the backup user is given in the roster.

The roster is also used to indicate the name of a replacement for a particular period if the selected user is occasionally unavailable. In this case, the message will be sent directly to the coordinates of the replacement.

You can also define a user who will be on-duty at all times, and receive messages by fax, E-mail, or sms. The user can be non generic, such as a supervision room, to which messages are faxed.

# Commands for the On Duty option

When the ONMS system is equipped with the On Duty option, the user has access to the following additional commands in the Administration function:

– accessed via the Region directory and the Region tab: a button to call up the On duty calendar for the region (see "Description of the On duty calendar" page 222).

– a Medias subdirectory for each user with an On Duty profile: it contains the parameters of the media allowing to contact the user (max. 4).

A user's on-duty scope is all his authorizations (regions, links, TAP, and organizations): See "User authorizations" page 40.

Medias subdirectory containing the characteristics of the contact media for the On Duty user.

Access to the On duty calendar
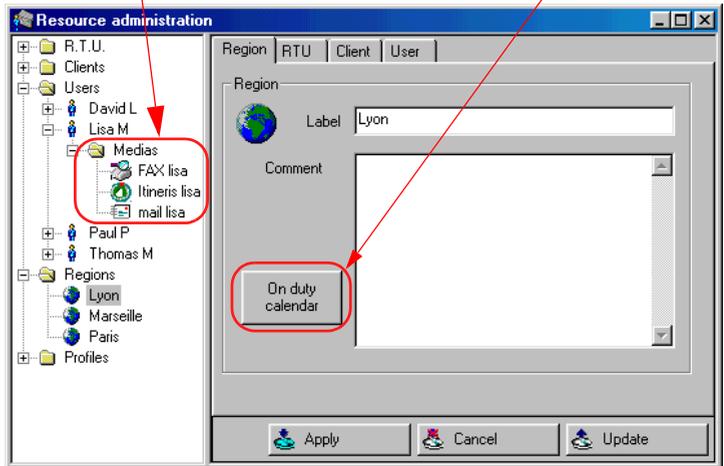


**Fig. 154**      Administration parameters for the On Duty option

Once the On duty calendar button has been pressed and the calendar is displayed, the following commands are available:

– an On Duty Calendar menu in the menu bar giving access to all the On Duty commands

– a shortcut menu obtained by a right-click on a period on the calendar allowing to perform standard operations for this period.

**Shortcut menu**
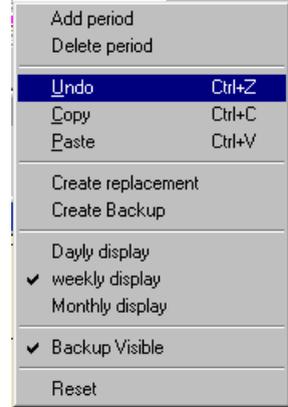
**On Duty Calendar Menu**

– an On Duty Calendar toolbar which can be displayed and moved like the other tool bars (see "Tool bars and activities" page 23). This bar provides rapid access to the different On Duty commands.
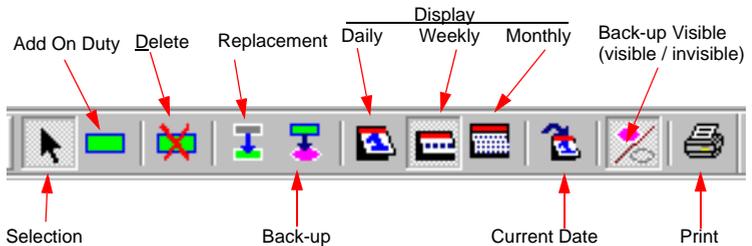
**Fig. 155**    On Duty Calendar toolbar

# Description of the On duty calendar

Click on the Region directory, then on the [On duty calendar] button: the calendar is displayed for the current week (centred on the current day).

– To display the calendar for the day or the month, click on the [icon] or [icon] button.

– To return to the weekly display, click on [icon].

This calendar is a chart with persons or external media for the region in the vertical axis and time (days or times depending on the type of display) in the horizontal axis.

Different periods are identified as follows: bank holidays shaded green; week-ends shaded yellow. These periods can be modified: see "Standard configuration of calendar" page 226.

To scroll the display window on the time scale, use either the lower cursor or the arrow keys on the keyboard.

If the display window has been scrolled, to return it to the current date click on  .

Name of region   On Duty users or medias   Legend to symbols for periods that are used   A double click on a day displays the daily calendar for this day



Active Duty   Backup Duty   Replacement Duty   week-ends/bank holidays shaded   Scrolling of image area
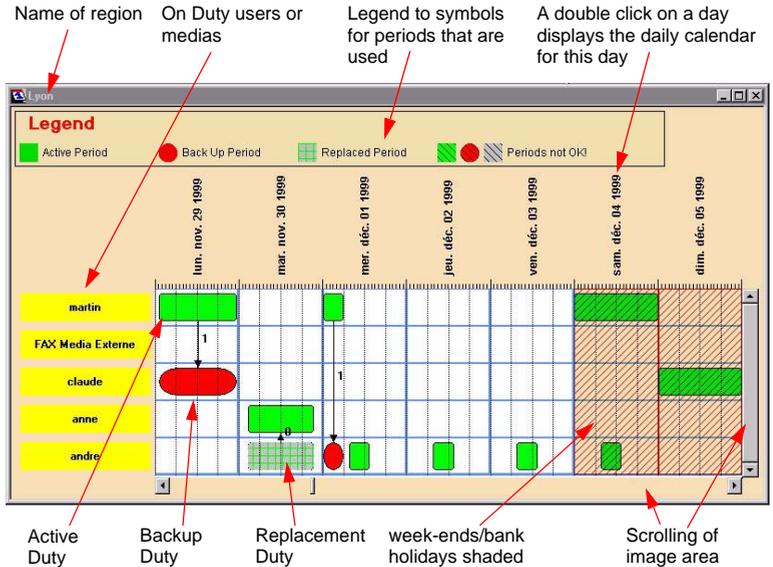
**Fig. 156**    Example of a weekly On duty calendar

### Period covered by the calendar

– The calendar stores the last 12 months by default. The period saved can be modified: see "Standard configuration of calendar" page 226.
– The calendar covers up to four years, including the saved months. When 12 months have been saved, there is still three years left in the future.

### Other types of display

– The daily display will show details of the On Duty times. It is called up by a double-click on the day in question in the weekly display (see "Example of a weekly On duty calendar" page 223) or by the Daily display command  in the bar or menu.
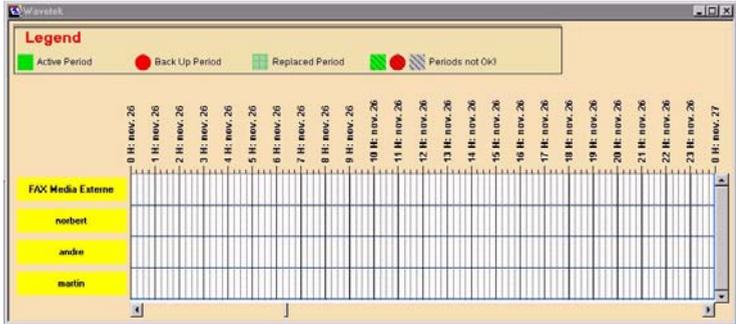
**Fig. 157**    Example of daily calendar

– The monthly display gives an overall view of the different duties for the month.
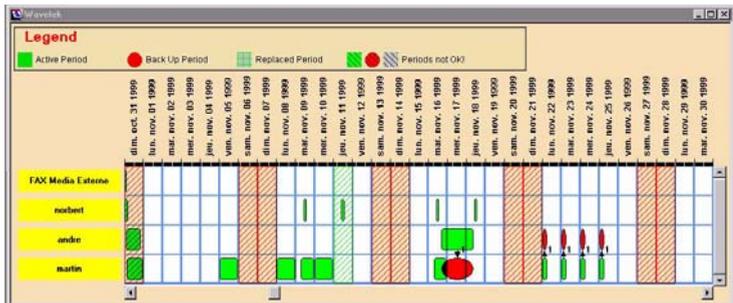


**Fig. 158**    Example of monthly calendar

## Definition of Duty periods

### Types of duty

There are three types of Duty periods:

| | |
|---|---|
| active period | during which the main user is called directly. |
| backup period | during which another user will be called if it has been impossible to contact the main user or if the main user has not responded to the call. This period is related to the active period. It is possible to define how much time should elapse before using the backup: see "Standard configuration of calendar" page 226. |

replacement period: during which the duty is transferred to another user if the main user is unable to assume duty responsibility temporarily. This period is related to the active period.

The Duty periods are indicated by symbols on the calendar: see .



Continuous duty          Periodic duty          Period selected
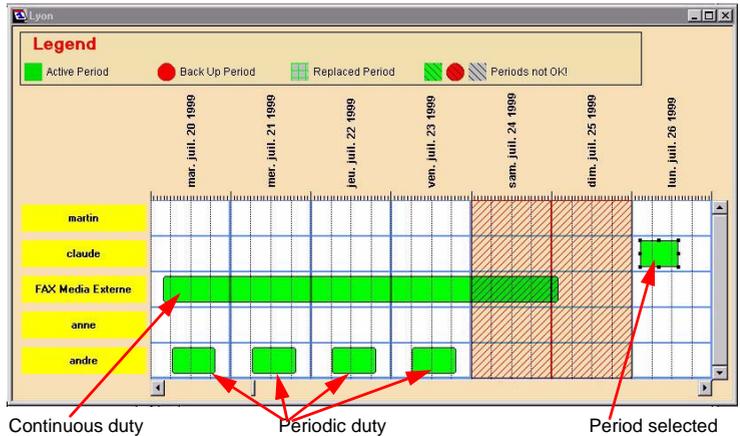
### Failed duty periods

A Duty period, irrespective of its type, may be **Periods not OK!** if all the primary media assigned for this period have been deleted, so that no calls are possible. The selected periods are shaded on the calendar. A period that is not okay is displayed with diagonally crossed lines.

### Continuous or periodic period

An On Duty period may be continuous or periodic:

– continuous period covers a single period of time
– periodic period is repeated according to predetermined times on selected days.

**Switching to Selection mode in the calendar**

Selection mode can be obtained in one of two ways:

– By double-clicking in an empty zone in the calendar.
– By clicking on the ![button] button in the menu bar.

**Cancelling an action**

The Cancel command in the On duty calendar menu can be used to cancel the last action if it was one of the following:

– addition of periods,

– copying of periods,

– creation of backup or replacement periods

⚠️ An action Delete can never be cancelled.

**Updating the calendar from the database**

The calendar can be modified by the administrator or by a qualified user from another Client server.

To update the calendar from the database, choose the Update command in the On duty calendar menu.

**Standard configuration of calendar**

To select how much time should elapse before transferring to the backup user and to select bank holidays, which differ from one country to the next, select the Server option in the Configuration menu, then the On Duty tab:
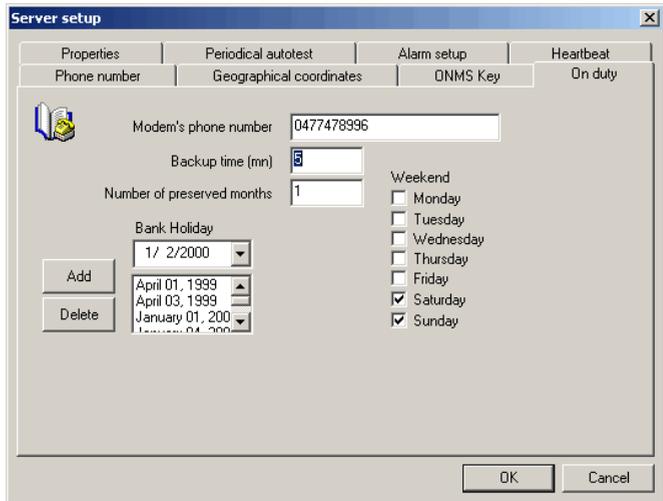


**Fig. 159** Server configuration of On duty calendar

– select the Back Up time during which the person on duty should be called before transferring to the backup user (default value 120 minutes).

– select the number of past months to be saved (default value 12 months).

– indicate the days of the week not worked under the heading Week End: these days will be identified by yellow shading.

– as required, select the bank holidays: these days will be identified by green shading.

### Recording bank holidays

– In the Bank Holiday field, click on ▾ . The following window will be displayed to select the day:



Select the month

Select the day in the month

Date selected
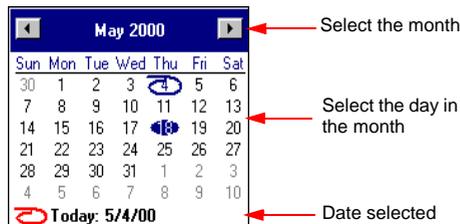
**Fig. 160** Selection of bank holiday

– Scroll the months using the ◀ or ▶ key; or click on the name of the month to display the 12 months and select one.

– Click on the year if necessary to modify it.

– Click on the day to select.

– Click on Add. The day is added to the list.

### Deletion of bank holidays

To delete a bank holiday from the list: select the day and click on Delete.

## Adding an On Duty user

**Configure the user**

1 Click on the User directory.

2 Choose an existing user or add a new user (see "Adding a user" page 38).

3   Under the Authorization tab, define the user authorizations (regions, links, TAPs and organization): see "User authorizations" page 40.

4   Under the Onduty tab, select the type of event for which the user is warned (see "Defining on duty levels" page 43).

**Define the media for contacting the user**

1   Click on the Medias subdirectory for this user.

2   Call the Add command ▪▄▄ . The dialog box Media detail will be displayed:
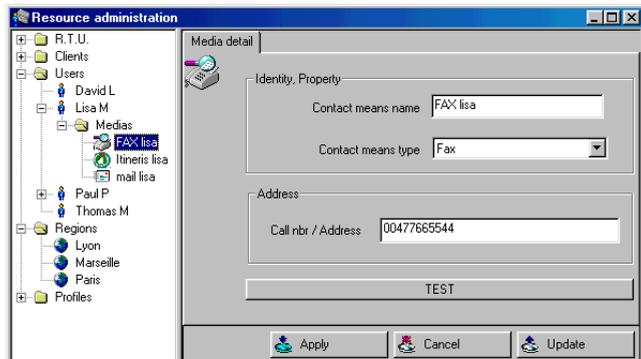


**Fig. 161**      Media detail

–   Give a name to this medium. This name will be used for the medium in the duty period definition boxes.

–   Select the type of medium in the scrolling list (fax, e-mail, etc...). This list provides access to all the media compatible with the system. The logo of the medium selected will appear in the upper left-hand corner of the window, if a logo exists. If there is no logo for the medium selected, a standard logo is used.

–   Type the n° or address of this medium.

–   Click on the TEST button if you wish to check that the information has been successfully sent to the server and wait for the confirmation message.

–   Record the medium in the database ( ▪▄▄ ).

–   Follow the same procedure to enter the different media for contacting the user (maximum of 4 media).

–   Click on the Medias directory to display the list of media (name, address and type).
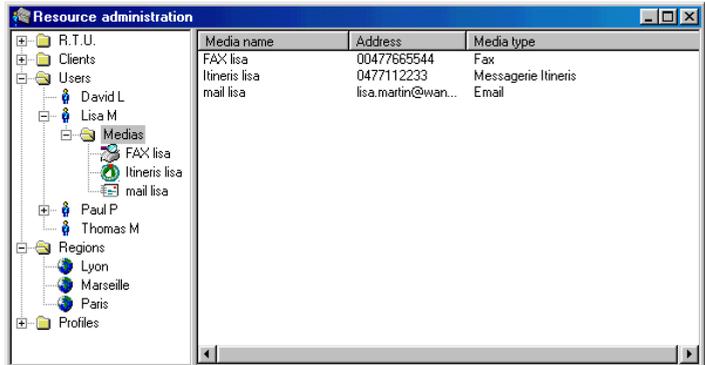
**Fig. 162** List of contact medias for a user

Once the user has been configured in this way, he can be assigned an On Duty period.

# Defining the Duty periods for a region

**Adding a new active Duty period**

**1** Open the Region directory and select the region concerned by the duty.

**2** Call up the calendar by clicking on the On duty calendar button. The calendar will be displayed with all the users on duty for this region.

**3** If necessary, move the display window in the calendar to show the time period concerned by the duty. If necessary, click on  so that the current date appears in the display. Depending on the length of the period to be defined, stay in weekly display or select daily or monthly display.

**4** Click on the  button, then, on the line corresponding to the selected user, drag the mouse to draw a rectangle on the desired period of time. The On duty period definition window will be displayed.
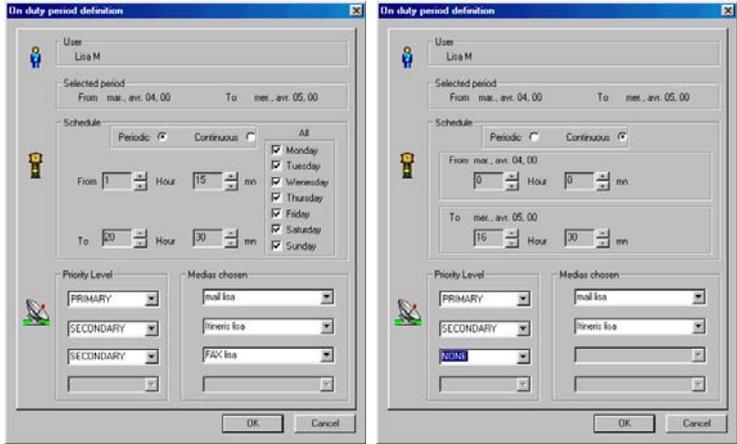
**Fig. 163**      Definition of an On duty period (periodic and continuous)

**5**   If the period chosen covers several days, continuous or periodic duty can be chosen in the Schedule field. In both cases, choose the start and end times for the Duty period. For periodic duty, choose the days in the week when the person will be on duty during the times selected.

**6**   Choose one or several contact media to be used for contacting the person on duty (4 contact media are possible). A scroll window proposes the different media which can be used to contact the user. They have been defined in the Medias sub-directory of the User directory, under the user's name.

**7**   For each contact medium selected, choose a priority level: PRIMARY or SECONDARY

   –   The PRIMARY contact media are used first.
       If they fail (for example if an e-mail cannot be sent because there is a network problem), the SECONDARY media are used.

**NOTE**

There must be at least one contact medium identified as Primary in order to be able to record the duty period.

**8**   Click on OK. The duty period that has just been defined will appear on the calendar.

Active          Backup          Replacement          Continuous          Periodic periods
period          period          period               period (selected)
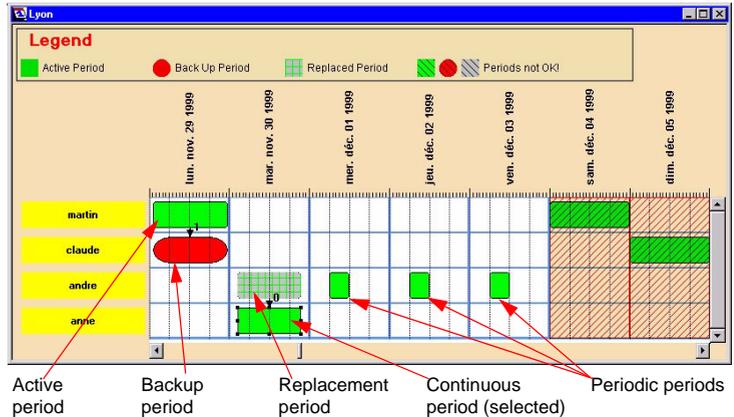
**Fig. 164**     Examples of duty periods

**Selecting a duty period**

– To select a duty period in order to delete it or to add it a backup or replacement period, choose the selection tool ▶ then click in the period on the calendar. The selected period has a frame with anchor points.

– To select several periods, once the selection tool has been chosen, hold down the SHIFT key and click the periods to be selected. Periods can also be selected by dragging the mouse to draw a frame around the periods to select.

**Copying a duty period**

**NOTE**

A duty period can only be copied to the same user since it contains parameters (contact media) specific to this user. If an attempt is made to copy it to a different user, the copy is forced to the original user, at the level indicated

**NOTE**

When an active period is copied, any related backup or replacement period will also be copied.

Several periods can be copied simultaneously. Copying can be performed in several different ways.

### Copying a period using «copy-paste» in the shortcut menu

**NOTE**

This method is recommended since it is the most accurate and the fastest.

– Select the period(s) to be copied.
– Click the right mouse button and choose the Copy command.
– Click in the calendar at the point where the information is to be copied.
– Click the right mouse button and choose the Paste command: the information is copied at the insertion point.

### Copying a period using Copy-Paste in the On Duty menu or using Ctrl C + Ctrl V

– Select the period(s) to be copied.
– Press Ctrl+C then Ctrl+V: the information is copied at the beginning of the display window.

## Deleting a duty period

A selected period can be deleted using the Delete period command in the bar or in the On duty menu or by using the Delete key:

– If an active period is deleted, any related backup or replacement period will also be deleted.
– If a replacement period is deleted, the replaced period is automatically reactivated.

## Assigning a backup duty period

To assign a backup period to an active period:

– Select the active period (see "Selecting a duty period" page 231).

**NOTE**

The backup can also be assigned to several active periods selected simultaneously provided they belong to the same user

– Select the function Create Backup in the toolbar or in the menu. The dialog box User will open. A scrolling list will be displayed of all the users in the region who have been assigned the On Duty profile.
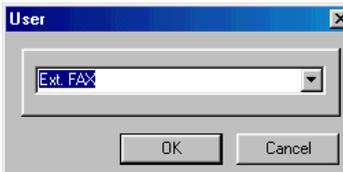
**Fig. 165** User Dialog box

– Select the backup user and click on OK. The On duty period definition box will open: the scheduled times of the active period are already entered.
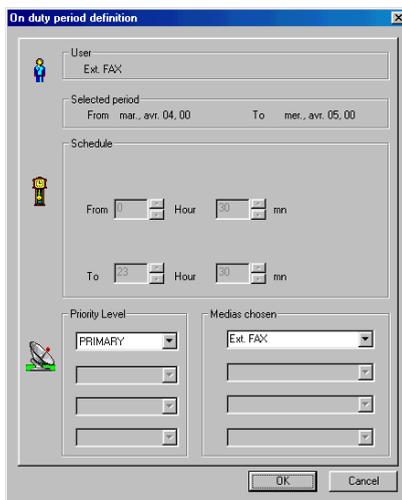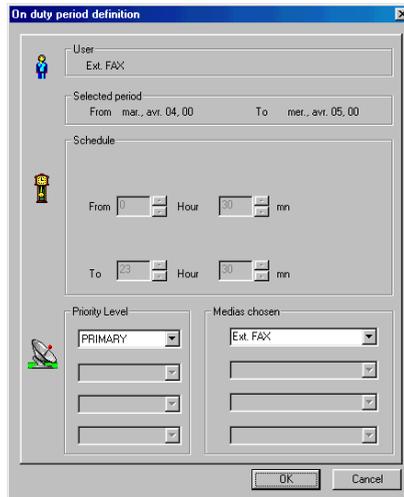


**Fig. 166** Assigning a backup duty period

Simply choose the contact media to be used and their priority level, as when defining the active period (see page 230).

– Click on OK.

The backup period(s) are displayed with their link to the active period.

**NOTE**

To avoid overloading the calendar and making it difficult to read, it can be displayed without the backup periods. To do this, invalidate the Backup-up Visible command in the On duty calendar menu or click on the [icon] button.

**Defining a replacement period** – Select the active period to be replaced (see "Selecting a duty period" page 231).

**NOTE**

The replacement may also concern several active periods selected simultaneously provided they belong to the same user.

– Select the Create Replacement function ![icon] in the toolbar or in the On duty menu.
The dialog box user will open.
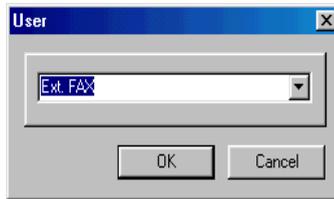


**Fig. 167**    User

– Select the replacement user and click on OK. The On duty period definition box will open.



**Fig. 168**    Defining a replacement period

– Simply choose the contact media to be used and their priority level, as when defining the active period (see page 230).

– Click on OK.

– The period(s) replaced will be displayed with their link to the new active period(s).

## Modifying a duty period

To modify an active duty period:

– double-click on the period concerned to call its definition box (or in Selection mode select Modify in the shortcut menu).

– type the changes to the duty periods or contact media

▬▬ | **NOTE**

When a period is modified, any related backup or replacement period will also be modified

▬▬ | **NOTE**

A group of periods can be modified simultaneously provided they belong to the same user. In this case, select the periods concerned and click Modify in the shortcut menu.

# Allocating on-duty to an organization

To shift an organization into standby, you must:

– Create a user: See "User identification" page 38.

– Authorize him for the organization, through Web access (see "User authorizations" page 40).

– Define the alarm category (RTU, Server, Client), type (OTDR, OSA) and level of severity for which the organization will be contacted, and check the Always on duty box: See "Defining on duty levels" page 43.

An on-duty period can be allocated to this user, whose name is mentioned in the calendar.

# Allocating on-duty to a link or a TAP

To shift a link or a TAP into standby, you must:

– Create a user: see "User identification" page 38.

–  Authorize it over one or several links or a TAP, through Web access (see "User authorizations" page 40).

–  Define the alarm category (RTU, Server, Client), type (OTDR, OSA) and level of severity for which on-duty is applied, and check the Permanent on-duty box: See "Defining on duty levels" page 43.

An on-duty period can be allocated to this user, whose name is mentioned in the calendar.

# How to acknowledge an alarm by telephone

In the alarm window are given:

–  The telephone number of the server which has been entered in the Server setup dialog box called-up by the Configuration/Server menu.
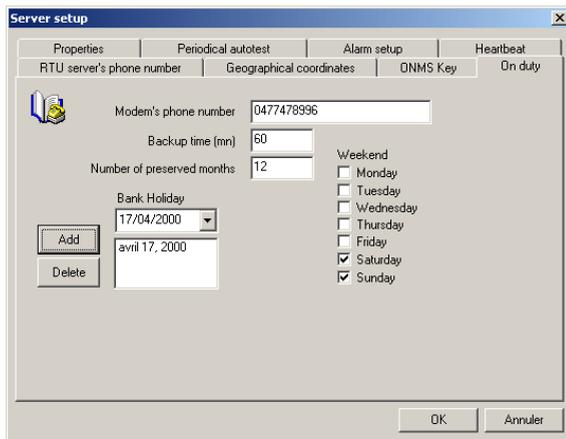


**Fig. 169**    Server modem telephone number to be contacted

–  An acknowledgement code.

To acknowledge the alarm by telephone, the user has to dial the telephone number, then enter the acknowledgment code when requested by a voice server.

A success or failure message is then given by the voice server. In the event of failure, another attempt may be made.

# Printing the calendar

It is possible to print the part of the calendar displayed on the screen, in either daily, weekly or monthly mode, if the server is connected to a printer.

To print the calendar, use the  button or the Print command in the On duty calendar menu.

# Web Client Option

# 12

The ONMS Web Client option is a specific application that can be installed on an ONMS server to enable, via Internet, a certain number of simultaneous connections.

– Check the condition of the optical cables monitored.

– Obtain detailed information on the optical alarms.

– Monitor the interventions and events detected on a link monitored and viewing the current status of this link.

  Acquire an OTDR trace or acquire an optical spectrum (WDM measurement) or launch a preset monitoring test.

All that is required to access the ONMS Web server, is:

– an **Internet Explorer** browser (version 6.0 or later) or **Firefox** version 3.5 or later.

– to be registered by the ONMS administrator and to have the necessary privileges.

The Web Client option is mainly intended for users who have a punctual need of the ONMS system, for maintenance technicians that are on the move and eventually for client operators wishing to check the state of fibers or for sub-contractors in charge of maintenance.

⚠️ – This application must only be installed and configured by JDSU's accredited technicians.

– The use of the Web Interface for the OTU-8000 is explained in the OTU-8000 user manual.

# Connecting to ONMS via the Web

### Conditions

To use the Web Client option, you only need to be a registered user by the ONMS system with a web access or to be a usal user of the system.

To request a test or acquisition on demand, you must get the necessary privileges from your ONMS administrator.

### To access the ONMS server from your Web browser

To access the ONMS server via your web browser, do as follows:

**1** Enter the Intranet address or the name of the ONMS server (defined by the system administrator).

When the web session is established, the following welcome window is displayed:



**Fig. 170** Enter the Alias and Password Fields

**2** Enter your **Alias** and **Password** fields to access the ONMS system.

**3** Select the required language from the **Language** field: English, French, German and Portuguese.

**4** Click the **SUBMIT** button to continue.

If a wrong alias or password has been entered, then a message is displayed. Contact your ONMS system administrator to get the right information.

The screen displayed indicates the total number of alarms active according to their severity (without taking filters into consideration). It also enables viewing the status of the links and TAPs accessible by the user, and running various tests.

---

**NOTE**

If you are connected to Internet, you can click at any moment on the JDSU logo to  access the JDSU web site.

The following window is displayed:



Total number of alarms active according to their severity
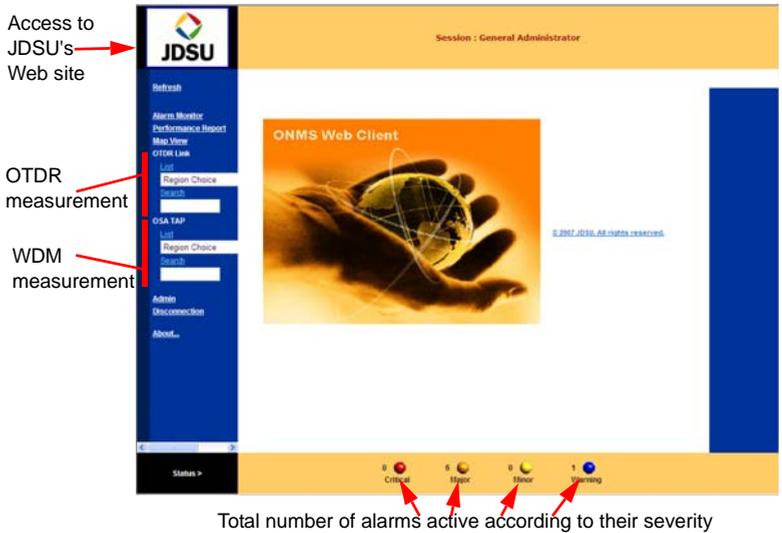
**Fig. 171**     Main screen of the Web Client station

All commands are accessible from the banner on the left-hand side:
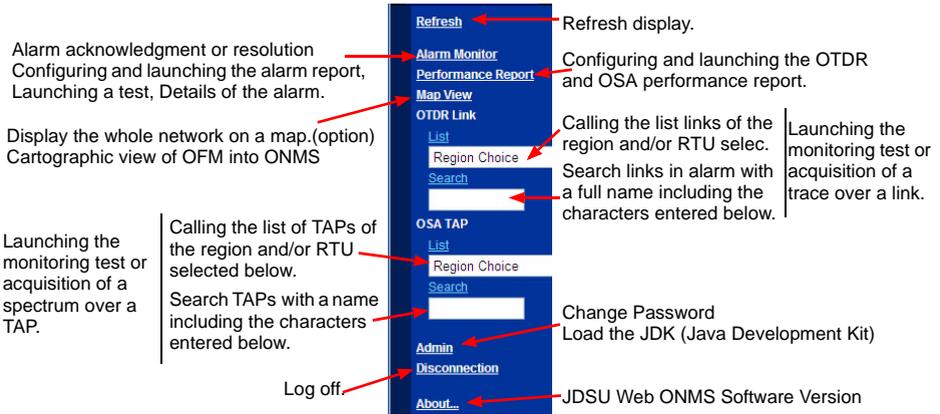
**Fig. 172** Main commands

# Alarm monitor

The **Alarm Monitor** enables displaying the list of links or TAPs in alarm you are authorized access to (see your administrator ONMS) and selecting one of them to:

– define and display its alarm report

– launch a monitoring test or acquisition of a trace

– display the alarm's detail.
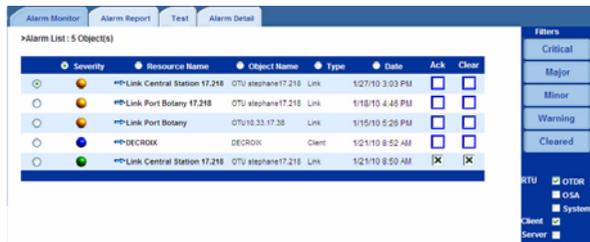


**Fig. 173** Alarm monitor

**NOTE**

In the list of links or TAPs in alarm, move the cursor to the icon indicating the alarm's severity to display its ID number

**Sorting alarms** Filters enable selecting the alarms to display.

– to eliminate the alarms below a given severity, click the max severity to display (**Critical, Major, Minor, Warning, Resolved**).
– to select the system alarms to display, check the appropriate boxes.
    – **RTU** (OTDR / OSA / System)
    – **Client** if you have access to a Client station
    – **Server** if you are an administrator.

**Acknowledging or resolving an alarm**
To declare an alarm Acknowledged or Resolved, if you are authorized to do so, check the appropriate box, under the **Alarm Monitor** tab.

## Alarm report

*Generating the alarm report*
To edit an alarm report, under the **Alarm monitor** tab, select the alarm, then under the **Alarm Report** tab:

– Select the elements to be integrated into the report (see "Selecting the sections in the OTDR alarm report" page 76).
– Click **SUBMIT** to print the report.



**Fig. 174** Configuring the alarm report over a link

The Alarm report is generated and directly open on the Web.
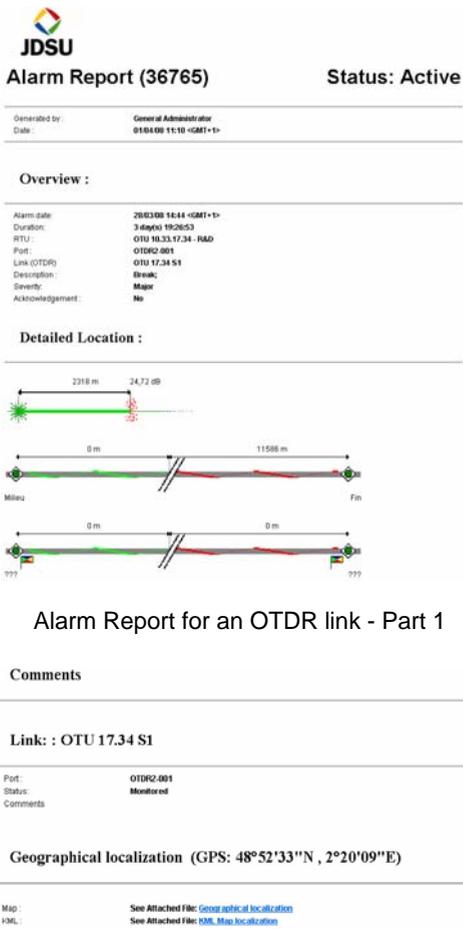
**Fig. 175** Alarm Report for an OTDR link - Part 1



**Fig. 176** Alarm Report for an OTDR link - Part 2

Tools

Radio button to designate the active trace

Parameters of traces

Distance unit kilometre, kilofeet, or mile

List of optical events identified on the OTDR trace
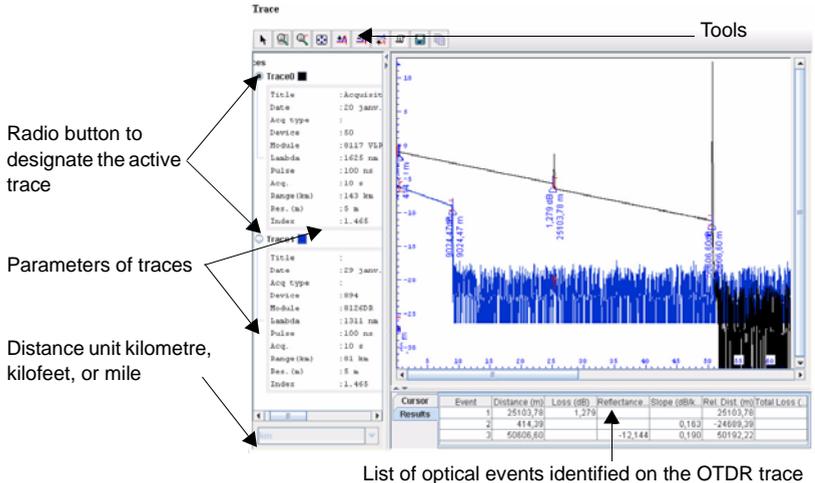
**Fig. 177** Alarm Report for an OTDR link - Part 3

The tool bar displayed above the trace allows different actions.

### Zoom on trace

The icons  allow respectively to zoom in, zoom out and fit to content on the trace.

### Cursors

Set cursor(s) on the trace using the icon . Up to two cursors can be set onto the trace.

The value of cursors is displayed under the trace, in the tab Cursor
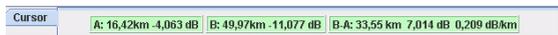


**Fig. 178** Example of cursors value

### Adding / Removing a trace

Several OTDR traces can be displayed in overlay. To do so:

**1** Click on the icon 

**2** Select the trace to be displayed in overlay with the current one

**3** Click on OK to validate.

Both traces are displayed in overlay.

To remove one of the trace:

**1** Select it on the left window

**2** Click on the icon ▣ . The trace is removed from the display.

To remove a trace, you can also right click on the trace in the left window
and select Remove trace  .
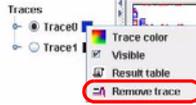


⚠ **Even if only one trace is opened, by clicking on the icon Remove
Trace ▣ will automatically remove the trace from the display.**

When several traces are displayed in overlay, you can hide one or
several of them. To do so, right click on the trace(s) which must be invis-
ible and click on Visible to deactivate it.



Trace still present
but invisible
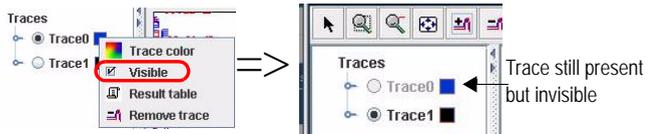
The trace is not displayed anymore but it is greyed on the left window.
Right click on the trace and reselect **Visible** to show it back.

### Traces in multi-window mode

To display traces in multi-window mode:

**1** Click on ▣ .

**2** Select the trace to be displayed in multi-window with the current
one(s).

**3** Click on **OK** to validate.

Traces are displayed in two windows.

### Events table

**1** Click on ▣ to display the list of the optical events detected on the
OTDR trace.

A results table for the active trace is displayed below the trace.

When several traces are displayed in overlay, select the trace for
which the table must be displayed by clicking on the corresponding
radio button.

To display the results table, you can also right click on the trace wished in the left window and select **Result table**
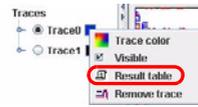
### Saving the OTDR trace

**1** Click on the icon 🖫 to save the trace on the computer.

**2** Select the directory in which it will be saved and click on OK.
The trace can be save to the «tfw» or «sor» format, in order to be read with Fiber Trace or Fiber Cable JDSU softwares.

⚠️ | **No actions on the current trace via the ONMS Web Interface (traces in overlay, position of cursors etc.) is saved.**

### Choosing the elements to be displayed on trace

**1** Click on the icon 🗐 , to open a new box and choose which elements are visible on the trace (results, cursors etc.)

**2** Select / deselect the elements to be displayed / hidden.

**3** Click on **APPLY** to validate

### Traces colors

The traces colors, wether one or several traces is/are open, can be modified.

**1** On the left window, right click on the trace for which color must be modified and click on **Trace Color**                .
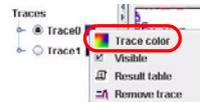
The Color dialog box opens
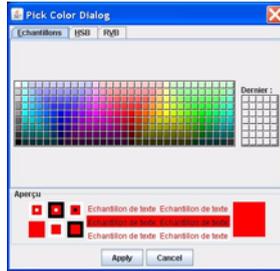
**Fig. 179** Colors dialog box

**2** Click on the color wished, having previously selected the right tab (Samples / HSB / RGB).

**3** Click on **APPLY** to exit the dialog box and go back to the trace display, with the new color applied.

Under the trace display, two other links may be available.

In the Geographical Localization part:

– the link **Geographical Localization**, on the line Map, allows to reach the display of the alarm localization, at the bottom of the report page (option) (see Figure 180).

– the link **KML Map Localization**, on the line KML, allows to open Google Earth™ and directly display the alarm on the Google Earth map.
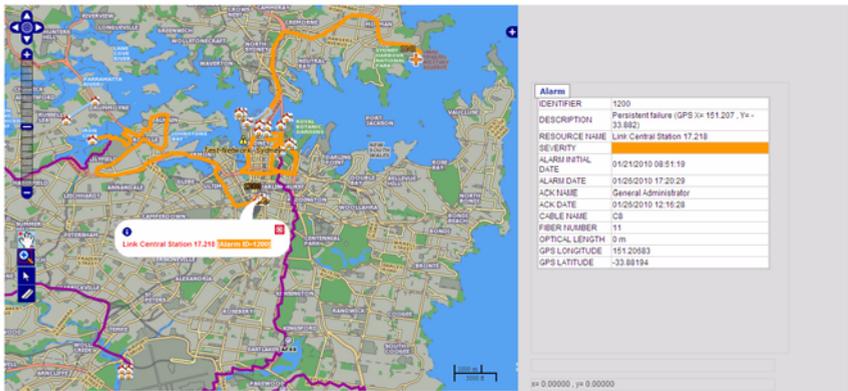


**Fig. 180** Alarm Report for an OTDR link - Part 4

**Launching a test**

To launch a test over a link or TAP in alarm, under the **Alarm monitor** tab, select the alarm, then under the **Test** tab: the link or port selected is reminded.
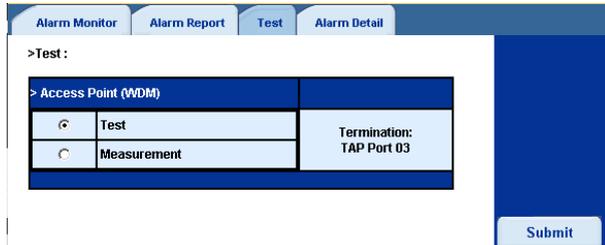


**Fig. 181**     Launching the test

– Select:
  – either **Test** to launch, over the link or TAP, the monitoring test defined (comparison of the trace acquired with a reference trace): see "Monitoring a link" page 141 or "Setting TAP under monitoring" page 169.
  – or **Measurement** to acquire an OTDR trace over the link or a spectrum over the TAP: see "Conducting an OTDR measurement on demand" page 123 or "Acquiring a spectrum" page 176.
– Click the **SUBMIT** button to launch the test or measurement.
– A window displays the characteristics of the measurement: click **START**. The various steps of the operation are displayed, as well as a measurement progress bar, then the result is displayed in the alarm report.

**Details of the alarm**



Event description get by moving the cursor on the event's 🛈 icon.

**Fig. 182** Displaying the detail for a system alarm

Move under the **Alarm Detail** tab, to display the main characteristics of the alarm:

– For system alarms, the list of events linked to the alarm is displayed under Description. Move the cursor onto an event's 🛈 icon, to display its description.

– For OTDR or OSA alarms, the alarm details are in the **Alarm report**.

# Performance report

The performance report can be generated for an organization in the case of OSA monitoring, for a region or a link in the case of OTDR monitoring.

**Performance report per organization**

– Click on **Performance report**.

– Under the **Organization** tab, select the organization concerned by the report, as well as the frequency band (nm or THz), then check the parameters to be included in the report.

**Fig. 183** Configuring the performance report per organization

– Click on the **SUBMIT** button.

The performance report requested displays.

**Performance report per region and/or link.**

– Click on **Performance report**.

– Under the **OTDR** tab, select the region(s) concerned by the report, as well as the link (or the **All** option for all regions and/or all links), and check the parameters to be included to the report.



**Fig. 184** Configuring the performance report

– Click on the **SUBMIT** button.

The performance report requested displays.

# Monitoring or acquiring a trace over a link or TAP

To launch a monitoring test or acquisition of a trace over a link or a TAP not in the alarm status, you must:

– display the list of links or TAPs,

– select the link or TAP

– select the monitoring test or acquisition and launch it.

**List of links or TAPs**

You can select either all the links (or TAPs) for a region, or the links (or TAPs) for a region monitored by an RTU, or search the links (or TAPs) with a name including a string of characters.

**To select all the links or TAPs in a region:**

– in the **OTDR Link** (or **OSA TAP**) field, select the the name of the region with an asterisk, e.g. «North: *»
– click on **List**: all links (or TAPs) for this region are displayed.

**To select the links or TAPs for a region monitored by an RTU:**

– in the **OTDR Link** (or **OSA TAP**) field, select the region followed with the name of the RTU, e.g. "North: RTU-North"
– clock on **List**: all links (or TAPs) monitored by the RTU are displayed.

**To search the links or TAPs with a name including a string of characters**

– Enter the string of characters in the field below,
– Click on **Search**.

The list then displays in the form of a table containing the radio button enabling to select the link (or TAP), the RTU ensuring monitoring, the RTU port the link is connected to, the date of declaration of the link (or TAP), the link's status (monitored or not).

Select the link (or TAP), and go under the tab Link Detail to display a descriptionb of the link and go to the tab Test tab to launch the monitoring test or trace acquisition.

**Displaying the Link / TAP Details**

Once the link / TAP to be displayed is selected in the list:

**1** Click on the tab **Link Detail**

A description of the Link is displayed: Link Id, RTU concerned with this link, Port used, ... indication if the link is monitored or not, comments if exist, link toward the KML file.

**Fig. 185** Link Detail

**2** If a cable route has been drawn via Google Earth™, and the KML file has been associated to the link in the ONMS, then a link toward this KML file is available in the tab **Link Detail** > **See link in KML**.

**Acquiring a trace** To launch the acquisition of an OTDR (or OSA) trace

**1** As specified above, display the list of links (or TAPs):



**Fig. 186** Display the links for a region

**2** Click on the radio button of the link (or TAP)

**NOTE**

In this list, you can choose to make the link available / unavailable by selecting / deselecting the parameter **Availability**.



**Fig. 187** Selecting the test

**3** Under the **Test** tab, select **Measurement**, then click on **SUBMIT**.

The acquisition configuration box displays.
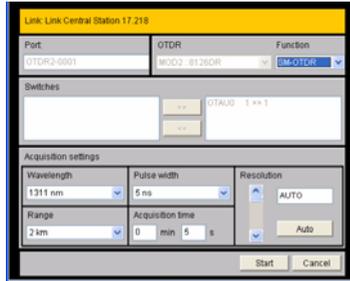
**Fig. 188**     Configuration of the OTDR measurement

– Select the acquisition conditions, then click on the **START** button. Connection is performed.

A progress bar indicates the progress status of the acquisition. Measurement can be interrupted

For the first acquisition, a warning dialog box opens after the acquisition, asking if the application for the trace visualization must be launched.

Select ***Always trust content from this publisher*** in order to not display this message each time a trace must be displayed, and click on Ok.



**Fig. 189**     Warning Dialog Box

– Upon completion of acquisition, the OTDR trace for the link (or TAP spectrum) displays. Buttons are available at the top of the screen to:

– set a cursor on the trace

– zoom in on the trace, and cancel it

– export the trace

– add another trace for a multiple trace analysis

– remove the display of parameters on the left-hand side.

**Monitoring test** To launch a monitoring test defined by the administrator, over a link (or TAP):

1   Click on the radio button of the link (or TAP).

2   Under the **Test** tab, select **Test**, then click on the **SUBMIT** button.

3   A window then shows the list of tests defined for the link. On a TAP, only one test is available.
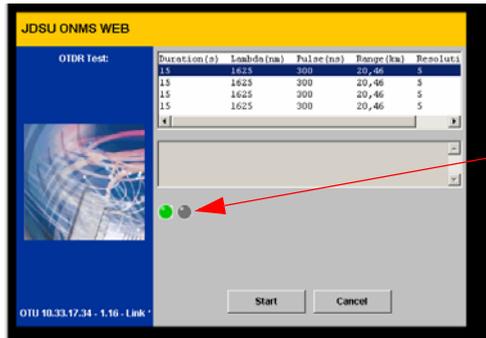


Red: indicates that the measurement is subject to the time zone's availability (when links are monitored at both ends) - see page 129.

**Fig. 190**   Selecting the OTDR monitoring test

4   Select the test, and click on the Start button. Messages indicate the test is started and successful logging on. Then, a progress bar indicates the test's progress status.
Upon completion of the test, the result is displayed in the form of an alarm report. See "Generating an alarm report" page 204.

**Test result analysis**

**OTDR or OSA trace**

The trace obtained in the alarm report, following a monitoring test, is superimposed on the reference trace defined in the test.

–   the reference trace displays in black.

–   the trace acquired during the test displays in red.

Check button to designate the active trace.

Parameters of trace

Distance unit kilometre, kilofeet, or mile
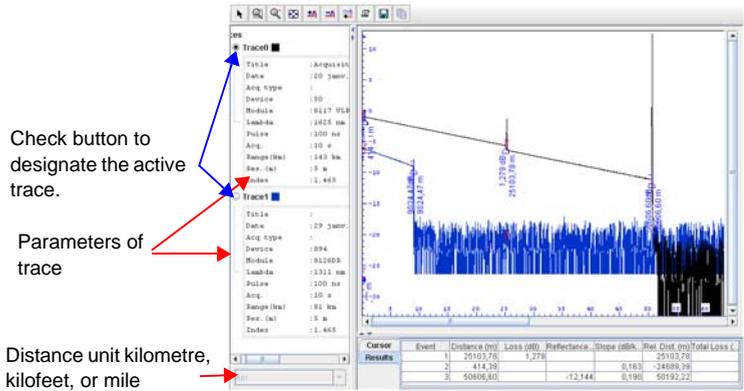
**Fig. 191** Example of test result (with OTDR traces)

Buttons are available at the top of the trace to zoom in / out on the trace, and cancel it, to export the trace in tfw or sor format and other actions (for more explanations on the actions available, see page 245).

### WDM results table

Click on the ▦ button in the spectrum display screen to call the table listing the results related with the active spectrum.
The banner above the table is black if the results are related with the reference spectrum, red if related with the acquired spectrum.

For each channel detected, the table specifies:

– its wavelength
– the last value measured for Signal / Noise ratio
– the average, max, and min Signal / Noise ratio during acquisition
– the typical deviation of the Signal / Noise ratio

# Map View

The **Map View** function is a cartographic view of the OFM-500 into the ONMS, in the Web application (option).

**The Map View is an option of the ONMS Web Application.**

Via this function, the cutomer can:

– Generate an alarm report
– Acknowledge and / or clear an optical alarm
– Make a link « unavailable »
– Launch a test or an OTDR measurement on links on alarm

**Main Window** On the ONMS Web application, click on the **Map View** on the left of the screen.



Navigation bar

Tools

Current GPS coordinates of the cursor

Generate an alarm report

Center the display on alarm

Refresh the display

Scale

List of active optical alarms
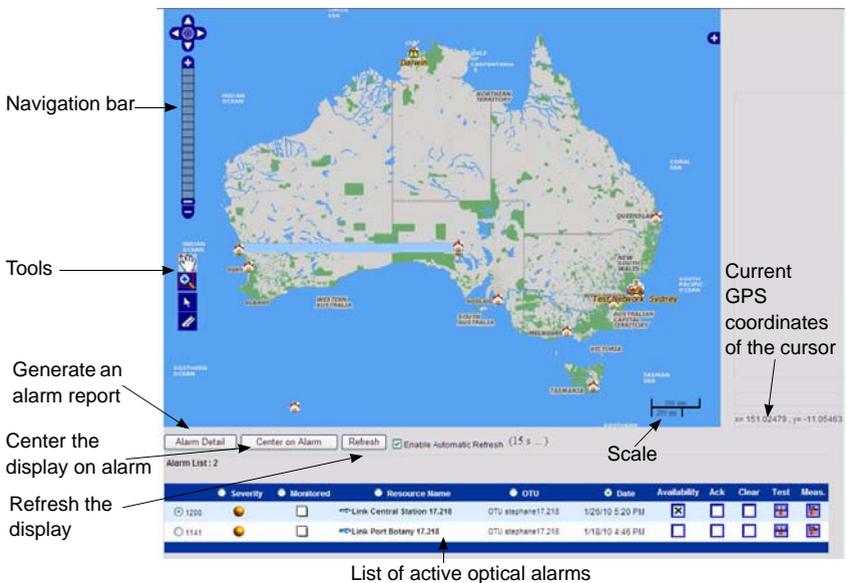
**Fig. 192** Map View

The screen displays:

– the navigation bar and tools available
– the general view of the map, with display of the different layers (display depends of layers configuration)
– functions related to active optical alarms
– a list of active optical alarms detected.

The display of map and alarm list can be refreshed by clicking on the button **REFRESH**, or is refreshed automatically if the parameter **Enable Automatic refresh** is selected.

## Navigation bar and tool bar

The navigation bar and tools bar on the left side of the Map view screen allows to:

– navigate onto the map: move the map, zoom on map...
– measure the distances
– display the network's objects attributes (see "Layers display" page 259).

Once the map is displayed in its initial position, you can use the tools on left of the screen to modify the display:



Use the arrows to move the display left, right, up or down.

Click on [icon] to retrieve the initial position of the map

Use the cursor [icon] up and down or click on «+» and «-» to rise or reduce the zoom level on the map.

- Move the map by dragging the mouse.
- Zoom in / out on map by using the mouse wheel.

Drag on the map to define the zoom area.

Click on network's objects to display their attributes (see "Layers display" page 259)

Measure a distance on the map (see "Measuring a distance onto the map" on page 259).

**Fig. 193** Navigation and tools bars

**NOTE**

Once selected, a tool is kept until another tool is selected.

***Measuring a distance onto the map***   In map view, you can measure a distance on the map.

**1**   Select the tool ✎

**2**   Click on one starting point on the map

**3**   Double click on the ending point.

   The distance between the two points is displayed on the right side of the map, and is represented by a black dotted line onto the map.
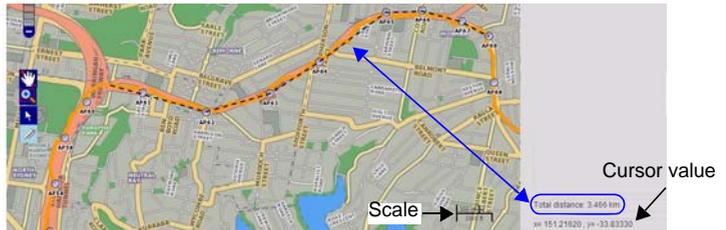


**Fig. 194**       Measuring a distance on the map

Note: you can click on several «halfway» points; this can be very useful, especially if you wish to measure a  long distance.

## Layers display

The display of map can be modified according to the layers you wish to activate.

**1**   Once the map view is open, click on ➕ on the upper right part of the map.

   The list of layers available displays.



**Fig. 195**       Layers available

**2**   Select/ deselect the overlays you wish to display / hide onto the map.

   The **Base Layer «Maps»** is always selected, and cannot be deselected.

**3**   Click on ▣ to close the layers configuration menu.

Once one or several overlays are selected, the network's objects are represented graphically on the map.



Cosmetic

Access point

Path with optical alarm (major)
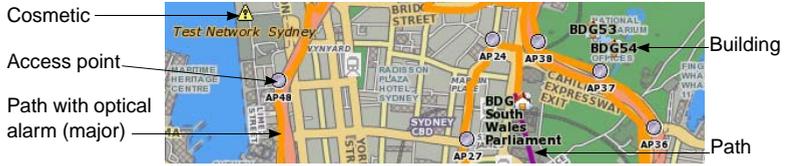
Building

Path

**Fig. 196** View of network's objetcs icons

### Buildings

– The icon is ,  or  according to the zoom level.

### Path

– The section of the path on which an optical alarm has been detected is colored on the map: orange for Major alarm and yellow for Minor.

**NOTE**

When an alarm is cleared, the path goes back to its standard color

– The colours of paths without alarms depend of colours applied on OFM 500

### Network objects attributes

– Use the tool  to click on one icon and display the correponding information on the right of the screen.



Active tab
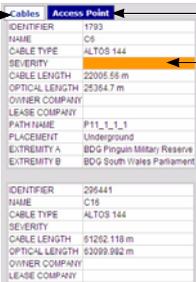
Access point(s) with this cable

A major optical alarm is active on this cable

**Fig. 197** Example of network's object information

**Alarms**  On the lower part of the Map View window, a list of active optical alarms is displayed.

Several actions are available via Map View, regarding the alarms.

*List of active optical alarms*  This list is automatically displayed when one or several optical alarms have been detected on the network.

It is updated when:

– an optical alarm is cleared
– a new optical alarm is detected



**Fig. 198**  List of optical alarms

The list of current optical alarms is ordered by last update dates by default.

– Click on one radio button of the column header to modify the order criteria.
– Make the link unavailable if human intervention is required on the monitored fiber, by deselecting **Availability** parameter (confirmation is required).

> **NOTE**
>
> By default the parameter **Availability** is selected

– Select **Ack.** parameter to acknowledge the alarm (a confirmation is required).
– Select **Clear** to clear an optical alarm (a confirmation is required). However, if the fault is still there, a new alarm appears.

> **NOTE**
>
> The alarm is automatically cleared when the OTU-8000 does not detect fault.

*Launching a test or an OTDR measurement on the link*
In the list of optical alarms, two icons are available, allowing to launch a test or an OTDR measurement on the link on which the alarm has been detected.

### Launching a test

– Click on the icon  to launch a test of the link

The OTDR test window opens

The link is tested

An alarm report generated.

━━━━━ **NOTE**

Select an Alarm ID in the list of Alarms and press the **ALARM DETAIL** button to generate a full alarm report at any time.

If the alarm is confirmed, a new persistent Failure Event is created (the date field updated).

See "Monitoring test" page 255

### Launching an OTDR measurement

– Click on the icon  to launch an OTDR measurement on the link with an optical alarm.

The user can select his own settings and launch an OTDR acquisition (see "Configuration of the OTDR measurement" page 254).

The trace is displayed after the OTDR measurement is completed.

*Optical alarms on map*
As soon as an optical alarm is detected on the network, the map is also updated, and the optical alarm is graphically represented by the icon .

Once an alarm is acknowledged (see 261), the icon turns to .

Select an alarm from the list below the map, and click on the button **CENTER ON ALARM** to center the alarm position onto the map, keeping the zoom level defined.

Select the tool  and click on the alarm icon on the map to display details related to this alarm on the right of the map.

Alarm General information

**Fig. 199**     Alarm on the map

Click on **ALARM DETAIL** button to launch an alarm report, which is the same as shown from Figure 175 page 244 to Figure 180 page 248.

# Administration

The tab **Administration** allows to:

– modify your password
– download the JAVA applet, for the display of the trace visualization applet, without using Internet.

**Modifying the password**

To modify the password:

**1** In the **Administration** tab, enter your actual password

**2** In the second field, enter your new password

**3** Confirm the new password by entering it a second time in the third field.

**4** Click on **APPLY** to validate the modification

**Fig. 200**      Password modification

**Downloading JAVA**      The tab **Configuration** allows you to download the JDK, in order to display the traces visualization software, without having to use Internet.

**1**   Click on the **Configuration** tab

**2**   Click on the link Download Java 2 Runtime

**3**   Follow the instructions for the installation



Version of the ONMS Web Access

Link for the JDK installation

link to JDSU internet site
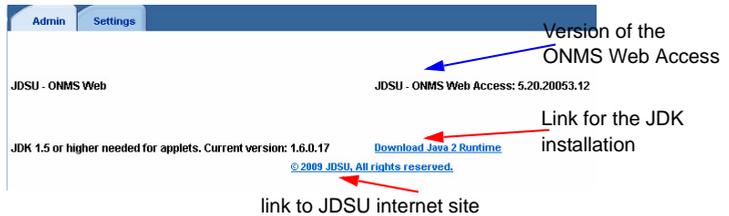
**Fig. 201**      Configuration page

# Disconnecting the Web Client option

To close the Intranet Client web session, single-click the **Disconnection** option that is located on the left-hand side of the window.

Clicking the **Close** window button (on the top right-hand corner) does not close the web session even though the window has been cleared. The web session is still open and remains that way until the time-out period closes the session. For the ONMS system, there is a limited number of Web Client sessions allowed simultaneously. Hence, by not correctly closing the web session, this could stop another user from opening a web session.

# Index

# W

WDM
    analyzis 213
    identification 57

Web
    alarm 242
    java 264
    launch test 249
    password 263
    performance Report 250

**JDSU**