SPATIAL*net*™

Release 5.0

SPATIALnet FM Tutorial

Release 3.0.2	April 19, 2004
Release 4.0.1(d)	August 22, 2004
Release 4.5	January 30, 2005
Release 4.6	November 29, 2005
Release 5.0	August 16, 2007

Contents

Welcome

About this manual

This manual is intended for use both as an introduction to SPATIAL*net* FM for beginners, and as a reference document for more advanced users. It contains detailed descriptions of the commands available in this configuration of SPATIAL*net*.

Who should use this manual?

This manual is intended for all users of SPATIAL*net* FM. Administrators and application developers should consult further documentation.

Prerequisite knowledge

A sound working knowledge of the Microsoft Windows© user interface is required (menus, toolbars, dialog boxes, mouse operations, etc.), as well as a good understanding of the AutoCAD application environment. Note that a high level of expertise in AutoCAD is not required to use SPATIAL*net* FM.

Related documentation

There are a number of other manuals that may be useful to you:

Document	**Description**
SPATIAL*net FM Administrator's Guide*	Describes the functionality available in JMS Administration.

Table 1 - Related documentation

The AutoCAD online help documentation is a useful guide to the AutoCAD environment and its commands, and is accessible via the AutoCAD **Help** menu.

Conventions used in this manual

Below is a list of the typographical conventions used throughout the guide.

Convention	**Meaning**
Bold, Arial font	Menu name, keyboard key, dialog box name, field name, command name or icon.
>	Used to separate menu commands, for example: **SPATIALnet > View > Keymap**

Convention	Meaning
Note:	Notes indicate information that emphasize or supplement information from the main text, as shown below. **Note:** This information may help in some situations, or contain hints and tips.
Important:	The information contained in the note is critical to completing a task.
... Details	The description of a generic dialog box in this manual will always include three full stops in the name of the dialog box. These full stops are used as a stand-in for an entity type. For example, the **...Details** dialog box can be interpreted as meaning the **Rack Details** dialog box, **Building Details** dialog box or the **...Details** dialog box of any other type of entity for which such a dialog box is capable of being displayed.
Italics	Indicates a reference to another section of the document, or to external documentation. Italics are also used when a new term is defined.
Hyperlink	Indicates a reference to a chapter or section within this document. There is an automatic cross-reference embedded to allow you to jump to a specifically referenced section when viewing this document as a PDF file or HTML page.

Table 2 - Conventions

Comments and suggestions

We welcome all comments on the software and documentation, and are very interested in suggestions, which would help us to enhance the SPATIAL*net* product and its usefulness to you. Please send your comments and suggestions to your distributor of SPATIAL*info* products.

How to use this manual

This manual both provides an introduction to new users of SPATIAL*net*, and also serves as a comprehensive reference for more experienced users needing information on how to perform any function in the system. The table of contents and index can be used to quickly find the area you are interested in.

If you are a new user, you should read Chapter 1 *Introduction: SPATIALnet Basics* and work through Chapter 2 *General SPATIALnet Functions*. This will introduce you to many aspects of the SPATIAL*net* FM system. Depending on the type of work you need to do, and the level of access your administrator has granted you, only some of the remaining chapters may be relevant to you. The table below provides a guide to the

type of user most likely to need the information contained in each chapter. If you are unsure which apply to you, speak to your SPATIAL*net* administrator.

The SPATIAL*net* FM Tutorial

This manual provides a brief but thorough introduction to SPATIAL*net* which walks you through several basic processes in the system. The tutorial is intended to help you familiarize yourself with the concepts and operation of the software. If you have not used SPATIAL*net* FM before, working though this tutorial is a good place to start.

The tutorial is divided into several chapters, each dealing with a specific aspect of the system, or set of functions relating to a particular kind of network.

The chapters are organized as follows:

Chapter 1. Introduction

- SPATIAL*net* Basics

Chapter 2. General SPATIAL*net* Functions

- Logging On and Selecting a Job
- CAD-Scrape Landbase
- Set Cad Layers, View Template & Symbol Search Path
- Control Object Visibility
- Setting Zoom Level Extents
- Save View
- Configure a New Boundary Type
- Add a New Boundary
- Editing Boundaries
- Working with Manipulators and Linear Edit Modes.
- Configure Network Support Structures
- Add New Poles and Strand
- Add Underground Structures
- Managing Conduits and Ducts
- Add New Site Type
- Add New Site
- Document References
- Generating Plottable Drawings
- Generating a Map Book

Chapter 3. Addresses and Service Areas

- Add Service Area Boundary
- Adding Buildings and Addresses (Manual Addition)
- Add Land Parcels
- Automatically Create Service Addresses from Land Parcels
- MDUs
- Edit Multiple Addresses
- Reassign Service Areas

Chapter 4. Outside Plant Fiber

- Add Term Panel to Site
- Adding More Equipment to a Site – Understanding Splice Locations.
- Moving Equipment Between Splice Locations.
- Configure New HFC Fiber Receiver Type and Usage Codes
- Add New HFC Node
- Add New Fiber Cable Type
- Connect Site and Node with a New Fiber Cable
- Insert “Tap Splice” into Existing Cable
- Editing Segments
- Cutting a Cable
- Reconnect Cable End by Dragging to New Site
- Add Fiber Annotation
- Splice Fibers
- Optical Coupling
- Trace Fibers
- Generate Splice Report
- Generate Link Loss Report
- Generate Trace Report

Chapter 5. Fiber To The Home

- Configure FTTX Distribution Cabinet Site
- Add New FTTX Distribution Cabinet Site
- Cloning a FTTX Distribution Cabinet Site
- Add Fiber Taps
- Associate Taps with Poles
- Create Drops
- Configure FTTX Fiber Cable Types

- Add FTTX Cable
- Assign Fibers to Tap Ports (“Tail Off” cable)

Chapter 6. Outside Plant RF (Coaxial Plant)

- RF Dictionaries
- Create Design Profile
- Associate Design Profile with Job
- Add RF Module to Existing HFC Node
- Add New HFC Node
- Connect RF Outputs to Optical Units in HFC Nodes
- Add RF Power Supply and Power Inserter
- Add RF Taps
- Associate Taps with Addresses
- Add RF Active Devices
- Add RF Active Device Components (Pads, EQs)
- Add RF Passive Devices
- Add RF Cable
- Create RF Leg With Inline Equipment
- Reconnecting RF Equipment
- Editing Taps
- Tracing RF Plant Connectivity
- MDU Detail Drawings

Chapter 6. Inside Plant

(Section 1) Physical Network

- Creating a site for recording inside plant.
- Adding floors, floorplans and equipment (racks, chassis, cards).
- Displaying rack elevation and connectivity diagrams.
- Connecting equipment.
- Connecting to Outside Plant.
- Tracing physical connectivity between inside and outside plant.

(Section 2) Logical Circuits

- Creating a logical protocol.
- Creating a customer and a logical service.
- Mapping the service to inside plant equipment.
- Configuring Routing and Multiplexing for the service.

- Tracing the logical service.

Notes:

1. **Chapters 1 to 3 should be worked through in order before subsequent chapters, as the later chapters assume familiarity with the material presented in the first three chapters.**
2. **Chapters 4 to 6 are independent. You may choose to do any of these, or all of them, and may do them in any order. Sections 1 and 2 of Chapter 6 must be done in the order presented.**
3. **You do not need to complete the entire tutorial in a single session. You can quit at any stage, log back in again at a later time and continue where you left off.**

Chapter 1

Introduction: SPATIAL*net* Basics

This chapter briefly describes the SPATIAL*net* system, and provides information about basic SPATIAL*net* concepts.

What is SPATIAL*net*?

SPATIAL*net* is a powerful application suite integrating CAD, network engineering, and GIS/reporting functions on a single platform. SPATIAL*net* includes functions for:

- planning
- designing
- construction
- operation
- maintenance
- management

of networks and their associated infrastructure and data sets.

SPATIAL*net* uses:

- AutoCAD to provide CAD-based representations of data, for both graphical visualization and editing.
- A relational database (either Oracle or Microsoft SQL Server) to store all data managed by the application (note that graphics displayed in AutoCAD are obtained from the database, and *not* stored in DWG files)

The use of a relational database allows all the network information to be stored and presented as a completely seamless map. That is, the map has none of the breaks between the edges of drawings or "tiles" which are common in traditional file-based systems. You can zoom out to any level, or *drill down* to any level, and can:

- View and edit selected entities or areas in the CAD window,
- View and edit the detailed attributes of any entity in a browser or dialog box, or
- Both.

What is SPATIAL*net* FM?

SPATIAL*net* FM is an application which has been developed on the SPATIAL*net* platform to manage and model fiber-optic, coaxial, and twisted-pair copper networks, including:

- Management of cable and equipment specifications
- Location and details of network facilities
- Location, configuration and connectivity of outside plant cables and equipment
- Physical splicing of individual fibers or pairs, and groups of fibers or pairs

- Usage and status management of end-to-end network paths
- Management of logical circuits and their relation to physical plant
- Comprehensive management of Inside Plant, including fully integrated connectivity, both inside a building, and also to outside plant.

This manual provides a step-by-step introduction to the user functions of the SPATIAL*net* FM application

The interaction between AutoCAD and SPATIAL*net*

SPATIAL*net* uses AutoCAD both to display viewable database objects on the screen, and to provide an interface for editing their graphical characteristics. There are two ways SPATIAL*net* allows you to interact with the objects it displays in AutoCAD, depending on the type of object. The types of object are:

- SPATIAL*net* entities
- CAD-Scraped entities

SPATIAL*net* entities

SPATIAL*net* entities represent objects which are managed by the SPATIAL*net* data model. Specifically:

- They have properties (or attributes) defined by SPATIAL*net*. The input and validation of these attributes is managed by SPATIAL*net*'s application functionality (as described throughout this manual), and their values are stored in the database tables and columns defined by the SPATIAL*net* object model.
- They satisfy specific rules and constraints (such as valid network connectivity or configuration).
- Their CAD representation is managed by SPATIAL*net*. Generally, you do not use AutoCAD commands to create or update SPATIAL*net* entities, but run commands from the SPATIAL*net* menus or toolbars.

SPATIAL*net* entities are defined by their data and relationships. This is in contrast to CAD entities, which are defined by their geometry and appearance. Consequently, many of the operations which are performed on CAD entities are not always appropriate for SPATIAL*net* entities, or do not provide the processing or rule-checking required to ensure that the SPATIAL*net* entity remains valid.

To make it easy to differentiate SPATIAL*net* entities from ordinary CAD objects, SPATIAL*net* displays a special kind of "grip" called a "manipulator" when an entity is selected in the AutoCAD window. SPATIAL*net* manipulators allow you to move, rotate, and perform other operations on objects in a way that ensures that the integrity of the underlying data model is always maintained. SPATIAL*net* manipulators look different from AutoCAD grips and there are several of them, each with a specific function as shown below:

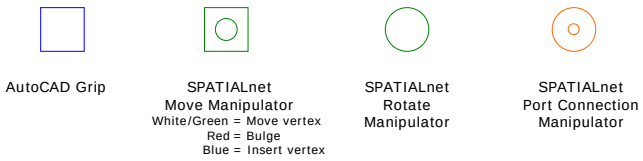


Figure 1 AutoCAD Grip and SPATIALnet Manipulators

When you select an object in AutoCAD and one or more manipulators appear, rather than AutoCAD grips, you can be sure you have selected a SPATIAL*net* entity.

There are other differences between the way SPATIAL*net* and CAD-Scraped entities behave. These are described in more detail in Chapter 10.

CAD-Scraped entities

While SPATIAL*net* entities (see above) are used to model complex network data, you can still create many kinds of AutoCAD objects using simple AutoCAD commands, and SPATIAL*net* will save these in the database. Whenever an area is queried from the database and rendered into AutoCAD, CAD-scraped entities will be included as if a DWG file containing that area had been opened.

To interact with CAD-scraped entities, use normal AutoCAD commands. These will work as they normally do when using AutoCAD without SPATIAL*net*. See Chapter 10 for further details of CAD-Scraped entities and the CAD-Scrape process.

Layout of the screen

If you have used AutoCAD software in the past, you will already be familiar with the layout of the SPATIAL*net* FM user interface.

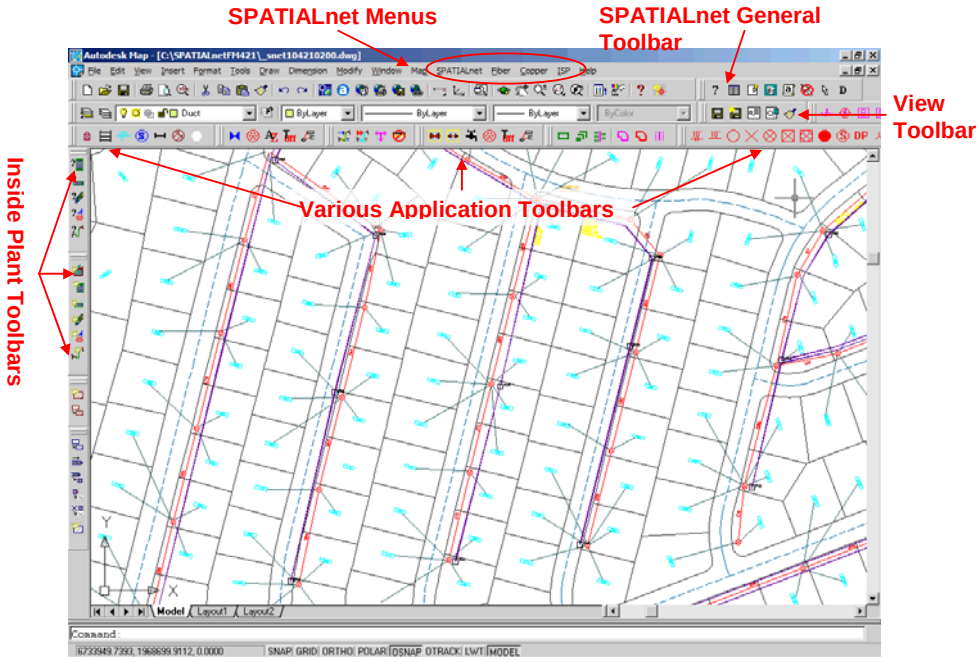


Figure 2 - A typical SPATIALnet session. Note that the visibility of toolbars and their locations can be set by the user. The arrangement may be different on different workstations.

General SPATIAL*net* commands are found under the **SPATIAL*net*** menu. Commands relating to Outside Plant Fiber functions are under the **Fiber** menu. Commands relating to Outside Plant Copper functions and under the **Copper** menu, commands relating to Inside Plant are found under the **ISP** menu, and commands relating to RF are found under the RF menu.

The network details in the display window are created directly from the spatial database. There are no DWG files required to produce or maintain map views.

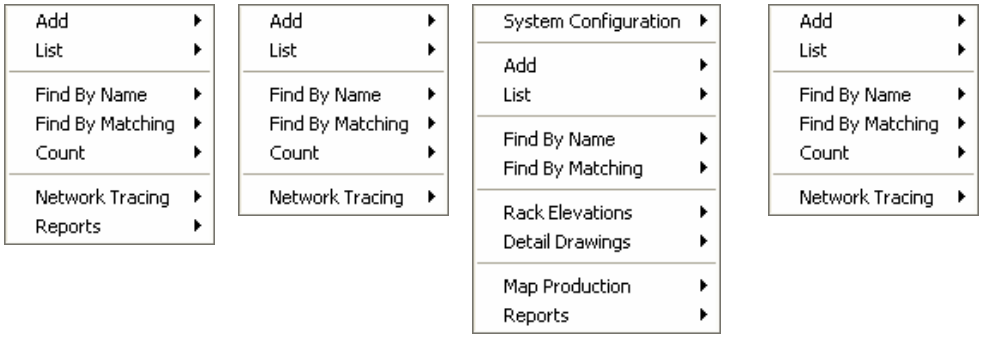
You use a combination of commands available from the menus, SPATIAL*net* toolbars and drag and drop functionality to interact with SPATIAL*net*.

Menus

*Figure 3 - The SPATIAL*net* menu*

The **SPATIAL*net*** menu is where all the commands relating to the usage of SPATIAL*net* are located.

The Fiber, Copper, ISP, and RF menus are also shown below.



Fiber Menu *Copper Menu* *ISP Menu* *RF Menu*

Figure 4

Note: The options that you see on the SPATIAL*net* menu or any of its submenus may differ from those above as menu layouts can be customized for each installation.

General toolbar

As shown in Figure 2, the SPATIAL*net* user interface contains a toolbar that provides shortcuts to certain key SPATIAL*net* commands. These enable you to:

- Set the selection mode (single, multiple) or clear the current selection.
- Examine the details of the currently selected SPATIAL*net* entity.
- Bring up the data browser for the currently selected SPATIAL*net* entity.
- Edit the properties of the currently selected SPATIAL*net* entity.
- Edit the annotation of the currently selected SPATIAL*net* entity.
- Delete the currently selected SPATIAL*net* entity.
- Add annotation for the currently selected SPATIAL*net* entity.
- Renumber the currently selected SPATIAL*net* entity.
- Clear the selection set.
- Toggle between single and multiple selection mode.
- Highlight the currently selected SPATIAL*net* entity.
- Clear the highlight entity(ies).
- Use the SPATIAL*net* Data Dictionary tool (administrator mode only).

The General toolbar is described in detail, below.

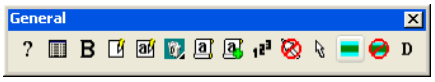
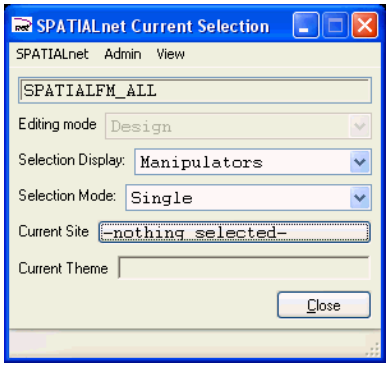


*Figure 5 -SPATIAL*net *General toolbar*

Note: The options that you see on the SPATIAL*net* toolbar may differ from those above. This shows the toolbar for a user with Administrator privileges.

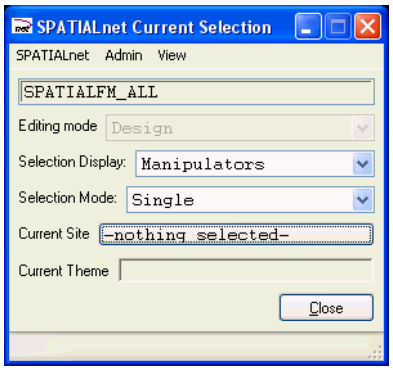
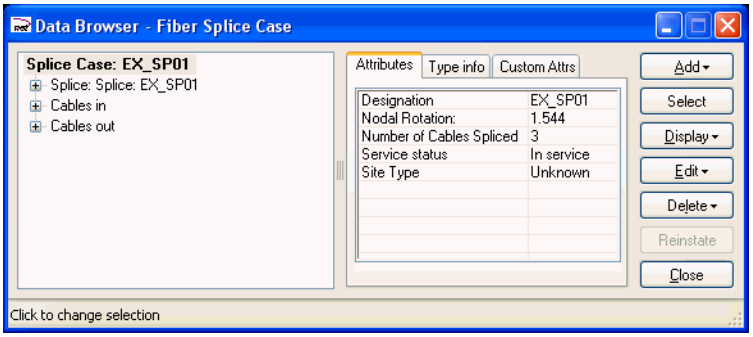
Button	Description
?	Activates the **Current Selection** dialog box that gives details about which entity is currently selected and what the current view type is. A dialog box similar to the following is displayed.



The fields and controls on this panel are:

Top Field (unnamed)	The current SPATIAL*net* view type
Editing Mode	Unused in this version
Selection Display	**Manipulators:** Use the SPATIALnet manipulators to edit SPATIALnet entities. **Native Grips:** Use AutoCAD's normal grips to edit SPATIALnet entities.
Selection Mode	**Single:** Only a single SPATIAL*net* entity can be selected at a time. **Multiple:** Multiple SPATIAL*net* entities can be selected at a time.
Current Site	The current ISP Site. This will is used to locate certain ISP functions within a specified site. To change or clear the selection, click on this field. A pop-up menu will be displayed with options **Select...** or **Null**.
Current Theme	Unused in this version
Bottom Field (unlabelled)	The currently selected SPATIAL*net* entity. (If nothing is selected this field does not appear.)

Activates the **...Details** dialog box of the currently selected SPATIAL*net* entity.

	A dialog box similar to the following is displayed: 
B	Opens the object browser window displaying a dialog box similar to the following: 
	Opens the Edit dialog box for the selected SPATIAL*net* entity, allowing its properties to be modified.
	Edits selected annotation.
	Deletes the selected SPATIAL*net* entity.
	Annotates the selected entity.
	Partially Annotate Selected Entity button. This creates annotation linking an object to a range of pairs/fibers in a cable.
	Auto-renumber street addresses
	Clears the current SPATIAL*net* selection (does not affect the current AutoCAD selection. See Currently selected entity, below).
	Changes the current selection mode between Single and Multiple selection. Also indicates current selection mode (double arrow is displayed in multi-selection mode).
	Highlights selected entity.
	Clears the current highlight.

D	Activates the SPATIAL*net* Data Dictionary tool, which is only available to users with administrator privileges. This is used for debugging purposes only, and the SPATIAL*net* administrator may be asked to run the Data Dictionary tool by hotline staff when reporting a problem to extract more information about a particular problem.

Table 3 – General toolbar icons

Currently selected entity

The currently selected entity (also sometimes referred to as the *current entity*, the *current selection* or the *selected entity*) is the SPATIAL*net* entity you have selected using one of the methods described below. The selected entity then becomes the subject of any SPATIAL*net* commands that you may run (Display details, Edit, Delete, Splice etc.) or may supply the container for adding a new piece of equipment.

If the selection mode set in the **Current Selection** dialog is set to **Single**, (see page 16) there can only be one currently selected SPATIAL*net* entity at any one time. If it is set to **Multiple**, then many objects can be selected, but not all commands accept multiple selections.

Several methods are available for selecting entities:

1. Clicking on a SPATIAL*net* entity in the AutoCAD window will select it.
2. Clicking a **Select** button on any dialog box will select the entity highlighted in that dialog box.
3. When creating a new entity, there is usually an option to "Auto-select" it as soon as it has been created.

Note: The current SPATIAL*net* selection is different from the AutoCAD "pickfirst" selection (usually indicated by AutoCAD grips being displayed, and the CAD object being highlighted with a dashed linestyle). Setting or clearing one does not affect the other. Generally, CAD-Scrape objects are selected and de-selected using the normal AutoCAD commands and keystrokes for doing this. Any selection technique when applied to a SPATIAL*net* entity will be used to set the SPATIAL*net* selection. The user does not have to distinguish between SPATIAL*net* and CAD-Scrape entities when making a selection as the system will ensure the correct action is taken.

ToolTips

To display a brief description of an entity, position the mouse pointer over a solid portion of the symbol. Rest it there briefly and a ToolTip will appear displaying the brief description for this entity.

The same method can also be used on manipulators to determine their function.

Command line

The command line at the bottom of the screen is used to communicate information about what is happening and what should be done next.

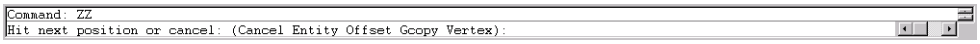
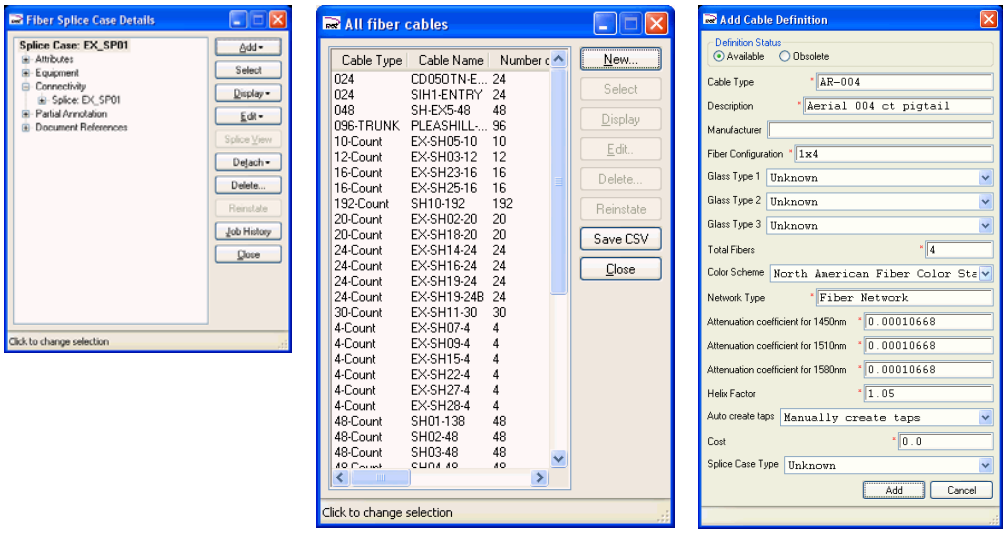


Figure 6 - Command line

If you are in the middle of an operation, and are unsure what to do next, check the command line to see if AutoCAD is prompting you to enter something.

Dialog boxes

Dialog boxes are used by SPATIAL*net* to communicate text information, or prompt for a response or textual input. There are broadly three classes of dialog boxes in SPATIAL*net*, examples of which are shown below:



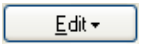
Details Browser *List Panel* *Edit Panel*

Figure 7 Basic Dialog Box Types in SPATIALnet

Both Details Browsers and List Panels provide a window for navigating information presented either as a tree (browser) or a tabular list (list panel), and a set of buttons each of which performs a specific operation on the entity highlighted in the table or tree.

The most common buttons and their uses are:

Select	Makes the highlighted entity the SPATIAL*net* Selected entity (see Currently selected entity, page 19).

<table>
<tr><td>  </td><td>Opens a pop-up with some or all of the following options:<table>
<tr><td>Details:</td><td>Displays the entity's details browser</td></tr>
<tr><td>Map View:</td><td>Opens a new map window in AutoCAD containing the highlighted entity.</td></tr>
<tr><td>List:</td><td>Displays the list of all entities of the highlighted type.</td></tr>
<tr><td>Floorplan:</td><td>Displays a floorplan view of an Inside Plant facility.</td></tr>
<tr><td>Rack Elevation:</td><td>Displays a rack elevation view of highlighted Inside Plant equipment.</td></tr>
<tr><td>Detail Drawing:</td><td>Displays a connectivity detail schematic
showing the highlighted piece if inside plant equipment</td></tr>
<tr><td>Highlight</td><td>Displays the chosen object on the map with a highlight so that it can be identified.</td></tr>
</table></td></tr>
<tr><td>  </td><td>Begins the operation for adding a new instance of the highlighted type.</td></tr>
<tr><td>  </td><td>Opens the edit panel for the highlighted entity.</td></tr>
<tr><td>  </td><td>Deletes the highlighted entity.</td></tr>
<tr><td>  </td><td>Allows you to associated file system documents or URLs (e.g. web site addresses) with the highlighted entity, or retrieve existing associated documents or URLs.</td></tr>
<tr><td>  </td><td>Closes the current panel.</td></tr>
</table>

Many more buttons are provided for performing context-specific operations which only apply to certain types of entities in certain situations. These are described in the documentation for each function throughout this manual.

If a button is inactive, it means that the option is unavailable because the current set of conditions means that it is not relevant. For example, a button might only become active when the correct type of entity is selected.

Confirmation dialog box

SPATIAL*net* will display a confirmation dialog box when certain operations (for example **Delete**) are selected. The **Confirmation** dialog box displays a list of entities that are affected by the operation. For the **Delete** command, the **Confirmation** dialog box shows all the entities that will be deleted as a result of deleting the specified entity. You must then confirm the delete operation or cancel it.

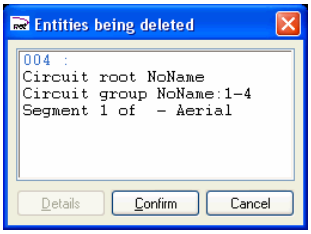


Figure 8 - The Confirmation dialog box

To confirm the operation for all entities shown in the dialog box, click **Confirm**. This deletes all the listed entities, regardless of whether or not they are highlighted.

To find out more information on a single entity displayed in the dialog box, highlight the entity. The **Details** button becomes active. Click **Details**. The **...Details** dialog box for the selected entity appears.

Note: When you select Confirm, you can only reverse the action by using the SPATIAL*net* Undo command. Undo cannot be used to reverse any command that was used to create, change, remove or restore a view.

Graphical Display of Entities in SPATIALnet FM

The display symbology of an entity in SPATIALnet FM is specified in the configuration of that object. SPATIAL*net* looks up this configuration each time it needs to generate a CAD representation of the entity. Since CAD files are not used to store data, all symbology is dynamically regenerated each time the system queries data from the database and displays it into the AutoCAD window.

Changing window focus during commands

SPATIAL*net* dialog boxes are provided as a form of communication between you and SPATIAL*net*. Basic Windows functions such as **Close** and **Minimize** are available. Similarly, while it is possible for many windows and dialog boxes to be open at a time, only one of these can be the focus of the keyboard/mouse commands.

It is necessary for you to manually switch the focus to alternate between windows and dialog boxes.

At any point, if you want to make any changes to settings on a window, you can switch back to the window, change the settings and then switch back to the view to continue the operation.

To switch the focus of commands to the active view, either click on the AutoCAD title bar or at a point within the view. This action is known as *re-activating a window*.

To switch the focus to a window, click anywhere inside the window. This action is known as *re-activating a window*.

Saving work

For most operations run from SPATIAL*net* commands, the system will write any changes you make to the database as you go. You will not need to explicitly save your work and if the system crashes, you will at worst only lose changes for a single operation.

For CAD-Scrape changes, you save your work using the normal save process in AutoCAD. That is, use the **File > Save** command, **Ctrl-S** or any other means for saving to disk. This will also save the change to the database.

If you have created a view and would like to retain it for use during a future SPATIAL*net* session, the view must be saved using the View Manager.

Terminating (aborting) a command

Some SPATIAL*net* commands require you to perform multiple actions, such as entering some information into a dialog box and using the mouse to select a placement position on the screen. At most points during the execution of a command, the operation can be aborted by pressing **ESC**.

Chapter 2

General SPATIAL*net* Functions

This chapter provides a brief but thorough tutorial which walks you through several basic processes in the system. The tutorial is intended to help you familiarize yourself with the concepts and operation of the software. If you have not used SPATIAL*net* FM before, working though this tutorial is a good place to start. Before beginning, however, you should read the Introduction in the previous chapter.

The tutorial is divided into two parts, one for outside plant and the other for inside plant. The inside plant part is presented in two sections, the first dealing with the physical network and the second with logical circuits.

The topics covered in each part of the tutorial are:

Part A: Outside Plant (HFC and Fiber To The Home)

- Importing some CAD backdrop with landbase and strand maps.
- Placing a new facility.
- Placing a new HFC node.
- Connecting the facility to the node with a new fiber cable.
- Splicing the fiber to create a traceable optical path between the facility and the node.
- Altering landbase.
- Creating a FTTX distribution cabinet site.
- Creating support strand and poles.
- Creating subscriber addresses.
- Creating fiber taps and associating with addresses and poles.
- Connecting Fiber and “tailing off” into taps.

Part B: Inside Plant

(Section 1) Physical Network

- Creating a site for recording inside plant.
- Adding floors, floorplans and equipment (racks, chasses, cards).
- Displaying rack elevation and connectivity diagrams.
- Connecting equipment.

- Connecting to Outside Plant.
- Tracing physical connectivity between inside and outside plant.

(Section 2) Logical Circuits

- Creating a logical protocol.
- Creating a customer and a logical service.
- Mapping the service to inside plant equipment.
- Configuring Routing and Multiplexing for the service.
- Tracing the logical service.

Notes:

4. **Each step in Parts A and B of the tutorial builds on activities in the earlier steps from that part, so you must complete the steps in the order they are presented.**
5. **Parts A and B are independent. You may choose to do either or both of these, and may do them in any order. Sections 1 and 2 of Part B must be done in the order presented.**
6. **You do not need to complete the entire tutorial in a single session. You can quit at any stage, log back in again at a later time and continue where you left off.**

Part A: Outside Plant Fiber Engineering and Management.

This part of the tutorial walks you through the outside plant fiber engineering and management functions of SPATIAL*net* FM. In the example, you will perform an end-to-end job for designing a new fiber route for supplying signal to a node in a HFC network. After completing this tutorial, you will be able to use SPATIAL*net* FM in most outside plant situations. Where the tutorial does not cover a subject, or does not fully cover a function, complete details are available in the later sections of this manual.

Step 1: Logging On and Selecting a Job

The first thing to do when starting any SPATIAL*net* session is log on to the system. To start this process find the **SPATIAL*net* FM** icon on the desktop. It should look something similar to this:

(Alternatively, find **SPATIALnet > SPATIALnet FM** under **Programs** on the Windows **Start** menu.)

Depending on the level of installation, you may also have the icons shown below. However, these applications are not covered in this Training Manual.

1. Double-click on the **SPATIALnet FM** icon to start SPATIAL*net*. You will see the SPATIAL*net* splash screen, which will disappear after about 10 seconds (**Hint: You can close the splash screen at any time by clicking on it**).
2. After a short time, the SPATIAL*net* login panel will appear, as shown below.

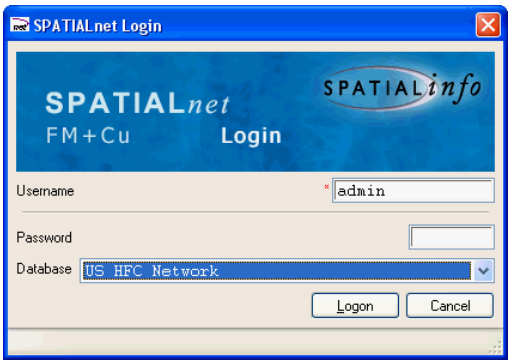


*Figure 9 - The SPATIAL*net* login dialog box*

3. Type in your username and password, and select the database you wish to connect to (if you do not know what your username or password is, or you do not know which database to connect to, see your SPATIAL*net* administrator, or refer to the *SPATIALnet FM Administration Manual*).

   After you log in, the system will display the Job Selection Dialog box, similar to that shown below:

Figure 10 - Job Selection dialog box

4. Click on the **New** button to create a new Job, and supply a Job Number and Description that will help you identify it later and re-connect to it in future sessions. Do not check the "Job is to be posted before construction" checkbox, select Plant Update for the Job Type, and for RF Design Profile select "(no RF profile)". When done, click the **Add** button to create the new job. This will also automatically start AutoCAD.

Notes:

1. **After you have created a new job, you can open it in future sessions by selecting it in the list shown in Figure 10, and clicking the** Open **button.**
2. **If this is the first time you have logged on to SPATIALnet FM on the machine you are using, or if you have just re-installed AutoCAD, you may see an alert box similar to the one shown below:**

Figure 11 – AutoCAD No Profile name alert box

If the above alert box is displayed, click OK and allow the startup process to continue (SPATIALnet creates an AutoCAD profile named `spatialfm` the first time it starts. After this, the profile should be available for use and the alert will not be displayed in future sessions).

5. When AutoCAD starts up, you should see a blank drawing, and a number of toolbars.

Note: If this is the first time you have run SPATIAL*net* on the machine you are using, or if you have just re-installed AutoCAD, you may see a large number of toolbars that you will not need to run SPATIAL*net* FM, or this tutorial. The menus and toolbars required are shown in Figure 12, below. You should verify that your AutoCAD environment contains these. Refer to AutoCAD user documentation for making toolbars and menus visible.

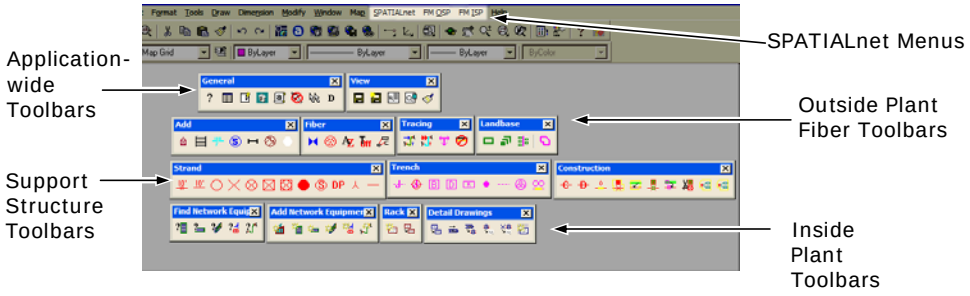


Figure 12 – Toolbars and Menus for SPATIALnet FM in AutoCAD

Note: The toolbars may be docked or may not be visible as shown in Figure 12. Right-clicking on any visible SPATIAL*net* toolbar pops up a menu that allows you to set the visibility of other SPATIAL*net* toolbars. For further information on managing toolbars and menus, see the MENULOAD command in the AutoCAD Users Guide.

This completes the startup process. We are now ready to import a CAD file which will form the backdrop of our fiber engineering job.

Step 2: CAD-Scrape Landbase and Strand

Now that we have a running session, let's start by importing (in the language of SPATIAL*net*, "CAD-Scraping") an AutoCAD DWG file which contains the landbase and strand underlying the network we will create in the next steps.

The steps are:

1. Run the AutoCAD **File > Open** command.
2. Navigate to the directory containing your SPATIAL*net* documentation (usually Program Files/SPATIALinfo/SPATIALnet/blklib/dwg)
3. Open the AutoCAD file **Landbase-Strand.dwg**.
4. When the file is opened, SPATIAL*net* will display the following panel:

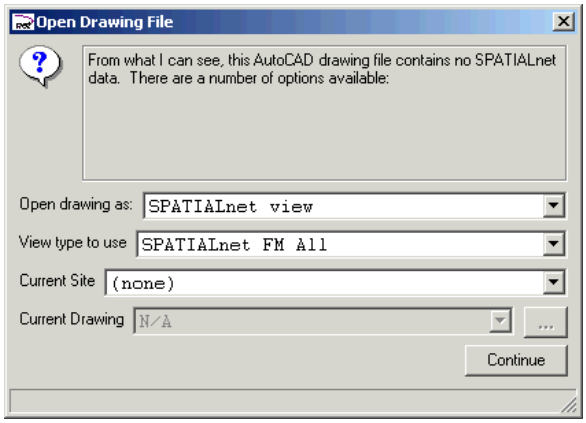


Figure 13 – SPATIALnet Open Drawing File Dialog Box

5. Select **SPATIALnet View** from the **Open Drawing as** pulldown list, make sure the other fields are filled in as shown in Figure 13 above, and click on the **Continue** button.
6. When the graphics are visible in the AutoCAD window and the Open Drawing File Dialog has closed, run the **File > Save** AutoCAD command to import the graphics into the SPATIAL*net* database (running the Save command will automatically initiate the CAD-Scrape process. The Scrape process may take a few minutes. See Chapter 10 for further information about CAD-Scrape).
7. When the scrape is complete, the system will apply the Layer Visibility settings to the objects imported into the database. This may cause some of the objects to no longer be visible. See Step 4, below, for controlling the visibility of objects in the database.

When this process is complete, we now have in place the landbase and strand relative to which we will create our new fiber network.

Notes:

1. **You can also CAD-Scrape any AutoCAD objects created during the session using normal AutoCAD drawing commands. You don't always have to open a DWG file. You do have to have a SPATIALnet view open, however (not a native CAD file).**

2. **Any blocks scraped in must be defined either in a DWG file on the AutoCAD Support File path (the usual method), or in the view template DWT file. CAD-Scrape only records block names; it does not read the block definition into the database.**
3. **Properties of CAD Layers must be defined in the view template DWT file. If the system renders a CAD object into a layer which is not defined in the view template DWT file, the default color, linetype, lineweight, etc. will be used.**
4. **Because of points 2 and 3, the objects rendered back into AutoCAD after being imported using CAD-Scrape may appear different from the original objects. This can be fixed by making the Layer and block definitions available as described in points 2 and 3.**

Step 3: Set Cad Layers, View Template & Symbol Search Path

Every object which is represented graphically is assigned to a layer in AutoCAD (this applies to both CAD-Scraped and Modeled entities). The CAD layers typically define such properties as color, linetype etc. However, when SPATIAL*net* scrapes a CAD object into the database, it only saves the *name* of the layer on which the object is drawn, and *not* the properties associated with the layer.

In SPATIAL*net* the definitions of the layers and their properties are stored in a CAD template file which is loaded whenever SPATIAL*net* opens a new drawing window.

To select a template, use one of the following methods:

a. Run the **SPATIALnet > View > Settings...**. command.

Or

b. Click the **View Settings** button on the **View** toolbar, as shown below.

This will display the **View Settings** dialog box. Click the **Edit** button on the **View Templates** option to display a list of template files.

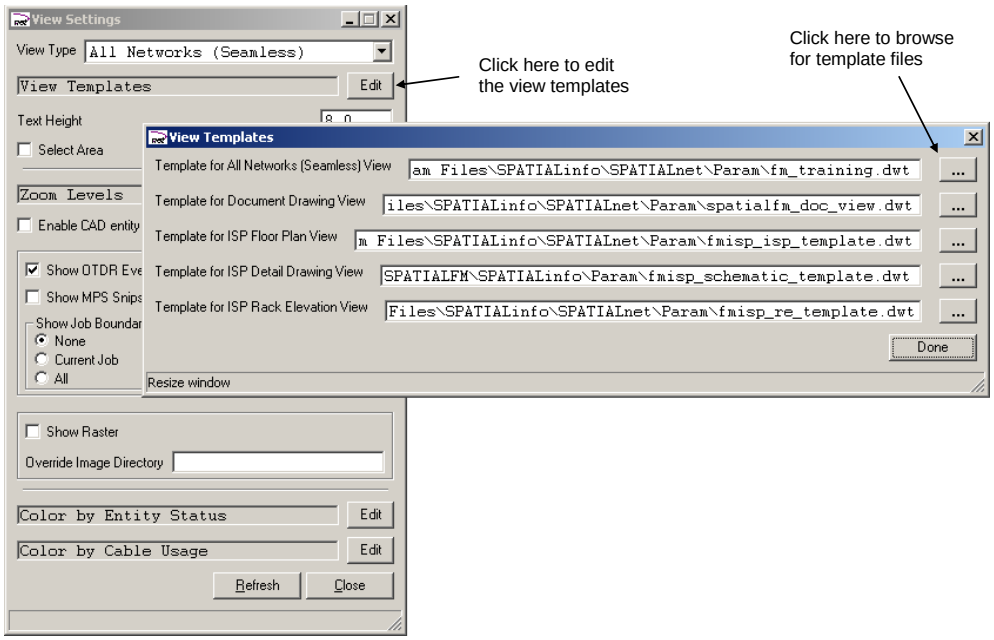


Figure 14 – SPATIALnet Open Drawing File Dialog Box

Click the ... button to browse for templates files. Check that the `fm_training.dwt` template is selected, (select it by navigating to the directory containing the file, highlighting it, and clicking the **Open** button in the **Select Template** panel). Then click **Done** in the **View Templates** dialog box.

Notes:

1. **The template files configured for each view type are saved against your user profile. So if you log on as a different user, you may get a different template.**
2. **If the template is absent, or the wrong template is used, all the CAD objects will most likely be displayed in black and white.**

Step 4: Control Object Visibility

SPATIAL*net* allows you to control the scale at which different types of CAD objects are made visible when a view is rendered from the database. By hiding detail not required in wide-scale views, view creation times can be dramatically reduced.

For this exercise, we will make all objects visible at all scales. To do this:

1. Run the menu command **SPATIALnet > View > CAD Layer Filter**.
2. Select all rows in the layer list (select the first row, scroll to the last row, hold down the **Shift** key and select the last row).

3. Click the **Edit Zoom** button on the dialog box shown in Figure 15, below.
4. Set the Zoom Level to **Zoom Level 5 (far)**, as shown in Figure 15, below.

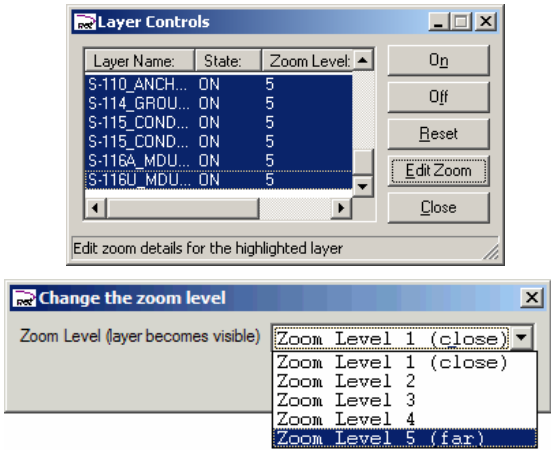


Figure 15 – Layer Visibility Control and Edit Zoom panel

5. To apply the visibility changes, refresh the CAD window by re-querying data from the database. This is done by clicking on the **Refresh** button on the SPATIAL*net* **View** toolbar, shown here:

*Figure 16 – **Refresh** button on **View** toolbar.*

Setting Zoom Level Extents

The above step enabled us to set the zoom level at which each object becomes visible, however it didn't allow us to specify the screen extent at which each zoom level becomes active. (The screen extent is the size of the area covered by the screen.)

To specify the screen extent assigned to each zoom level, use one of the following methods to display the **View Settings** dialog box.

a. Run the **SPATIALnet > View > Settings...**. command.

Or

b. Click the **View Settings** button on the **View** toolbar, as shown below.

Click the **Edit** button next to the **Zoom Levels** option to display the Zoom Levels dialog box.

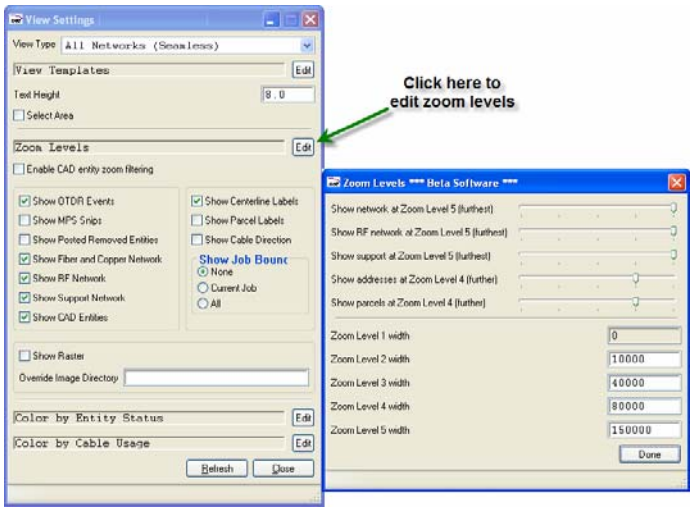


Figure 17 – View Settings & Zoom Levels dialog box

Step 5: Save View

We have just added a new landbase area to our database, but we don't have any defining features which will allow us to easily navigate back to this area in future sessions. There are two ways to make this easy: "Saved Views" and "Boundaries". We will save a view of this area in this step, and create a Boundary in the next step.

A Saved View is like a bookmark or shortcut to a specific area in the database. Once you have created a saved view, you can instantly navigate back to that area by restoring the view.

To find the area we wish to save, locate the area labeled SHARED HUB SITE in the new landbase we have just scraped. (Hint: You can use the AutoCAD **Edit > Find** tool to locate this text (by typing any single word from the string) and zoom directly to the area.) The area in question is shown below.

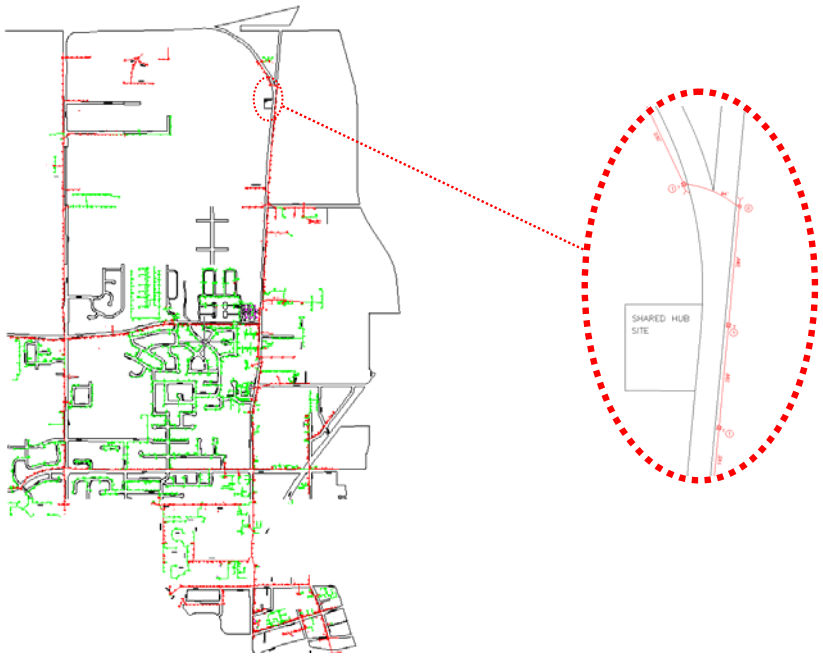


Figure 18 – Location of Shared Hub Site in sample Landbase-Strand drawing

Now we will save a view which will allow us to navigate directly to the shared hub site in future sessions:

1. Zoom the AutoCAD window to an extent centered on the SHARED HUB SITE shown in Figure 18, above.
2. From the menu, run the command **SPATIALnet > View > Save View**.
3. Fill in the fields on the **Save View As** dialog box, as shown below.

Note: Fields flagged with a * are mandatory.

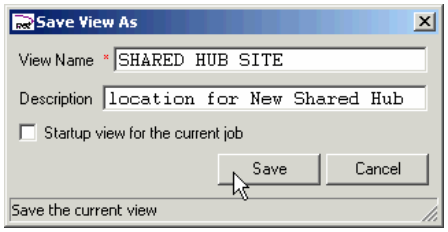


Figure 19 – Saved View details

4. Click the **Save** button.

To retrieve a saved view, run the menu command **SPATIALnet > View > View Manager**. Select the view you wish to restore from the list of saved views and click the **Open>>** button. This will automatically zoom the view window to the extent you saved, and will refresh the contents of the view by re-querying from the database. Checking the startup view for current job checkbox will set the saved view as the default startup view so that everytime this job is opened it will automatically zoom to this location.

Step 6: Configure a New Boundary Type

To make it easier to locate and manage areas within the map, it is often convenient to create a boundary around a specific area, such as a county, a city, the area serviced by a single hub, etc. You may already have these boundaries in your existing landbase and/or network maps. Once it is saved as a boundary in SPATIAL*net*, you can easily search the database for an area you require and zoom to it directly. The differences between a Boundary and the Saved View from Step 5 are:

- A Saved View is visible only in the job in which it was created. Other jobs cannot see it, and it disappears when the job is Posted or Cancelled. A boundary, however, is visible to everyone once it is Published or Posted, and persists in the database until it is deleted.
- The outline of a Saved View is not visible on the map, whereas a boundary is.
- There is only one type of Saved View, but you can configure as many types of Boundary as you wish.

As implied by the last bullet point, before you can create a boundary, you must first ensure that the type of boundary you with to create is in the **Boundary Definitions** dictionary. Let's assume it is not, and use this example as an opportunity to configure a new type of Boundary. Looking ahead to the type of site we will add in Step 7, let's call this new type of boundary a "Shared Hub" boundary.

1. From the menu, select **SPATIALnet > Dictionaries > Landbase Definitions > Boundary Definitions...**. A dialog box will be displayed, listing all the boundary types currently defined.
2. Click on the **Add** button and fill in the field values as shown in the dialog box below:

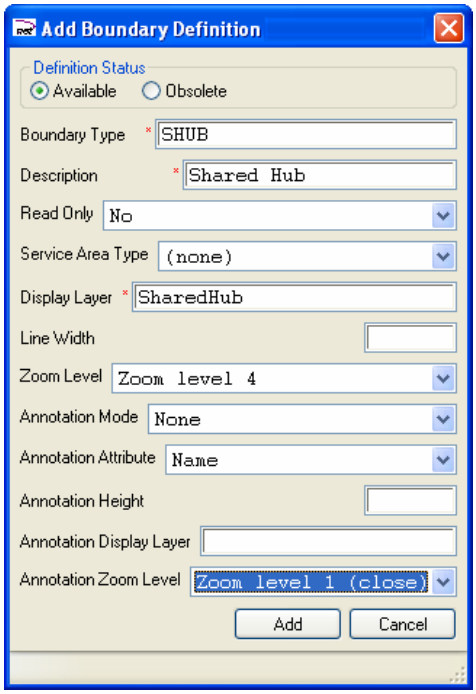


Figure 20 – New Boundary Dictionary Entry Form

3. Click on the **Add** button shown in Figure 58 to create the new Boundary Type. It should now be listed in the **Boundary Definitions** list.
4. Click the Close button on the **Boundary Definitions** list.

Step 7: Add a New Boundary

Now let's draw a boundary of type "Shared Hub" around the map area we imported in Step 2, and shown in Figure 18.

1. Make sure the area shown in Figure 18 is visible on the screen. If it is not, run the menu command **SPATIALnet > View > View Manager…** to bring up the **Saved Views** dialog box, highlight the view named "SHARED HUB SITE", and click on **Open>>**. Zoom until you have the area you need in the window, and use the **Refresh** button on the **View** toolbar if you need to bring in more data from the database.
2. From the menu, select **SPATIALnet > Add > Landbase > Boundary…** to bring up the **Boundary Creation** dialog box.
3. Fill in the fields of the **Boundary Creation** dialog box with the values shown below.

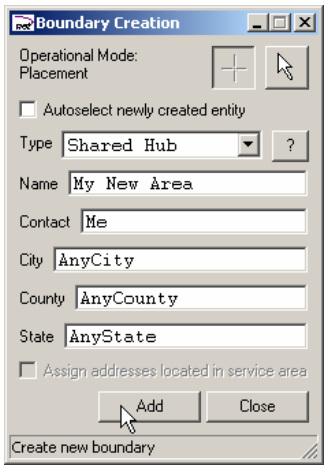


Figure 21 –Boundary Creation dialog box

4. Click the **Add** button shown in Figure 21. This will bring up the **Line Capture Control** panel, as shown in Figure 22, below.
5. With the **Vertex Mode** cross-hair button selected in the **Line Capture** control panel (see Figure 22, below), click on each point at which you wish to place a vertex of the boundary.
6. Click the **Close** button as shown in Figure 22, below when done. This will close the boundary polygon geometry and complete the addition of the new boundary.
7. Click the **Close** button on the **Boundary Creation** dialog box. (Clicking the **Close** button the **Line Capture** control panel will refresh the **Boundary Creation** dialog box, allowing for a new boundary to be created, if needed.)

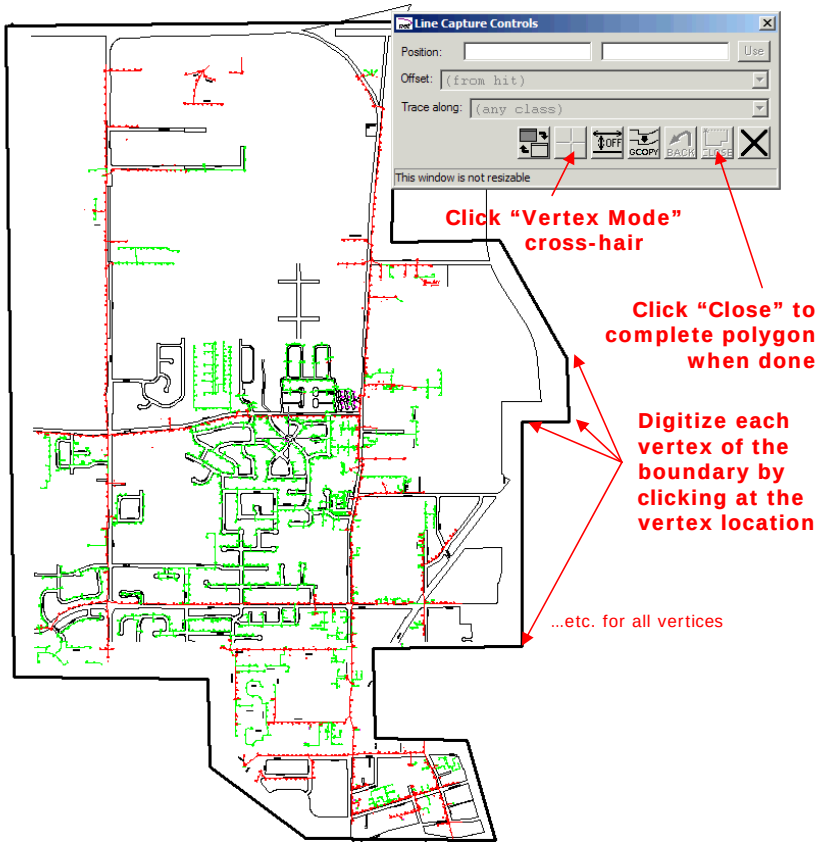


Figure 22 – Capture Boundary Vertices

Step 8: Editing Boundaries

Once you have added a boundary it may become necessary to edit or change its properties or geometry.

Editing Boundary Properties

To edit a boundary's properties, first click on the boundary, and then click on the **Edit Selected Entity** button on the **General** toolbar.

Figure 23 – Edit Selected Entity button on General toolbar

This action will display the **Boundary Modification** dialog box. Edit the properties as required, then click **Apply.**

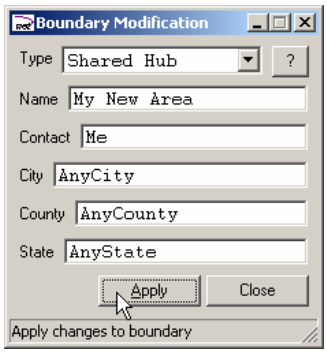


Figure 24 – Boundary Modification dialog box

Editing a Boundary Geometry

Moving a Vertex

To change the geometry of a boundary by moving a vertex using SPATIAL*net* manipulators:

1. Click on the manipulator and drag it to the required position and click on the mouse button.
2. A pop-up selection box will appear. Highlight and click on the **Move to location...** option.
3. The boundary geometry will now be updated to include the newly updated vertex.

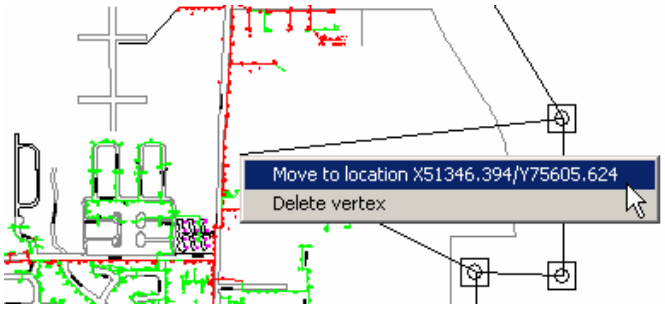


Figure 25 – Moving a vertex

Deleting a Vertex

To change the geometry of a boundary by deleting a single vertex:

1. Click on the manipulator at the vertex to be deleted, drag it, then click on the mouse button.
2. A pop-up selection box will appear. Highlight and click on the **Delete vertex** option.
3. The boundary geometry will now be updated with the deleted vertex removed.

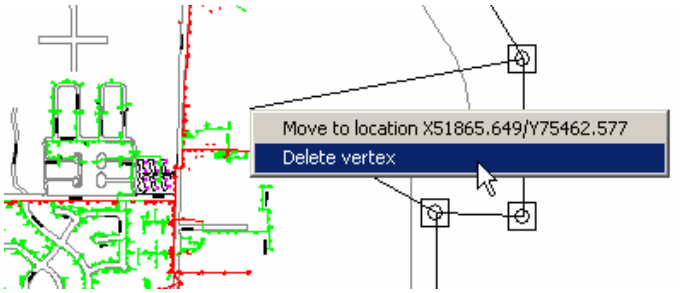


Figure 26 – Deleting a vertex

Deleting Multiple Vertices

To change the geometry of a boundary by deleting multiple vertices in a sequence:

1. Click on the manipulator at one of the end vertices, and drag it and place it over the vertex at the other end of the sequence.

   Note: The selected end vertices will not be deleted, only the intervening vertex/vertices will be deleted.

2. A pop-up selection box will appear. Highlight and click on the **Delete multiple vertices** option.

   Note: When dragging the first end manipulator, make sure that you place the mouse directly over the corner of the second end manipulator, or else the Delete multiple vertices option will not appear.

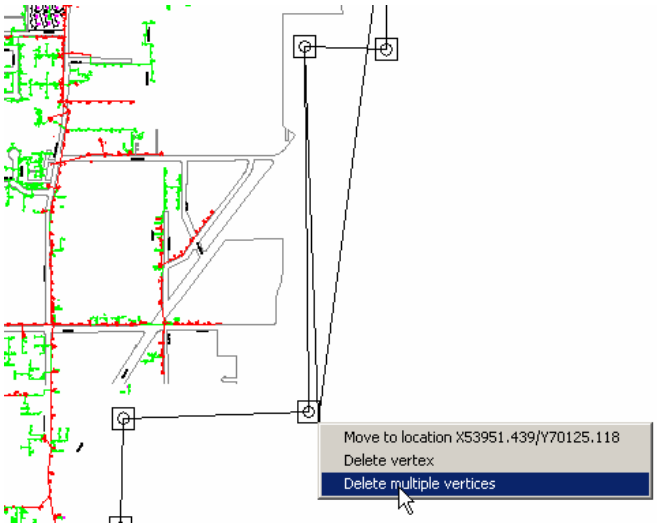


Figure 27 – Delete multiple vertices

3. The boundary geometry will now be updated with the deleted vertices removed.

Step 9: Working with Manipulators and Linear Edit Modes

Manipulators provide many different ways of editing linear geometry. Linear manipulators are controlled by the **Manipulator Edit Modes** panel, which we will learn to use in this step.

To display the **Manipulator Edit Modes** panel:

1. Select the **Spatial*net* > Linear Edit Modes...** option which will produce the **Manipulator Edit Modes** panel, shown below.

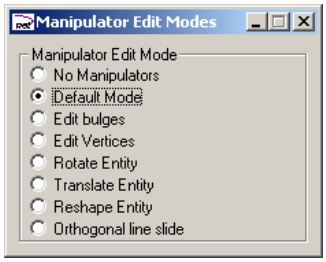


Figure 28 – Manipulator Edit Modes Panel

The **Manipulator Edit Modes** panel contains the following fields:

No Manipulators

Selecting this option removes the manipulators from the map view.

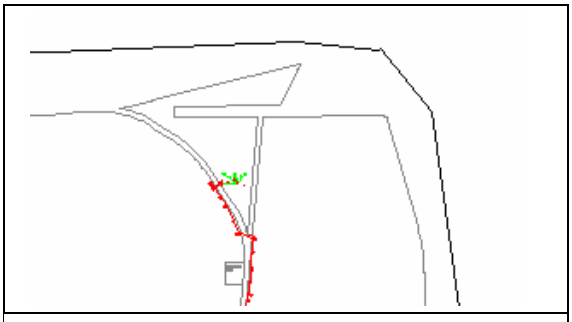


Figure 29 – Boundary view without manipulators

Default Mode

This option displays the manipulators on the map view.

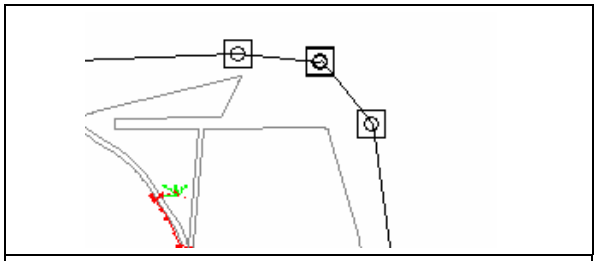


Figure 30 – Boundary view with manipulators

Edit bulges

Note: "Bulge" is defined mathematically as $\tan(\theta/4)$ **where** θ **is the angle included in the arc.**

Selecting the **Edit bulges** option on the **Manipulator Edit Modes** panel (see Figure 28) places an additional manipulator (displayed in red) in between the default manipulators. Clicking and dragging a red manipulator "bulges" a line segment allowing polylines to include curved arcs.

To edit a "bulge":

1. Click and drag a red manipulator.

2. When the desired curvature is achieved, left click the mouse, and then click on the pop-up, as shown below.

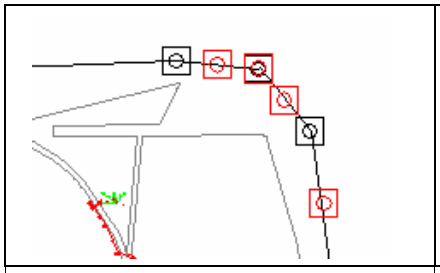


Figure 31 – Additional red manipulators placed between default manipulators

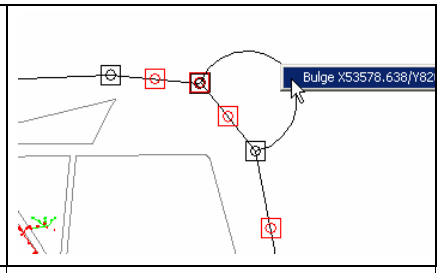


Figure 32 – Red manipulator "dragged" to produce an arc.

Edit Vertices

Selecting the **Edit Vertices** option on the **Manipulator Edit Modes** panel (see Figure 28) places an additional manipulator (displayed in blue) in between the default manipulators. Clicking and dragging a blue manipulator inserts a vertex into an existing line segment.

To insert a vertex into an existing line segment:

1. Click and drag a blue manipulator.
2. When the desired line shape is achieved, left click the mouse, and then click on the pop-up, as shown below.

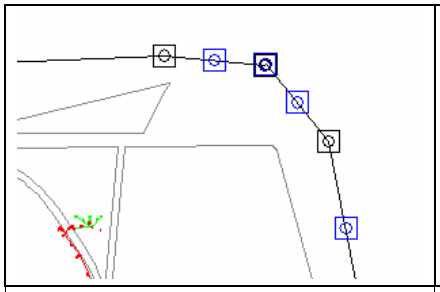


Figure 33 – Additional blue manipulators placed between default manipulators

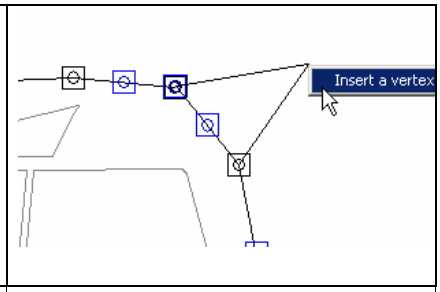


Figure 34 – Blue manipulator "dragged" to produce a line segment.

Rotate Entity

Selecting the **Rotate Entity** option on the **Manipulator Edit Modes** panel (see Figure 28) enables the user to rotate the boundary about its centroid. The user can either manually rotate the boundary, or else specify the rotation angle by typing in the desired number.

To rotate an entity, either:

1. Click and drag a manipulator.
2. When the desired rotation is achieved, left click the mouse, and then click on the pop-up, as shown below.

Or

1. Click and drag a manipulator.
2. Type in the desired angle of rotation (in degrees), and press **Enter**. A pop-up will appear. Click on the pop-up, as shown below.

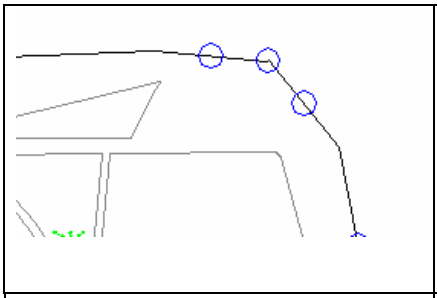


Figure 35 – Rotation manipulators

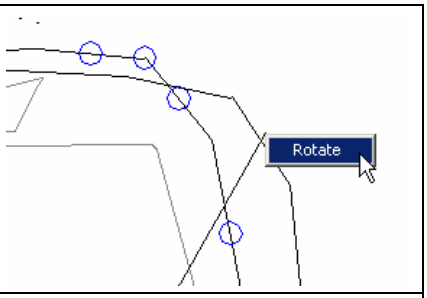


Figure 36 – Rotation manipulator "dragged" to rotate boundary.

Translate Entity

Selecting the **Translate Entity** option on the **Manipulator Edit Modes** panel (see Figure 28) enables the user to move the entire boundary to a new location without changing its shape, scale, or rotation.

To translate an entity:

1. Click and drag a manipulator.
2. When the desired location is achieved, left click the mouse, and then click on the pop-up, as shown below.

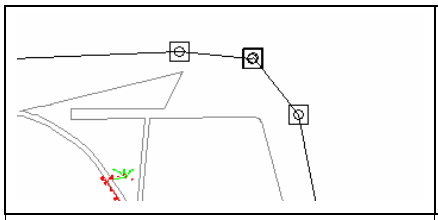


Figure 37 – Translate manipulators

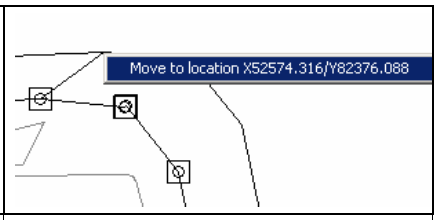


Figure 38 – Translate manipulator "dragged" to new location.

Reshape Entity

Selecting the **Reshape Entity** option on the **Manipulator Edit Modes** panel (see Figure 28) places an additional manipulator (displayed in red) in between the default manipulators, enabling the user to reshape the boundary. When a red manipulator is clicked on and dragged, the **Line Capture Controls** panel is displayed, allowing the line segment to be replaced by a new set of segments captured using the tools on the line capture panel. (See ***Spatialnet FM User Manual*** for details on how to use the **Line Capture Controls** panel.)

To reshape an entity:

1. Click and drag a red manipulator.
2. Enter the desired data into the **Line Capture Controls** panel.

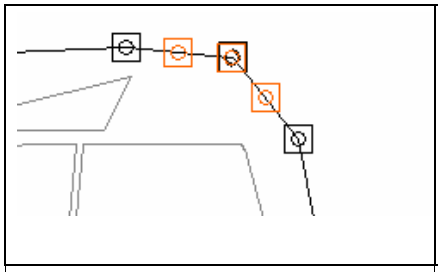


Figure 39 – Reshape Entity manipulators

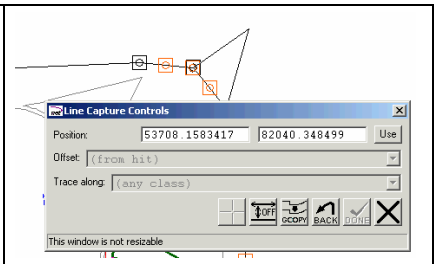


Figure 40 – Reshape Entity manipulator "dragged" to display Lind Capture Controls panel.

Orthogonal Line

The option is not used for OSP.

Using AutoCAD Grips instead of SPATIAL*net* Manipulators.

It is possible to use the normal AutoCAD grips to edit the geometry of SPATIAL*net* entities, instead of manipulators. This allows you to use AutoCAD drafting functions to edit the geometry of modeled entities. However, it also disables the ability to select

modeled entities from the map and run the SPATIAL*net* functions associated with them. "Native Grips" are useful when:

- The geometry editing you need to perform is not supported by SPATIAL*net* manipulators (e.g. the AutoCAD STRETCH command), or is more easily done using AutoCAD functions.
- You wish to edit many objects at once (e.g. move all objects within an area 120 feet to the north).

To use normal AutoCAD grips instead of SPATIAL*net* manipulators:

1. Click the **Current Selection** button  on the **General** toolbar. The will display the **SPATIALnet Current Selection** panel.
2. Click on the **Selection Display:** dropdown menu, and highlight and click on the **`Native Grips`** option, as shown below.

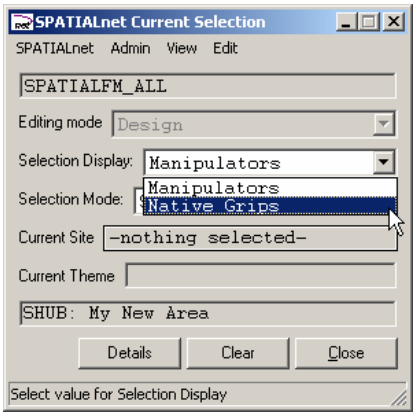


*Figure 41 – SPATIAL*net* Current Selection panel*

The differences between **Native Grips** and **Manipulators** modes are described below:

	Native Grip Mode	**Manipulator Mode**
Grip Display	Normal AutoCAD Grips	SPATIAL*net* Manipulators
Selection Behavior	Additive (selecting one or more objects adds them to the current selection set)	Depends on whether Single or Multiple selection mode enabled.
Drafting Operations	Normal AutoCAD drafting operations available for all objects.	Normal AutoCAD drafting for CAD-Scraped entities, manipulators for Modeled entities.
Snap Modes	All AutoCAD snap modes work	All AutoCAD snap modes work except for ORTHO.
Save Changes	Modeled entities saved to database immediately, CAD-Scraped entities only saved when the AutoCAD **Save** function is run.	
SPATIAL*net* functions	Most SPATIAL*net* functions are not available for any entities. Eg, it is not possible to select a modeled entity, display its details, edit its properties, etc.	SPATIAL*net* functions are available for modeled entities.

- SPATIAL*net* manipulators will no longer appear, and will be replaced by AutoCAD grips.
- All normal AutoCAD drafting functions will work.
- All of AutoCAD snap modes with work.
- Changes made using the AutoCAD editor will be immediately saved to the database, even for SPATIAL*net* modeled entities, but will still require the use of the **Save** button for CAD scraped entities.
- No SPATIAL*net* functions are available in native edit mode; e.g., it is not possible to select a modeled entity, display its details, or edit its properties.

Note: If the SPATIAL*net* Manipulators option is selected, AutoCAD author modes will NOT work.

Step 10: Configure Network Support Structures

Before we add any FTTX equipment, we must extend the support network along the path the network will follow. SPATIALnet FM provides a configuration dictionary for specifying the types of support structures (Poles, Manholes Vaults, etc.) in your system, and the linear support that connects them (strand, trenches, etc.) The process is very similar to those you have already seen.

As a concrete example, we will configure the Joint-use Pole type, prior to adding a run of Joint-use Poles to the network map.

To configure a type of support structure:

1. Run the menu command **SPATIALnet > Dictionaries > Support Structure Definitions**
2. Select **Joint** from the displayed list.
3. Click the **Edit** button. A Panel similar to that shown below should be displayed. Edit the values to match those shown.

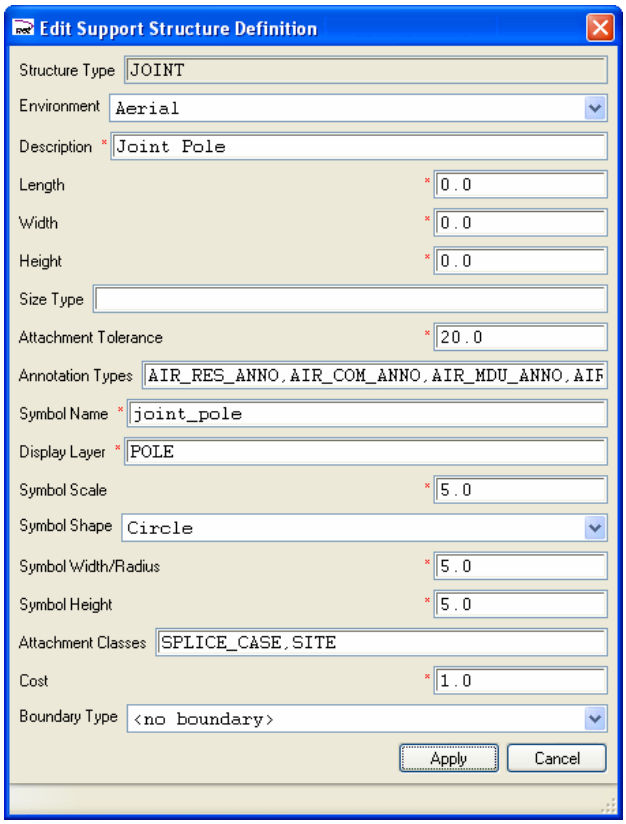


Figure 42 – Configure Network Support Structure.

4. Click the **Apply** button to commit the changes to the database
5. Close both the **Edit Support Structure Definition** panel, and the **Support Structure Definitions** list panel.

Note that the panel shown in Figure 42 allows you to use your own strand symbol library, and configure the system's design functions to work with it.

Comment [ms1]: Is this still valid?

Step 11: Add New Poles and Strand

Having configured the Joint-use Pole type, let's now run a set of Joint Use Poles. To do this:

1. Select the **Joint Pole** button on the **Strand** toolbar.

2. A panel similar to that shown below will appear. Enter desired values for the fields, and click back in the AutoCAD window to place a new pole on the map. (Note the first click will transfer input focus to the map window, subsequent clicks will add a new Pole with the values specified in the dialog box).

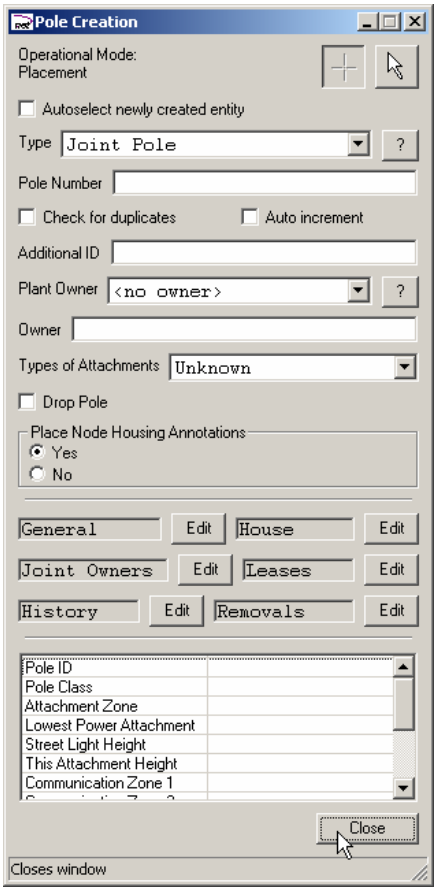


Figure 43 – Pole Creation Panel

3. Run a line of Joint-use poles following the road-edge, similar to that shown below.

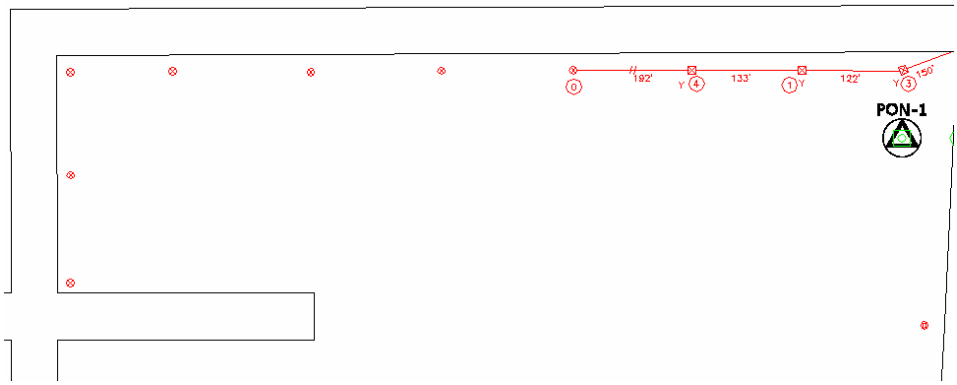


Figure 44 – New Poles

4. To add strand linking the poles, select the **Draw Strand** button on the **Strand** toolbar.

5. Draw a single line passing close to the center of each of the poles. There is no need to draw separate lines between each pole, or to add a vertex at each pole. The system will segment the strand lines automatically.
6. When you have finished drawing the strand lines, run the AutoCAD **Save** command. This will segment the strand line and attach each strand segment to the poles at which it starts and ends.
7. To edit the footages of each strand segment:
   - Select the segment (click on it in the map window).
   - Grab the middle manipulator and drag (and drop) it on the side where you want the distance placed – click on "Add footage to this Side of Line".

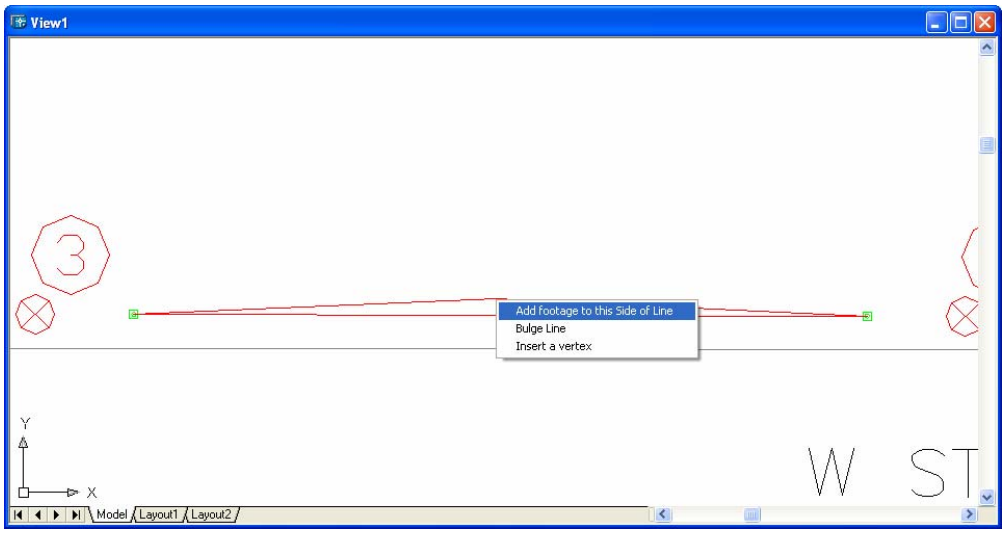


Figure 45 – Add footage pop up

- The Strand Modification dialog will appear, enter the strand segment length, and click **Apply**.

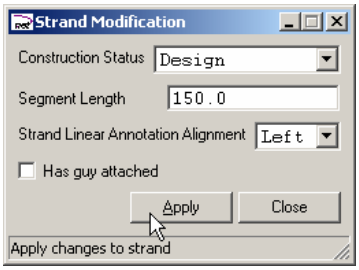


Figure 46 – Strand Modification dialog box

The **Strand Details** dialog box and the map will be updated automatically.

When done, the new strand should look similar to that shown below:

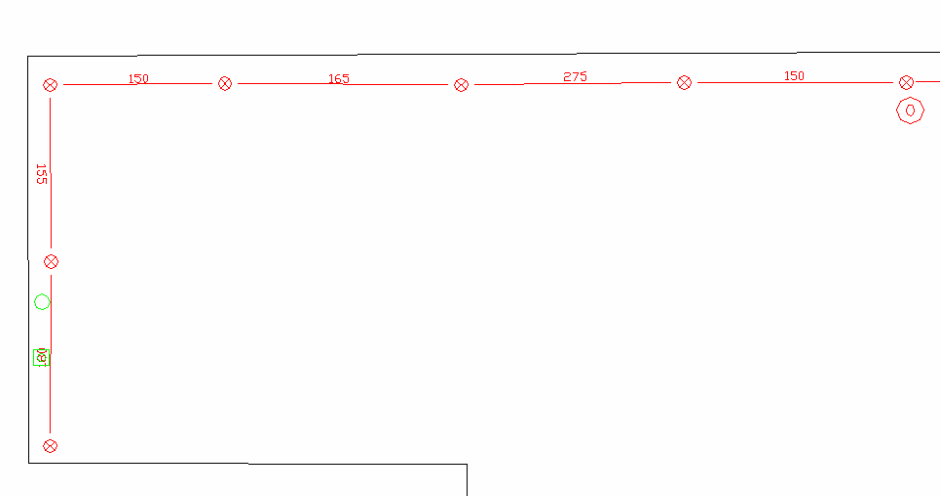


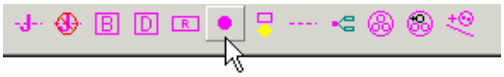
Figure 47 – New strand connecting the poles

Step 12: Add Pedestals

Let's now move to the underground plant, with our first task being the placement of pedestals.

To place a pedestal (or any other underground structure):

1. Click the **Add Drop Pedestal** button on the Trench toolbar, as shown below, or select **SPATIALnet > Add > Support Structure > UG Structure** from the menu.



2. The **Underground Structure Creation** panel similar to that shown below will be displayed.

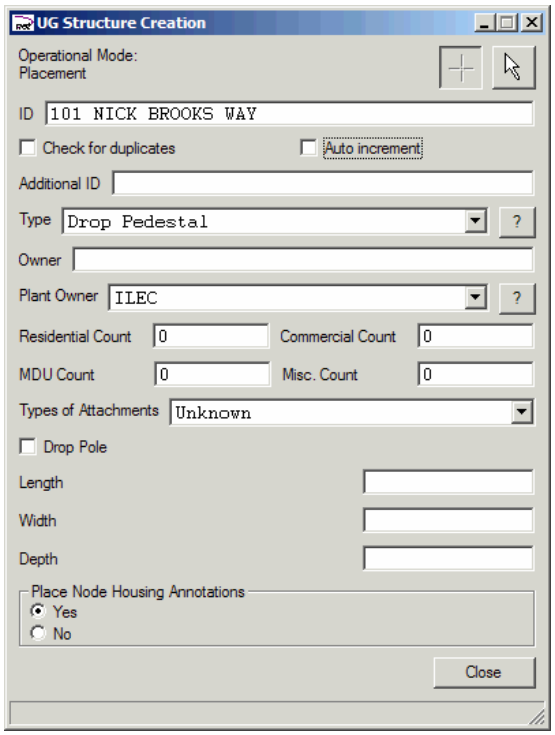


Figure 48 – Underground Structure Creation Panel

3. Choose the structure type to be added, and fill in other information as required.
4. Click on the map location at which each pedestal is to be placed. It will be displayed on the map.
5. Add the remaining pedestals to complete the design.

Step 13: Add Trenches

In SPATIAL*net*, underground routes containing conduits and/or cables are referred to as "Trenches". In this step, we will create the trenches which link the underground housings (pedestals, etc.) that we have placed so far. To create a trench:

1. Click the **Add Trench** button on the Trench toolbar, as shown below, or select **SPATIAL*net* > Add > Support Structure > Trench** from the menu.

2. The trench tool begins the AutoCAD POLYLINE command and sets the currently layer to the layer on which trenches were configured in the

dictionary. Digitize the route of each trench using the AutoCAD drafting functions available in the POLYLINE command.

Notes:

1. **Unlike other linear objects in SPATIALnet, trenches (and aerial strand lines also) are first drafted as CAD Polylines. When the AutoCAD "Save" function is run, SPATIALnet automatically "promotes" each CAD Polyline to a modeled trench line. The length attribute of the line is initially set to 0, and the trench is connected to the start and end housings at either end of the line.**
2. **SPATIALnet will only "promote" Polylines that meet the following criteria:**
   (a) They are on the correct layer
   (b) They begin and end at an underground support structure
   (c) They pass close to the insertion point of any housings part way along the Polyline (SPATIALnet will break the trench into individual segments which begin and end at each intermediate structure).
3. **If any Polyline segments fail to meet the conditions in 2, they will not be "promoted" to trenches. Instead, they will be moved to Layer 0. You may then correct the problem, move the polyline onto the correct layer and try to save/promote again.**

Step 14: Edit Trench Footages

Linear support structures (trenches and aerial spans) provide a unique way for editing their lengths. When a linear support structure is selected in SPATIALnet, a special red manipulator is placed at the center of the line when the linear object is selected. This manipulator can be used to enter or update the length property of the linear support object.

To edit the length of a trench,

1. Select the trench whose length you wish to edit.
2. A red Length manipulator will be displayed, as shown below.

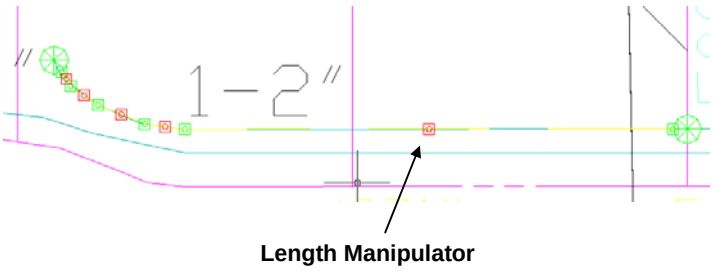


Figure 49 –Edit Length manipulator on selected trench line

3. Drag the manipulator to the side of the trench line on which you want to display the length annotation.

4. Click the **Add footage to this side of line** option on the pop-up, as shown below.

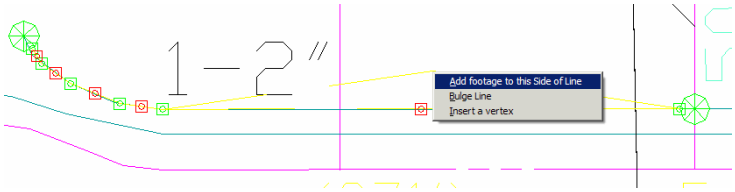


Figure 50 –Selecting side on which length annotation will be placed

5. The Trench Modification panel will be displayed, and the cursor automatically placed in the Length field. Type in the length you wish to assign to the trench segment as shown below.

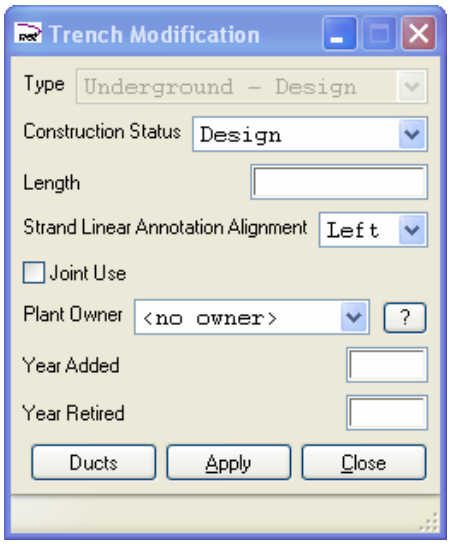


Figure 51 –Editing Length in the Trench Modification Panel

6. Hit Enter on the keyboard to accept the footage and commit it to the database. Annotation should be displayed on the side of the line you chose, displaying the footage you entered.
7. Repeat the above process to specify the lengths of the remaining trench segments in the new subdivision.

Step 15: Add Duct Forms

Having established the routes of the trench and entered the length of each segment, let's now specify the ducts and inner ducts, if any, in each trench.

SPATIALnet has a sophisticated conduit and duct manager, which we will use to add various duct configurations to our subdivision.

Let's begin by adding four 6" PVC ducts in a 2x2 grid to one of the trenches we created. We shall explore how to change the duct configuration shortly.

To add the bank of ducts:

1. Select the trench to which the duct bank is to added.
2. Click the **Add Ducts** button on the Trench Toolbar, as shown below:

3. The **Add Ducts...** panel will be displayed. Fill in the values as shown below and click the **Add** button.

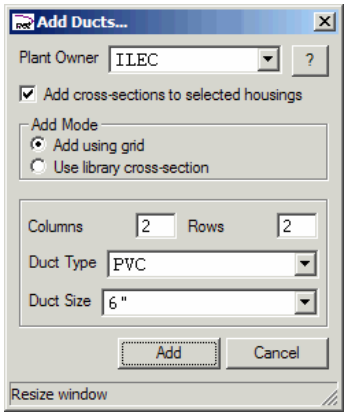


Figure 52 – Add Ducts panel

4. The system will automatically display an array of ducts matching the specifications in the **Add Ducts** panel and place this near to the trench as shown below:

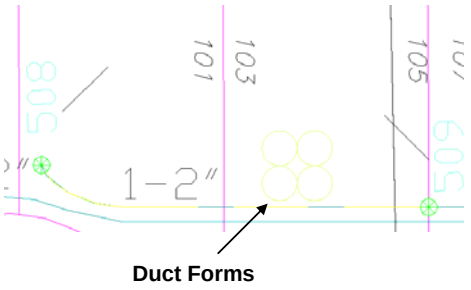


Figure 53 – Add Ducts panel

5. Place other duct forms around the subdivision.

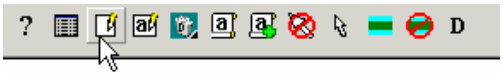
Note: The way the duct forms are displayed depends on the values configured in the Duct Radius dictionary for each Duct Size, and for the Grid Size.

Step 16: Edit Duct Forms

Existing duct forms can be edited, both in terms of the number and configuration of the ducts, and the information associated with each duct.

To edit information ducts in a duct bank:

1. Select either the duct bank itself by clicking on any of the duct graphics, or select the trench segment to which it belongs by clicking on the linework.
2. Click the Edit button on the General toolbar, as shown below.



3. The **Edit Duct Information** panel will be displayed. Select each duct in the list to the left, and edit its properties in the fields to the right. When you wish to commit the changes to the database, click the **Save** button.

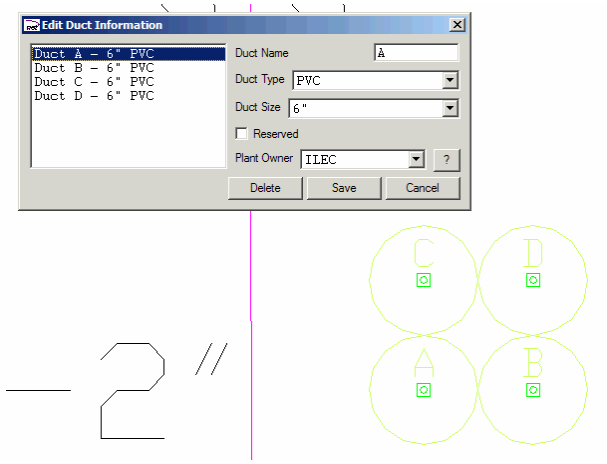


Figure 54 – Edit Duct Information

Note: When the Duct Name of each duct is filled in, SPATIAL*net* automatically displays the name on the duct form shown on the map.

SPATIAL*net* displays manipulators To change the geometry arrangement of the duct forms. A manipulator is provided for each duct, allowing the ducts to be moved individually. Another manipulator is provided which allows the duct bank to be moved and rotated as a whole.

4. Use both the individual and bank-wide manipulators to rearrange the layout of the ducts into a 4x1 row, similar to that shown below.

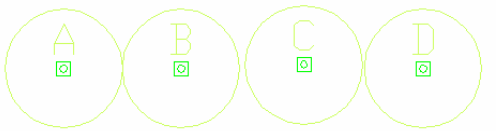


Figure 55 – New Duct Configuration

5. Use the **Delete** button on the **Edit Duct Information** panel to remove duct D from the above configuration.

Step 17: Add New Ducts and Inner-ducts

The number of ducts in a bank can be changed by using the Add Ducts tool. Ducts of any size can be added to a duct bank, and they can be added as inner-ducts, or sub-ducts, of the existing ducts.

To add new ducts to a duct bank:

1. Select the duct bank into which the new ducts are to be placed.
2. The **Add Another Duct** panel will be displayed, as shown below.

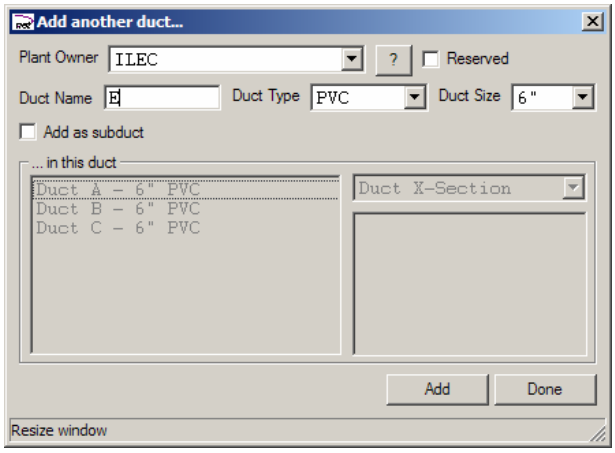


Figure 56 – Add another duct panel

3. Specify the name of each new duct and click the **Add** button to add it to the duct bank.

To add ducts as inner ducts, or subducts, of the existing ducts, check the **Add as subduct** box on the **Add another duct** panel shown in Figure 56. This displays the duct configuration in the right window of the **Add another duct** panel, as shown below. You can choose the outer duct for the new subduct either by clicking on the diagram, or on the list of ducts.

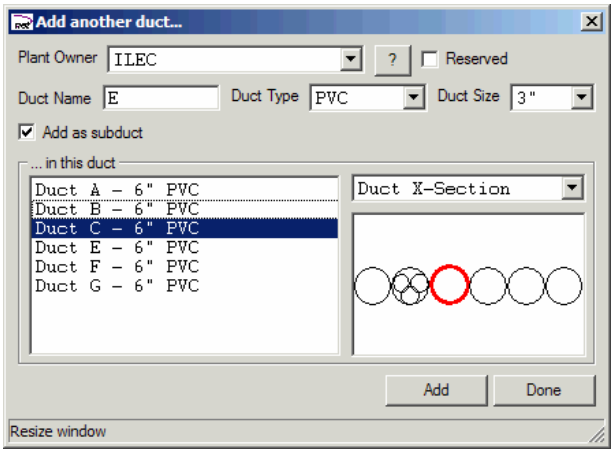


Figure 57 – Add a subduct

Step 18: Add New Site Type

The next thing we will do is place a new network building (also referred to as a "facility" or "site"). Before we can create a new facility and place it on the map, we must first ensure that its characteristics have been defined in the system's *Site Dictionary* (as a general rule, this is true for all network entity types managed by the system). Most of the time, the dictionary will contain the type of object you wish to create and you will not need to perform the actions described in this step. However, assume that this time, the kind of site we need to create—let's call it a **Shared Hub**—is new to the system and therefore, not in the Site Dictionary. Let's see how the characteristics of a new kind of facility (including its display symbology) are stored as a Site Type.

The steps for creating a new Site Type are:

1. From the menu, select **SPATIALnet > Dictionaries > Other Definitions > Site Definitions...**. A dialog box will be displayed, listing all the site types currently defined.
2. Click on the **Add** button and fill in the field values as shown in the dialog box below:

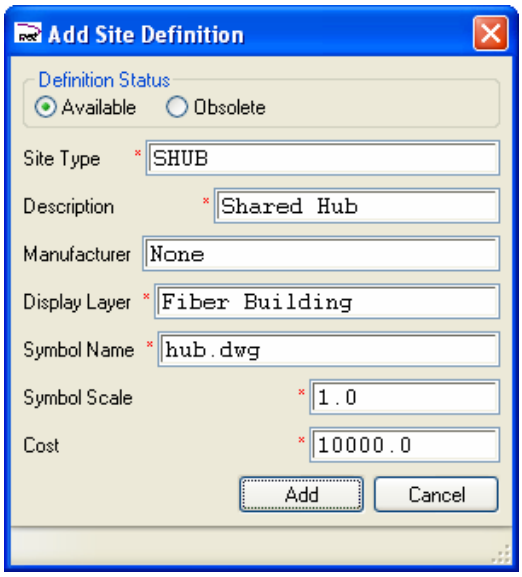


Figure 58 – Add Site Definition dialog box

3. Click on the **Add** button shown in Figure 58 to create the new Site Type. It should now be listed in the **Site Definitions** list.

A very useful feature of the various configuration dictionaries is the ability to nominate certain attributes of an object to be displayed as annotation. This is done by mapping a database field stored against the object to an AutoCAD attribute definition in the block used to represent the object. When such a mapping exists, SPATIAL*net* will automatically fill in the block attributes with the values stored in the associated database field.

An attribute mapping may be of one of the following types:

- Pre-defined field.
- Custom attribute.

Pre-defined fields are those which appear on the various panels by default. These are supplied by the system and cannot be changed.

Note: Pre-defined fields are only pre-defined in the database and on the panels. They are not pre-assigned to AutoCAD block attributes. You must perform the steps below to make them appear as annotation.

In addition to pre-defined fields, the system allows up to 15 custom attributes to be defined. This allows you to add your own properties to each type of object, store these in the database and, if desired, display them as annotation on the map views.

Items 4 to 7 below illustrate the process for adding a new custom attribute to the system. Suppose that you wish to track another piece of information about each Shared Hub, in addition to the attributes that SPATIALnet FM supplies by default. For example, let's say you wish to record the square footage of floorspace your equipment is occupying in the shared facility. To do this:

4. Select the Shared Hub entry you just created in the **Site Definitions** list.

5. Click the **Attributes** button on the list panel. Since there are no attributes mapped for the Shared Hub type, the **Attribute Mapping** list will initially be empty.
6. Click the **Add** button on the **Attribute Mapping** list panel. This will allow you to specify the details of the new attribute you wish to associate with Shared Hub records.
7. Fill in the fields of the **Add Attribute Mapping** panel, as shown below. Click the Add button when done, then close the **Site Definitions** and **Attribute Mapping** panels.

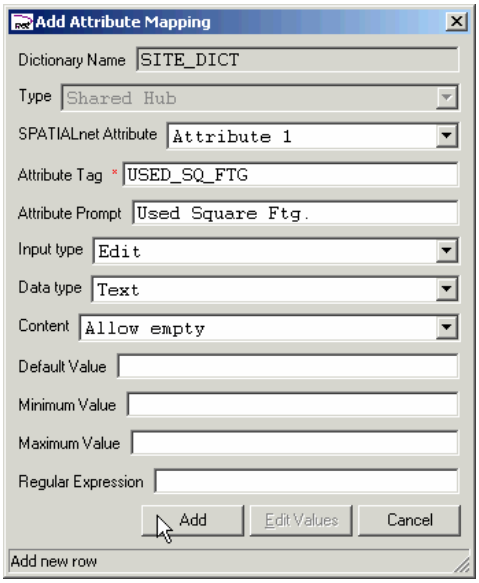


Figure 59 – Adding a custom attribute to the Shared Hub type.

You should also try to map a pre-defined field to an AutoCAD attribute. The **hub.dwg** file already contains an attribute definition with attribute tag **NAME**. This is intended to display the name of the site, but will not do so until an Attribute Mapping is created which maps the pre-defined field **Designation** to the **NAME** attribute tag. You should create this now, using the same process as described in items 4 – 7, above.

Notes:

1. **The SPATIAL*net* Attribute field shown in Figure 59 determines which database column contains the values for the attribute. The choices in the pull-down list fall into two groups:**
   (a) All of the pre-defined fields specified by SPATIAL*net* for the type of object being configured.
   (b) A set of 30 placeholders for custom attributes, labeled SPATIALnet FM Attribute 1 to SPATIALnet FM Attribute 30.
2. **The Attribute Tag field provides you with the option of tying your custom property to an AutoCAD block attribute. This allows you to display the property as annotation text associated with the type's symbol. To**

display this property as annotation using a block attribute:

(a) In AutoCAD, open the DWG file containing the block associated with the type being configured in the dictionary. This will be hub.dwg in the current example (if you are running SPATIAL*net*, open it as a "native drawing"). Note that the block files supplied with SPATIAL*net* (including hub.dwg) are stored in the Symbol\smt sub-directory of the SPATIALnet installation folder.

(b) In the block's DWG file, Create an attribute with a Tag that matches the value you entered into the Attribute Tag field in Figure 59 (USED_SQ_FTG in this example). Details for adding block attributes can be found in the AutoCAD Users Guide under the ATTDEF command.

(c) Save the block DWG file.

3. **The system is supplied with the file hub.dwg, which will be used to represent the Shared Hub in this exercise (you can find it in the Symbol\smt sub-directory of the SPATIAL*net* installation folder). You can create your own block, however, and use it instead.**

4. **If you change any of the display characteristics of a device type, those characteristics will be used to render the device the next time a view is created in AutoCAD. Network characteristics however (such as number of ports) will only affect new instances of the type. Any existing instances will continue to use the values present when they were created.**

Step 19: Add New Site

Now that we have defined the "Shared Hub" site type, we can create an instance of it in the database.

To do this, find the location labeled SHARED HUB SITE and zoom to it, as described in Step 4 above. To add the new site:

1. From the menu, run the command **SPATIALnet > Add > Other > Site...**.
2. Fill in the fields of the Building Creation dialog with the values shown in Figure 60, below (Note: for this exercise, it is important to ensure that the **Capable of modeling inside plant** check box is cleared. We'll visit inside plant in Part B of the tutorial.)

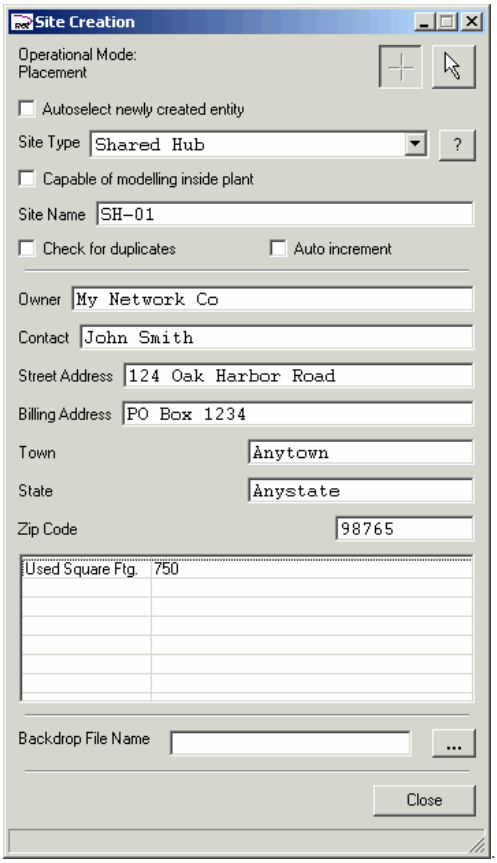


Figure 60 – Building Creation Dialog

Notes:

1. **The grid control showing the custom Used Square Ftg. attribute you added in Step 18 will only become visible when you select Shared Hub in the Facility Type field.**
2. **To update the value of a custom attribute in the grid control, double click on the cell in the right hand column, into which you can then type the value.**

3. Ensure that the "Placement" cross hair is selected (in the depressed position) in the Building Creation Dialog, as shown in Figure 60, above, and also here:

4. Place the new Shared Hub on the map by clicking twice on the location shown below (first click to set input focus back to the drawing window, second click to place the new entity).

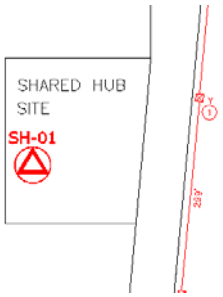


*Figure 61 – Place new Shared Hub **SH-01***

Notes: If you added the USED_SQ_FTG block attribute as described in Step 18, you will also see the value of Used Square Ftg. shown on the map. This is not shown in Figure 61, above.

Step 20: Document References

SPATIAL*net* provides the capability for attaching references to files, programs, websites, URL's, etc. to any object in the system. This can be useful for making all the information associated with an object or job easily accessible from one place.

Note: The SPATIAL*net* database only stores a *reference* to the document, and *not* the document itself. If the filename or path changes, the reference will no longer be valid.

Attaching a Document Reference

To attach a reference to an object:

1. Select the object by clicking on it.
2. Click the **Details of Selected Entity** button on the **General** toolbar.

3. This will display the **Site (OSP) Details** dialog box, shown below.

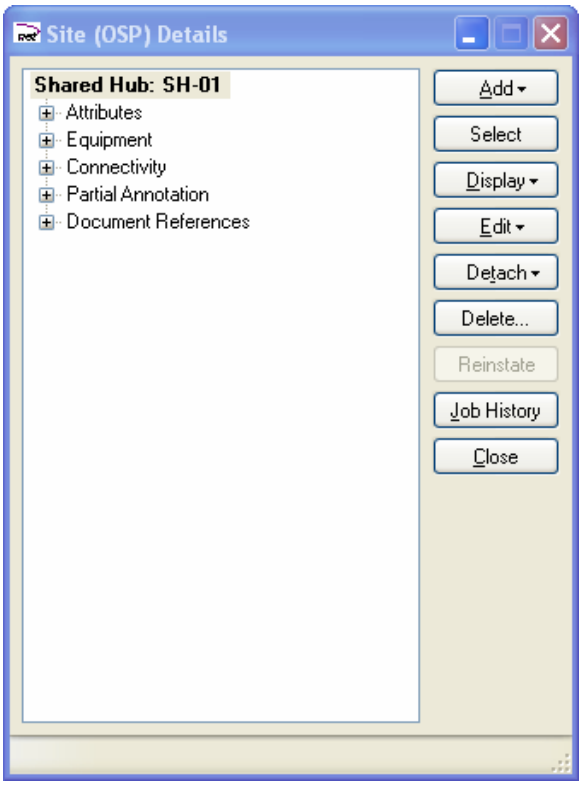


Figure 62 – Site (OSP) Details dialog box

4. Click on the **Add >>** button and then select the **Document...** option from the pop-up.

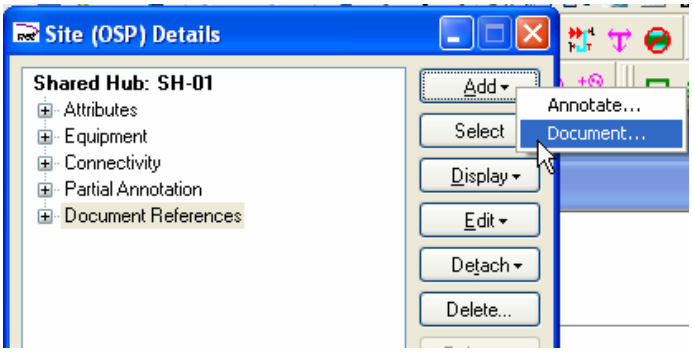


Figure 63 – Site (OSP) Details dialog box

5. This will produce the **Document Reference Creation** dialog box, shown below.

Figure 64 – Document Reference Creation dialog box

6. Enter the document title in the **Document Title** field. (The asterisk indicates that this field is mandatory.)

   Note: The document title can be any text you wish, and does not have to correspond with the document file name.

7. Enter the appropriate document file name in the **Document File Name** Field.

   (Click on the  button to browse for file name.)

8. Click the **Add** button, then click the **Close** button.

9. The document reference should now be visible in the **Site (OSP) Details** dialog box.

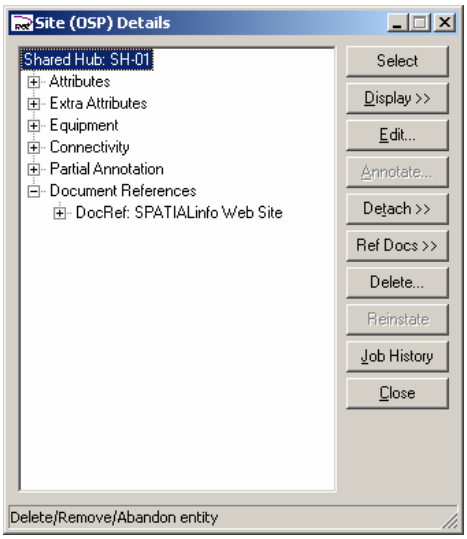


Figure 65 – Document Reference Creation dialog box

Deleting a Document Reference

To delete a reference to an object:

1. Select the object by clicking on it.
2. Click the **Details of Selected Entity** button on the **General** toolbar.

3. This will display the **Site (OSP) Details** dialog box, shown below.

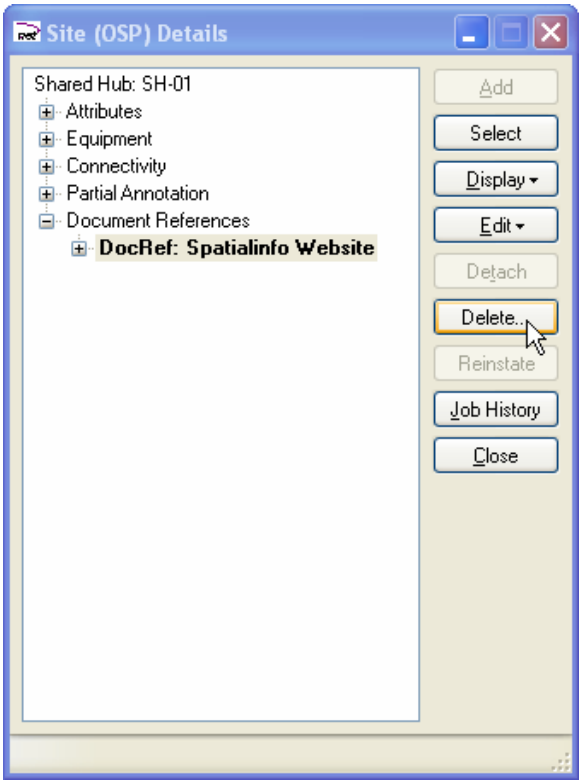


Figure 66 – Site (OSP) Details dialog box

4. Highlight the reference to be deleted, and click on the **Delete** button.

5. This will display the **Entities being deleted** dialog box, shown below.

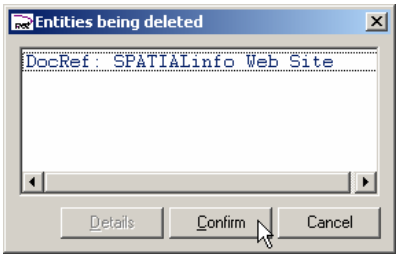


Figure 67 – Entities being deleted dialog box

6. Click the **Confirm** button. The deleted document should now no longer be visible in the **Site (OSP) Details** dialog box.

Chapter 3

Addresses and Service Areas

This chapter reviews the management of service addresses in SPATIAL*net*. Service addresses are typically used to represent residential service addresses for both single and multiple dwelling units, and for small businesses. Larger sites, for example a major office complex or large institution, are usually modeled using SPATIAL*net's* inside plant tools. This is because providing service to these sites normally involves the deployment of devices of various kinds, rather than just the installation of a drop line.

Included in this chapter is an introduction to the use of Service Areas. Service Areas are boundaries (defined in the Landbase Boundaries dictionary) with a special feature, in that they can automatically locate all addresses contained within them, and assign those addresses to the boundary. This is particularly useful for finding all addresses contained with, for example, a Telephone Rate area or an HFC Node boundary.

Step 21: Extend Landbase

Let's now prepare to design a new residential distribution network. Let's assume the homes to be fed are in a new subdivision whose landbase is not yet completely captured in the system. So, our first step is to update the landbase to include the changes made in the subdivision. The area we will be working in is shown below:

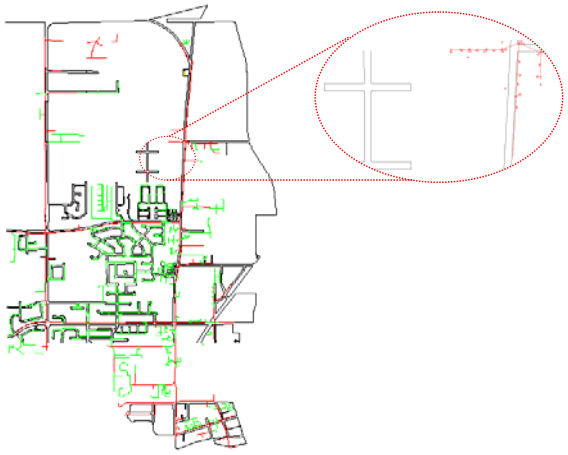


Figure 68 – Location of new area in existing landbase

Our first task is to complete the landbase by joining the partially drawn roadway to the existing road. This can be done using normal AutoCAD operations (not covered in this course), and running the AutoCAD **Save** function when done. The end result should look similar to that shown below:

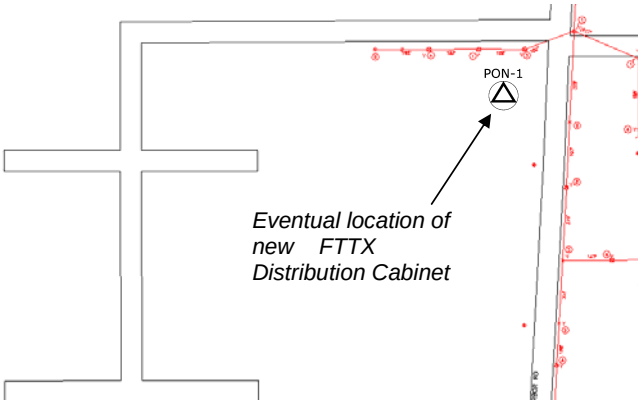


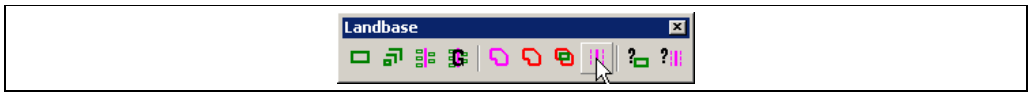
Figure 69 – Completed new road edge, showing location of new FTTX cabinet

Step 22: Add Street Centerlines

Altering the CAD graphics for the road edge, as in Figure 9, only provides a visual indication that the new roads exist. However, since there is no intelligence behind these CAD line segments they cannot be used as a means for searching the database for roads, addresses, etc. To create an intelligent representation of the new roads we must add Street Centerlines to the map.

To add Street Centerlines:

1. Select **SPATIALnet > Add > Landbase > Road** or click the **Add Road** button on the **Landbase** toolbar.



2. This will display the **Road Creation** dialog box. Fill out the box, as shown below.
   Note: Make sure the City field is filled out. Although it is not mandatory, it is needed for later steps.

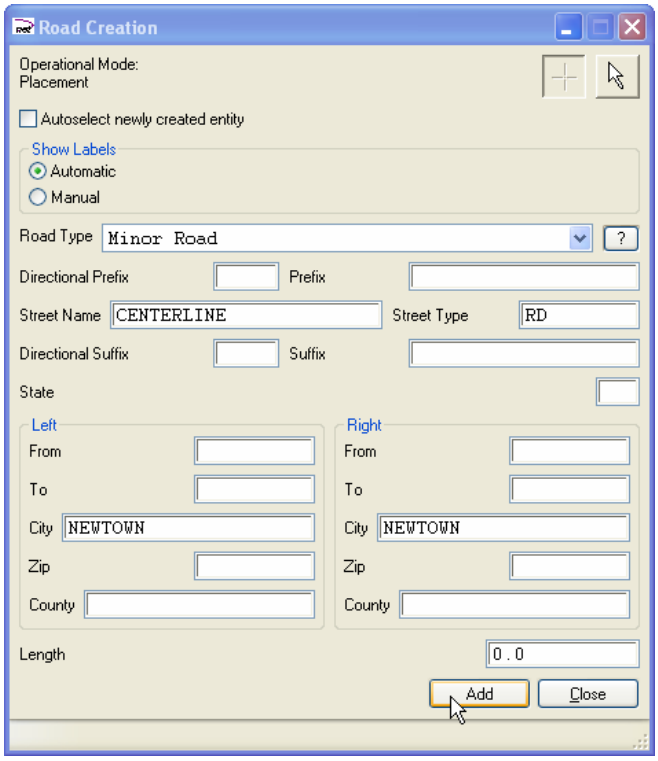


Figure 670 – Road Creation dialog box

3. Click Add to bring up the line capture controls and run a centerline down the middle of the road, as show below.
4. Repeat again to create four centerlines, and name the roads SOUTHLINK CT, NORTHLINK CT, CENTERLINE RD, and NEW RD, as shown below.

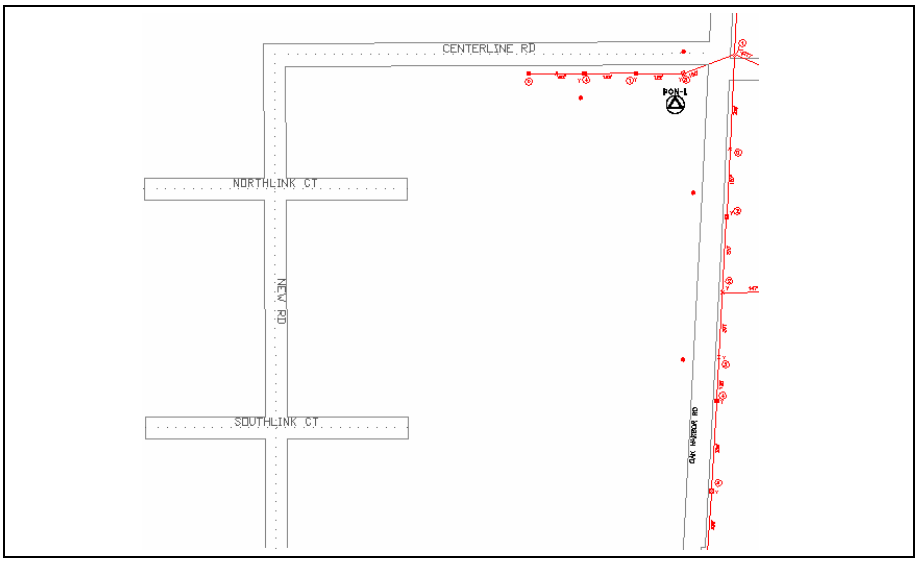


Figure 71 – Creation of centerlines

Step 23: Configure Service Area Boundary

There are many cases where a geographical boundary is used to delimit a particular characteristic of a distribution network. For example, a telephony rate center area denotes the region within which all addresses are assigned a specific rate code. Alternatively, it is common practice to draw a boundary enclosing the geographical area which is served by a single network distribution point, such as a specific Optical Node in an HFC network, or a particular distribution cabinet in an FTTX network.

When such boundaries are used, it is convenient to assign each address which is located within that boundary to the boundary itself. In this way, addresses can be automatically allocated to the correct rate center, distribution area, etc. with no input required from the user.

In SPATIAL*net* FM, this automatic association between addresses and certain boundary types can be made via the use of Service Boundaries. A Service Boundary is a normal geographical boundary, with the added feature that any address placed within that boundary will be automatically associated to that boundary in the database. SPATIAL*net* FM supports up to 3 different types of service boundaries (e.g. so that an address can simultaneously be associated with (say) a telephony rate center, a distribution node boundary, and a dispatch area). The remainder of this step explains how Service Boundaries are configured.

To configure a service boundary:

1. Select **SPATIALnet > Dictionaries > Landbase Definitions > Boundary Definitions…** to display the **Boundary Definitions** dialog box.

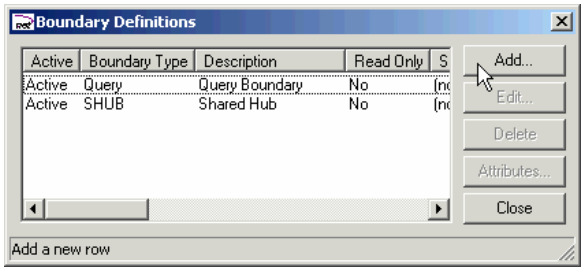


Figure 72 – Add Boundary Definition dialog boxl

2. Click the **Add...** button to display the **Add Boundary Definition** dialog box.
3. Fill out the fields, as shown below, then click **Add**. Note that the **Service Area Type** field should be set to **1**, as shown below. This will define the Rate Center boundary as the first of the three available Service Area types. In turn, this will allow SPATIAL*net* FM to automatically associate the boundary to any addresses located within the area it encloses (we shall see this in practice below).

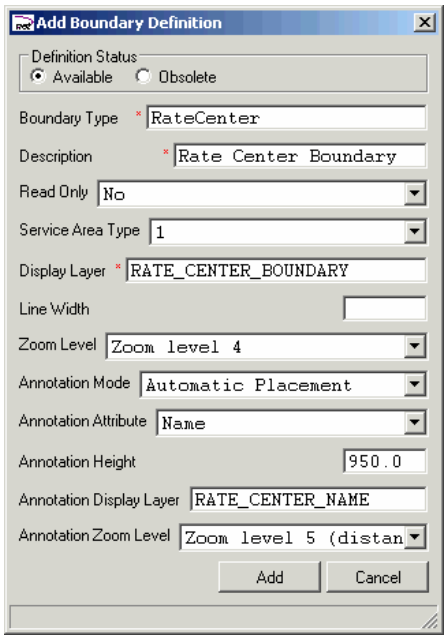


Figure 683 – Add Boundary Definition panel

4. Click the **Close** button.

Step 24: Add Service Area Boundary

Now that we have configured FTTP Distribution areas as a Service Area Boundary type, we will add one to our map. Specifically, we will draw the boundary which encloses the area fed by the FTTP Distribution Cabinet we will place in Step 47. Because FTTP Distribution areas are configured as a Service Area, any addresses included in that area can be automatically associated with the boundary.

The process for adding a service area boundary is identical to adding any other boundary (see Step 7):

1. Select the menu option **SPATIALnet > Add > Landbase > Boundary…** or click the **Add Boundary** button on the **Landbase** toolbar.

2. This will display the **Boundary Creation** dialog box. Fill out the fields, and click the **Add** button, as shown below.

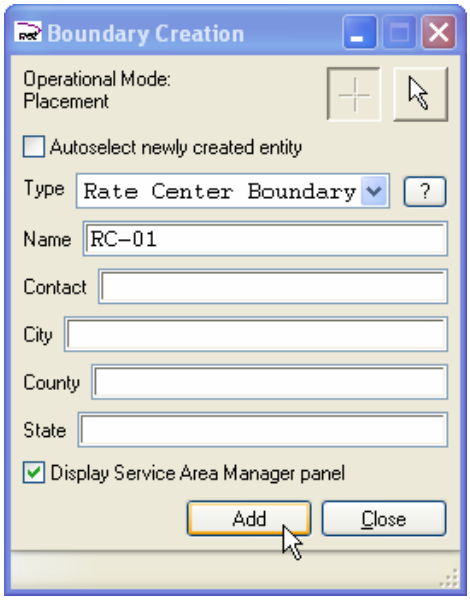


Figure 694 – Boundary Creation dialog box

3. Draw the boundary on the map, similar to that shown below.

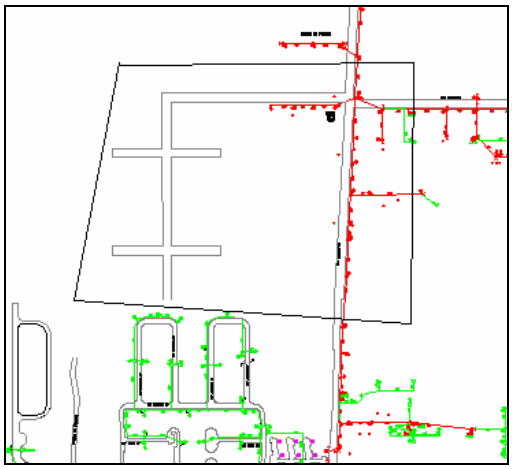


Figure 705 – Boundary Creation on map

Note the **Display Service Area Manager Panel** check box. If checked when we add a service boundary, the service area manager dialog box is displayed, providing options to list or assign addresses or devices within the boundary area (none yet in this area).

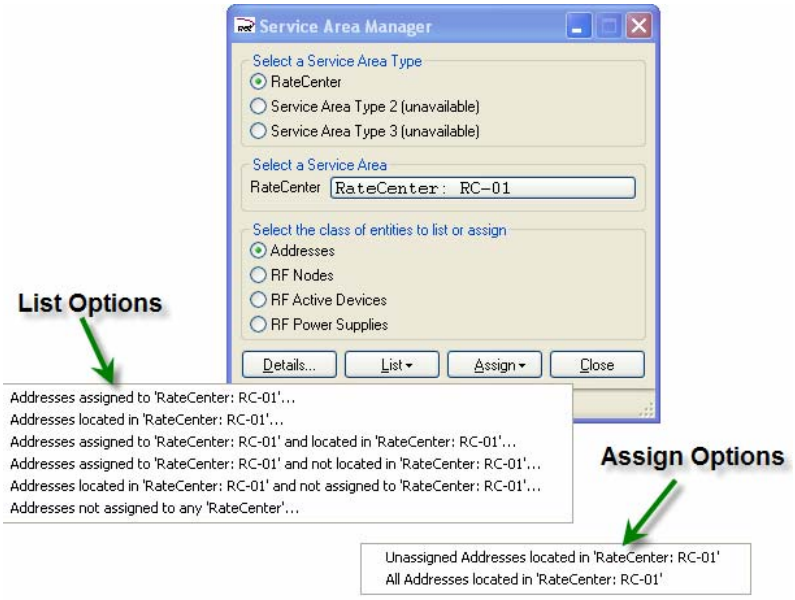


Figure 716 – Service Area Manager panel displaying List options and Assign options available.

Step 25: Adding Buildings and Addresses (Manual Addition)

Having captured the pole and strand network, let's now add the addresses which will be connected to the network. As with all other data types managed by the system, address types are configured in the Address Definitions table, under the **STATIAL*net* > Dictionaries > Landbase Definitions > Building Definitions ...** menu. The process for configuring new addresses is almost identical to that for configuring new site types (see Step 18) so we won't repeat it here. Your database should already have an SDU address type configured in it (if it does not, you can configure it now. The symbol **sdu.dwg** is provided in the library for this purpose).

To add addresses to the system:

1. Run the menu command **SPATIAL*net* > Add > Landbase > Address...**
   or
   Click the **Add Building and Addresses** button on the **Landbase** toolbar

2. A panel similar to the one below will appear. Fill in the values as shown.

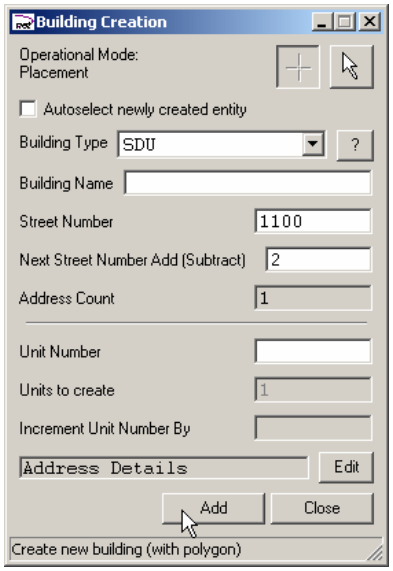


Figure 727 – Building Creation dialog box

3. Click on the AutoCAD map area once to transfer the input focus, and then click once at the location of the first new address.
4. Switch the **Operational Mode** to **Selection** (click on the arrow button, not the cross-hairs shown in Figure 77**Error! Reference source not found.**).

5. Rotate the address to be 90º to the road edge (drag the rotation manipulator onto the road-edge and select **90º to ...** in the pop-up). The system will remember the rotation of this address, and will place all subsequent addresses at this rotation.
6. Change the **Operational Mode** back to **Placement** (click on the cross-hairs in **Error! Reference source not found.**). Then add a sequence of new addresses to the map, just by clicking at the location of each one. Note that the address number will automatically increment by the value specified in the **Next Street Number Add (Subtract)** field (enter a negative number if you wish the street numbers to decrement).

The end result should look similar to that shown below.

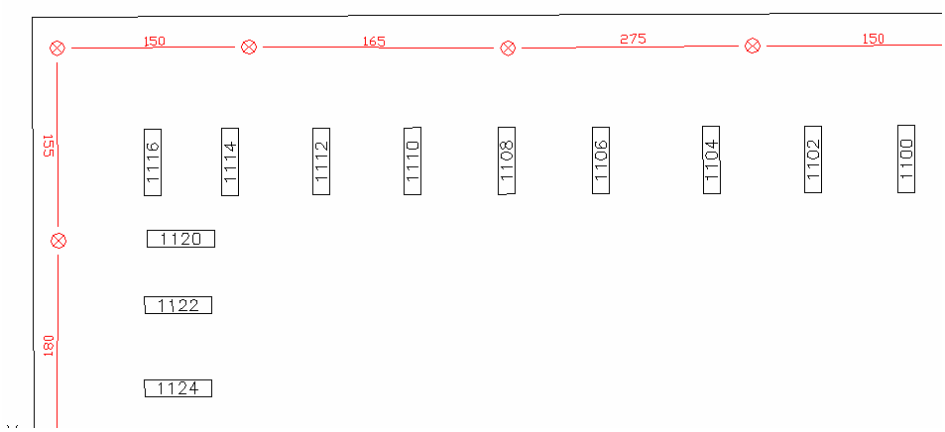


Figure 73 – New addresses entered into map

Note that addresses which were placed inside a Service Area boundary will have their first Service Area link automatically set to the boundary. This can be seen under the **Service Areas** link in the Details Panel for the address, as shown below.

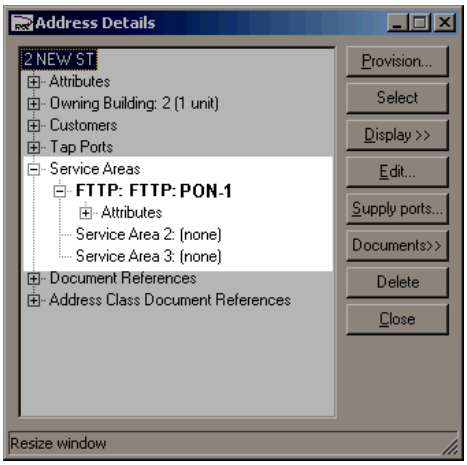


Figure 79 – Service Area "FTTP" automatically set for new address.

Step 26: Add Land Parcels

In addition to modeling service addresses, SPATIAL*net* FM also includes the ability to model land parcels. In many cases, land parcel data along with the associated address information, is obtainable from various landbase suppliers and can be automatically loaded into the system (the loading process is outside the scope of this course). The major advantage this brings is the ability to use parcel information to automatically create service address records via SPATIAL*net* FM's **Migrate Parcel** tool. That tool is the subject of the next step. In this step, we shall learn how to manually add land parcels and capture the information associated with them.

To add land parcels:

1. Select **SPATIALnet > Add > Landbase > Parcel...** or click on the **Add Parcel** button on the **Landbase** toobar.

2. This will display the **Parcel Creation** dialog box, shown below.

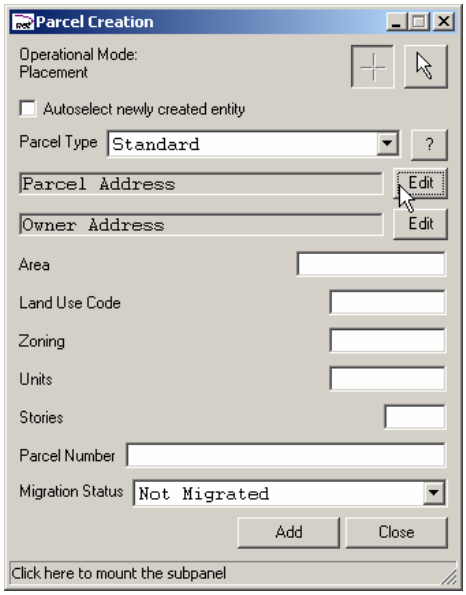


Figure 740 – Parcel Creation dialog box

3. Click on the **Edit** button on the **Parcel Address** field.

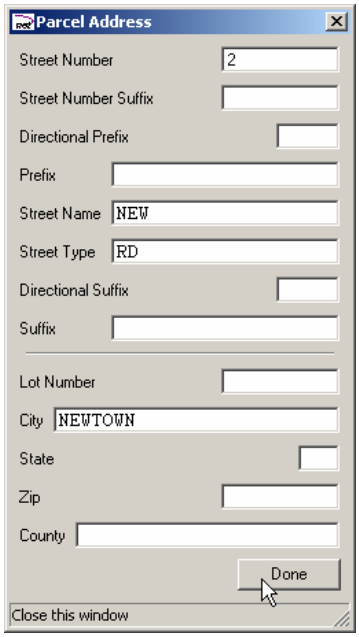


Figure 751 – Parcel Address dialog box

4. Fill out the fields, as shown above, then click on the **Done** button.
5. Draw a land parcel, making sure that the parcel is at the same location as the street name, as illustrated below.
6. Repeat the process to create four land parcels, making sure that you update the **Street Number** field for each parcel.

Note: The parcel numbers will be automatically displayed in the center of each parcel if the Show Parcel Labels box is checked on the View Settings Panel.

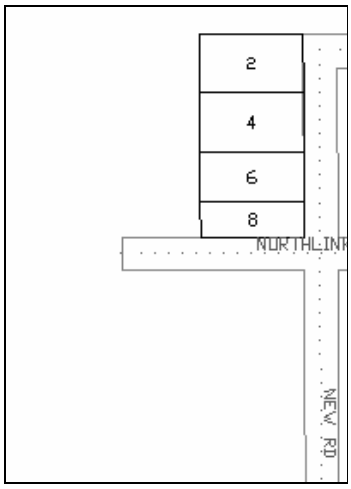


Figure 762 – Land Parcels on New Rd

Step 27: Automatically Create Service Addresses from Land Parcels

The data in the land parcels can be used to automatically create the service addresses using SPATIAL*net's* **Migrate Parcels** tool. Where land parcel data is available, this tool provides a more efficient means for creating service addresses than the manual entry approach used in previously.

To use the **Migrate Parcels** tool:

1. Check that the multi-select mode is enabled, by confirming that the double arrows are displayed on the **Toggle Selection Mode** button on the **General** toolbar, as shown below. If the single arrow is displayed, click on the **Toggle Selection Mode** button to toggle back to the double arrow.

2. Draw a box around the land parcels to select them, as shown below.

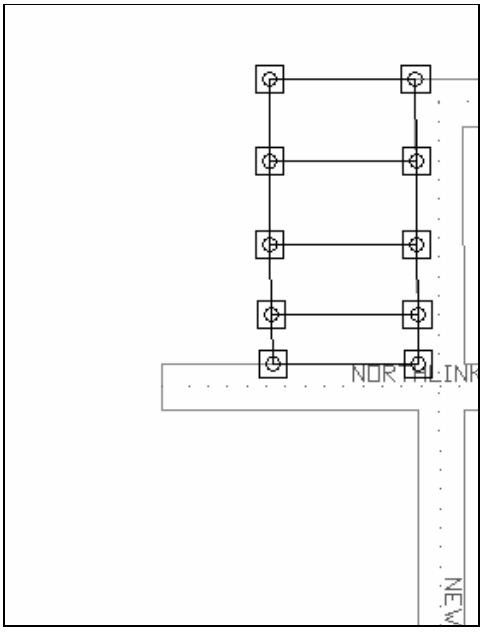


Figure 773 – Selected Land Parcels

3. On the **Landbase** toolbar, click on the **Migrate Parcel** button.

4. If the migration was successful, the **Parcel Migration Results** panel should appear.

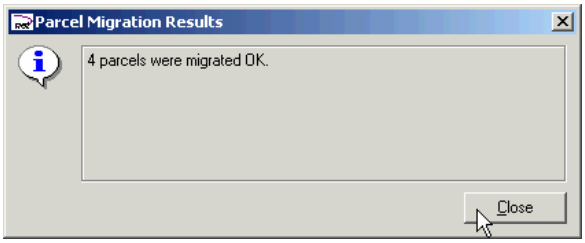


Figure 784 – Parcel Migration Results Panel

5. The service addresses should now be visible on the map.

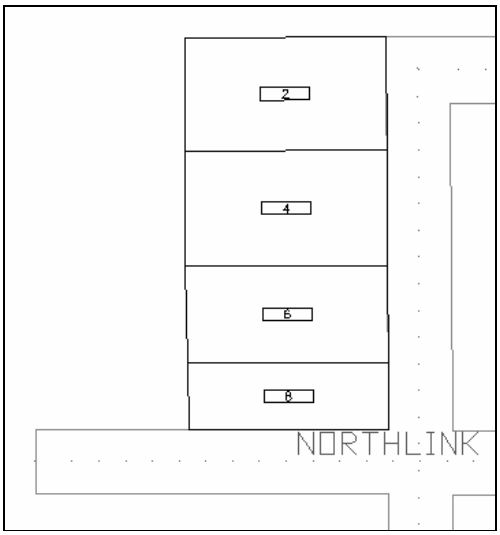


Figure 795 – Service Addresses shown on map

Step 28: Edit Multiple Addresses

SPATIAL*net* provides the ability to edit multiple fields on multiple addresses, making it easy and efficient to correct or update many service address records with a single operation. The multi-address edit panel also includes functions for filtering the specific addresses which are to be edited from the initially selected set.

In our example, we have not as yet, entered any zip codes or a State for any of the addresses we have created either manually, or by using the Migrate Parcel tool. We shall now use the multi-address edit tool to set these values for all of the addresses we have created so far.

There are two methods that can be used to access the **Multiple Address Modification** Panel; the *Select Boundary* method, and the *Select Multiple Addresses* method.

Select Boundary Method:

1. Select a boundary on the map by clicking on it.

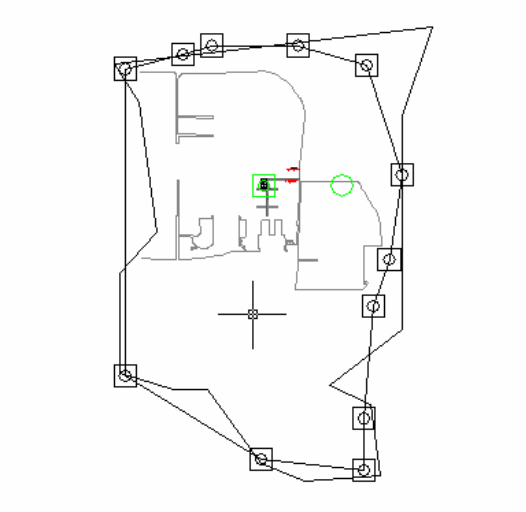


Figure 806 – Selecting a boundary

2. Click on the **Edit Multiple Addresses** button on the **Landbase** toolbar, as shown below.

3. The **Multiple Address Modification** Panel should now be displayed, with all the addresses that are enclosed within the boundary.

OR

Select Multiple Addresses Method

1. Check that the multi-select mode is enabled, by confirming that the double arrows are displayed on the **Toggle Selection Mode** button on the **General** toolbar, as shown below. If the single arrow is displayed, click on the **Toggle Selection Mode** button to toggle back to the double arrow.

2. Draw a box around the addresses to select them, as shown below.

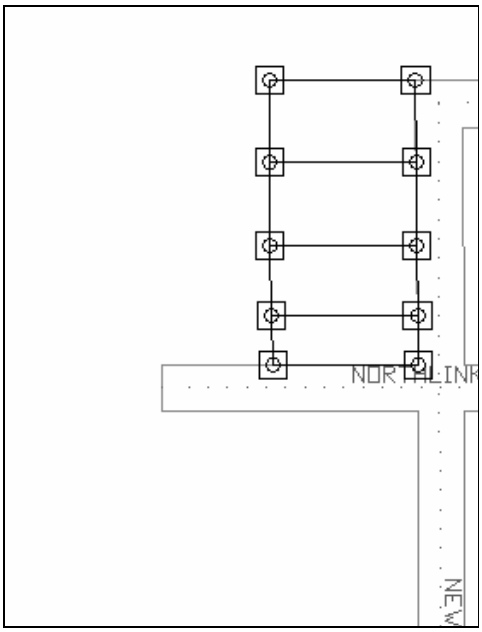


Figure 817 – Selected Land Parcels

3. Click on the **Edit Multiple Addresses** button on the **Landbase** toolbar, as shown below.

4. The **Multiple Address Modification** Panel should now be displayed.

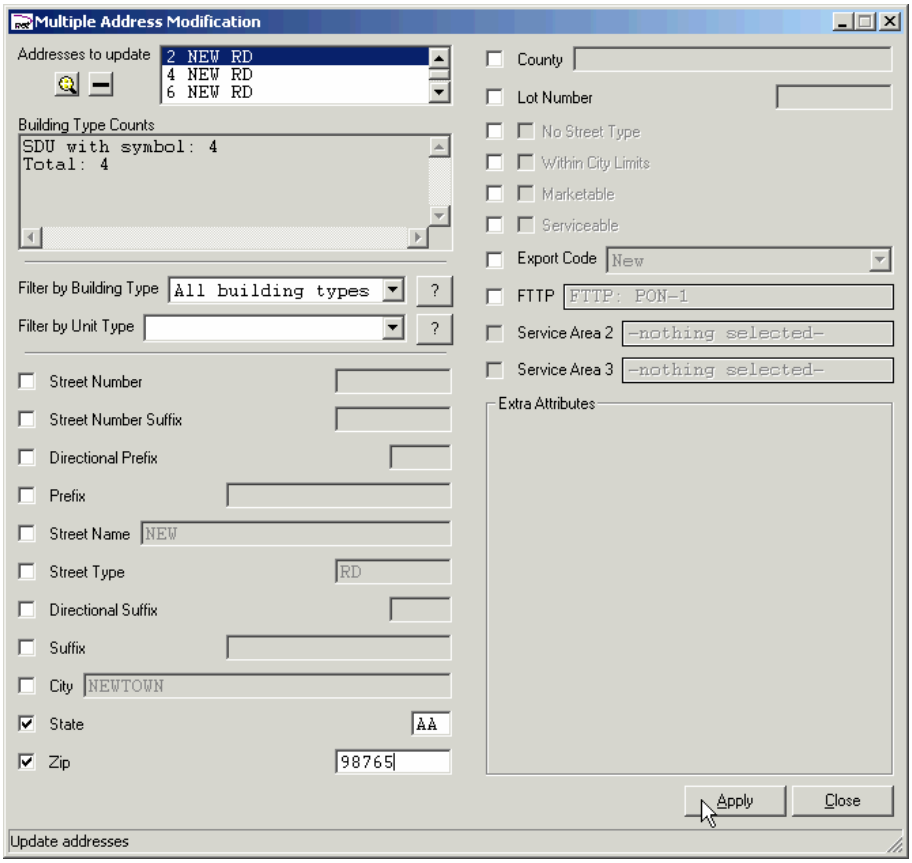
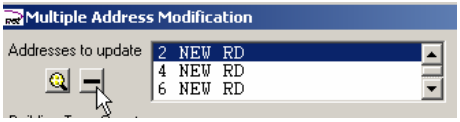


Figure 828 – Multiple Address Modification panel

5. Click on the fields you wish to edit to turn the field from gray to white. In this step we are going to edit the zip code and State. Enter the data, then click **Apply**.

Note: It is possible to remove addresses from the list of those to be updated by highlighting the address in the list window and then clicking on the – button, as shown below. To add an address to the list, simply click on it in the map.



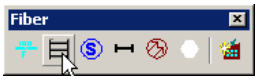
Chapter 4

Outside Plant Fiber

Step 29: Add Term Panel to Site

Now that we have created the site, let us place a term panel within it, ready for us to connect fiber cables.

1. Select the new Shared Hub site by clicking on its symbol on the map. When it is selected, the SPATIAL*net* Move and Rotate manipulators shown in Figure 1 will be displayed.
2. From the menu, run the command **Fiber > Add > Term Panel...,** or click the **Add Term Panel** button on the **Fiber** toolbar:



3. Fill in the fields of the **Fiber Term Panel Creation** Dialog Box with the values shown below:

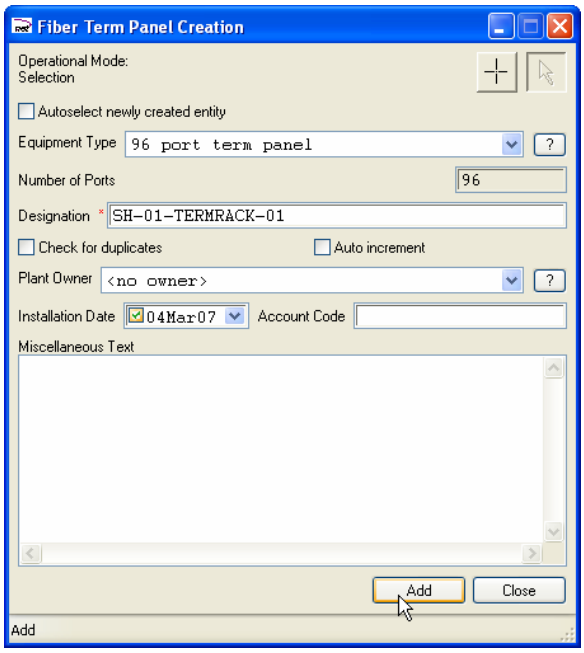


Figure 89 – Fiber Term Panel Creation Dialog Box

4. Ensure that the "Selection" arrow is selected in the **Fiber Term Panel Creation** Dialog Box, as shown in Figure 89, above, and also here:

5. Click the **Add** button to place the new term panel in the selected Shared Hub.
6. The click yes on the confirmation panel to complete the operation:

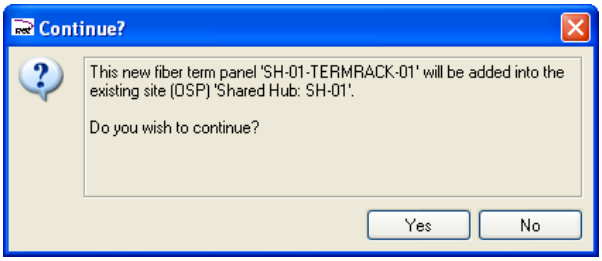


Figure 90 – Term Panel Confirmation Dialog Box

7. To verify that the term panel was placed successfully, display the details of the shared hub by clicking on the **Details** button on the **General** toolbar (ensure the Shared Hub is selected first):

8. Examine the Equipment branch of the browser window as shown below to ensure that the Term Panel was placed within the Shared hub.

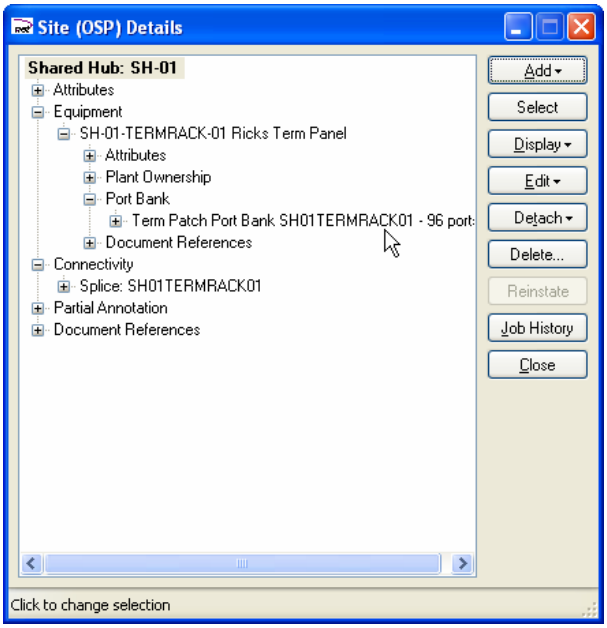


Figure 83 – Site (OSP) Details showing Term Rack placed in the Shared Hub.

Step 30: Adding More Equipment to a Site – Understanding Splice Locations.

In the previous step we have just added a fiber termination panel. However, many facilities have more than one term panel, and it's important to understand how to place these additional term panels in SPATIAL*net* so that fiber cables can be spliced to them correctly.

In this step, we are going to add additional term panels to our Shared Hub and learn about the important concept of *splice locations* within a site. Understanding splice locations is important when using SPATIAL*net* since, as we shall see shortly, splicing locations are the key to being able to connect fiber optic equipment together (that is, to splice fibers to other fibers or to equipment ports).

Examining Figure 83 from the previous step, we can see that when we placed a new term panel into our site, the system created *two* new entries in the Site Details panel—one under the **Equipment** branch and one under the **Connectivity** branch. These are highlighted in Figure 84 below.

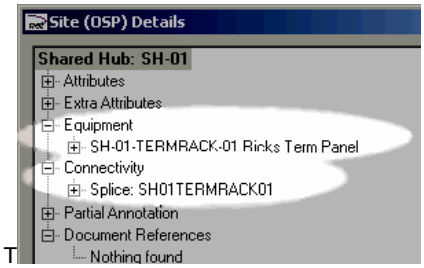


Figure 84 – Site (OSP) Details highlighting term panel and splice location

Each **Splice** entry under **Connectivity** represents a *distinct splicing location* within a site. For example, a building may have distinct fiber entry locations on its north wall and on its south wall. In such a situation, only fiber cables located at a single entry point could be spliced together, or connected to termination ports at that entry point. A fiber cable entering through the north wall could not, for example, be spliced to a termination panel at the south wall.

When you placed the new term panel in Step 29, SPATIAL*net* automatically created a splice location to contain the term panel. If we expand the **Site** branch under **Connectivity** in Figure 84, we see that the ports associated with the term panel are located there:

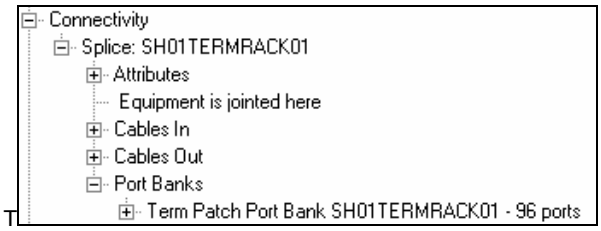


Figure 85 – Showing Term Ports located at a Splice Location

Let's now add another term panel and observe SPATIAL*net's* behavior.

Adding Additional Term Panels to a Site – "Select Site" Method

To add another term panel to the Shared Hub site we created in Step 19, we follow the same procedure as in Step 29:

1. Select the Shared Hub Site either by:
   (a) Clicking on it in the map, or
   (b) Highlighting it in the details panel and clicking the **Select** button, as shown below:

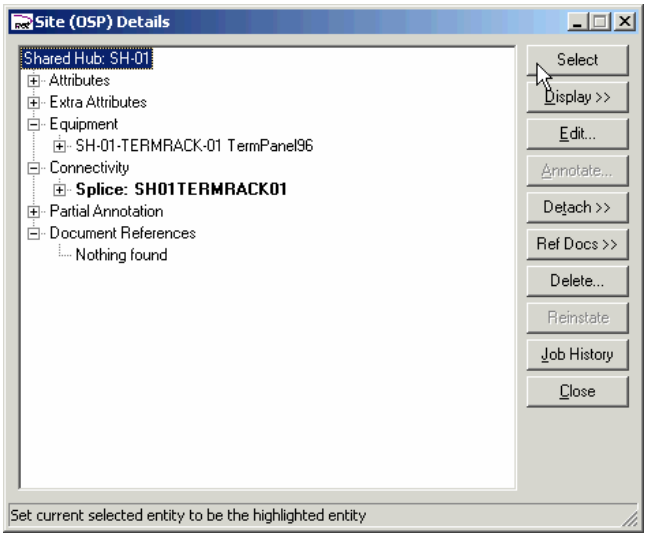


Figure 86 – Site (OSP) Details panel, with Shared Hub selected

2. Follow steps 2–8 on pages 88–90 to add the new term panel.

Note: Use a different designation when completing the Fiber Term Panel Creation panel to make it easy to tell the two term panels apart.

When this operation is complete, the details of the site will appear as shown below:

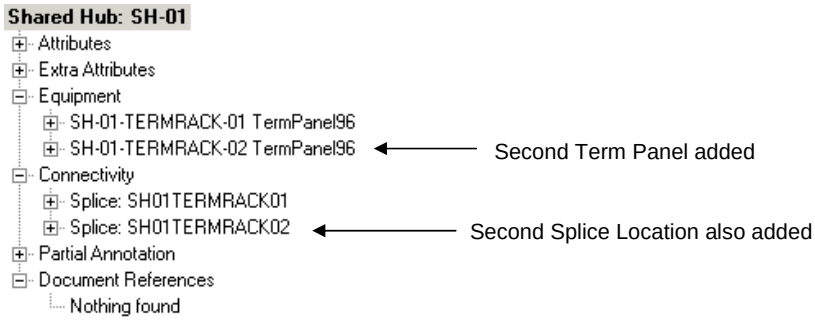


Figure 87 – Second Term Panel and Splice Location added to a Site

Note that not only was a second Term Panel added to the site under the **Equipment** branch, but that a second Splice Location was also added under the **Connectivity** branch. Also, observe that the new Term Panel was added to this second location (test this by expanding the second **Splice** branch). This means the two term panels were placed in separate locations, and therefore cannot be spliced to a single cable.

In some cases, this may be what you wish to do. But in many cases, you wish to add a second Term Panel to an existing location so that fibers from a single cable can be spliced across more than one panel. We shall learn how to do this in the "Select Splice" method, below.

Adding Additional Term Panels to a Site – "Select Splice" Method

If you have a *Site* selected (in the above example, it is **Shared Hub: SH-01**) when you are adding equipment such as a Term Panel, SPATIAL*net* will always create a new, *distinct* splice location for each new piece of equipment. A consequence of this configuration is that fibers from a single cable can *only be spliced to a single term panel*. However, there will often be cases where you wish to place more than one Term Panel at a single location, so that the fibers from a single cable can be spliced to ports on any panel at that location.

To place *adjacent* term panels at a single Splice location:

1. Select the Splice location by highlighting it in the details panel and clicking the **Select** button, as shown below. (Compare panel below, with panel in Figure 86.)

   Note: Merely highlighting the splice location is not enough to select it. To properly select it, you must click on the Select button.

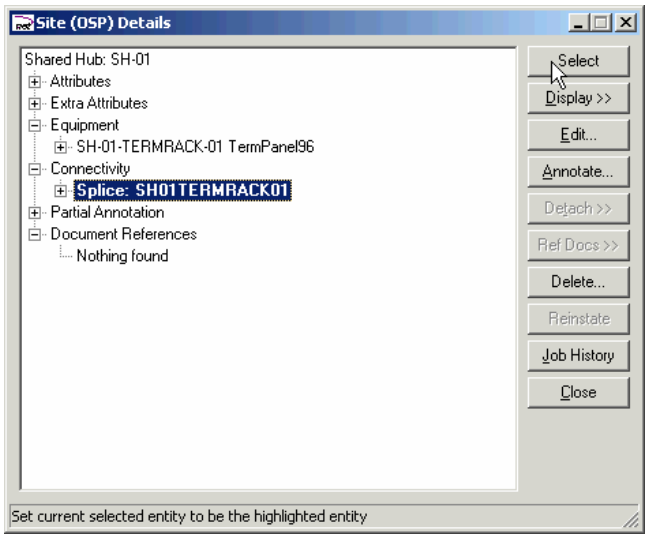


Figure 88 – Site (OSP) Details panel, with splice selected into which a new term panel will be placed.

   Note: It is only possible to select the splice location by selecting it in the Site (OSP) Details panel. It is not possible to select it by clicking on the map.

2. Follow steps 2–6 on pages 88–90 to add the new term panel.

3. Examine the Equipment branch of the browser window, as shown below, to ensure that the Term Panel was placed within the Shared hub. If done correctly there should now be two term panels, but only be one splice entry.

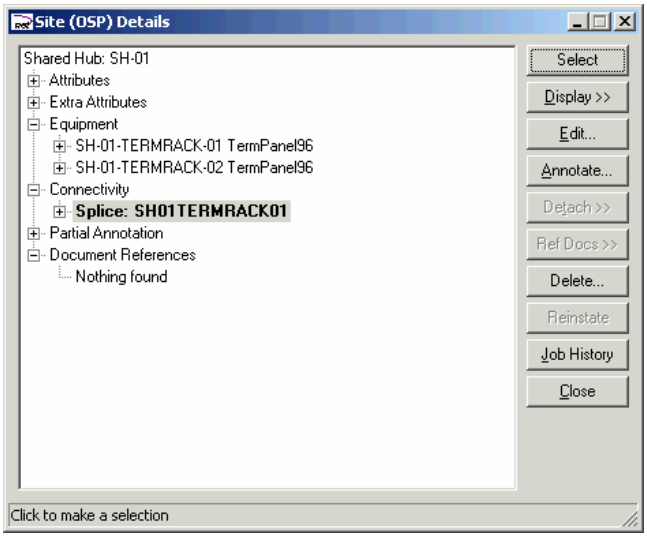


Figure 89 – Site (OSP) Details panel, with Splice Location selected

4. Finally, examine the contents of the Splice Location by expanding the **Splice** branch under the **Connectivity** heading in Figure 89. You should find the port banks for the two term panels located there, as shown below.

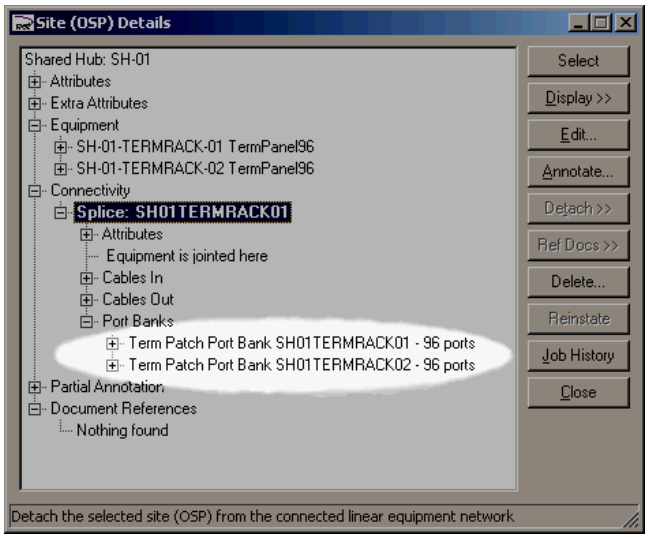


Figure 90 – Both Term Panels located at a single splice location.

Step 31: Moving Equipment Between Splice Locations.

The above steps discussed how to place a Term Panel into the correct splice location within a Site. However, from time to time you may need to move equipment from one splice location to another. This is achieved by a simple drag and drop operation in the **Site (OSP) Details** panel, as we shall now see.

Let's repeat the earlier steps for adding an additional panel. This time, however, after we have added the panel, we will move it to a different splice location from the one it was created in.

1. Add another Term Panel to the *building* (i.e., **Shared Hub: SH-01**) as in Step 29. The new term panel should be visible in the **Site (OSP) Details** panel in its own splice location, as shown below.

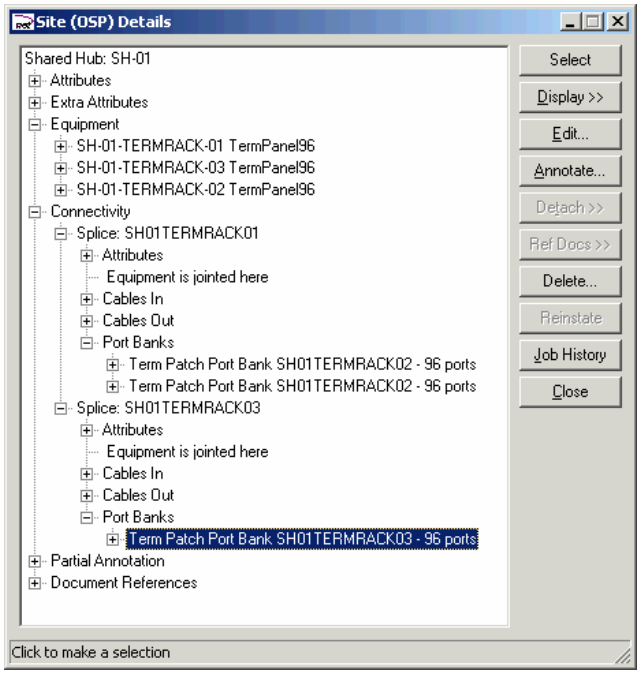


Figure 91 – Additional Term Panel added to Site (OSP) Details panel.

2. To move the Term Panel to a new Splice location simply click on the Term Panel and drag it over the desired splice location, as shown below.

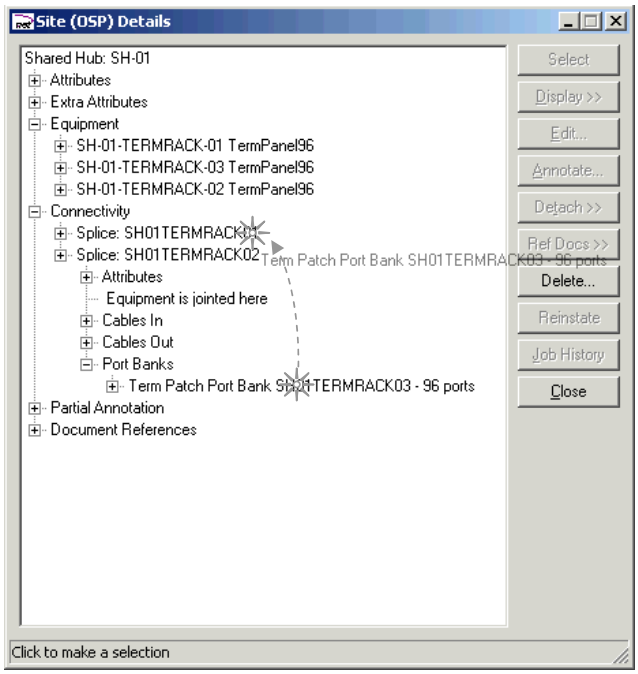


Figure 92 – Drag and drop term panel from one splice location to another.

3. When the Term Panel is over the desired splice location, click on the pop-up, as shown below.

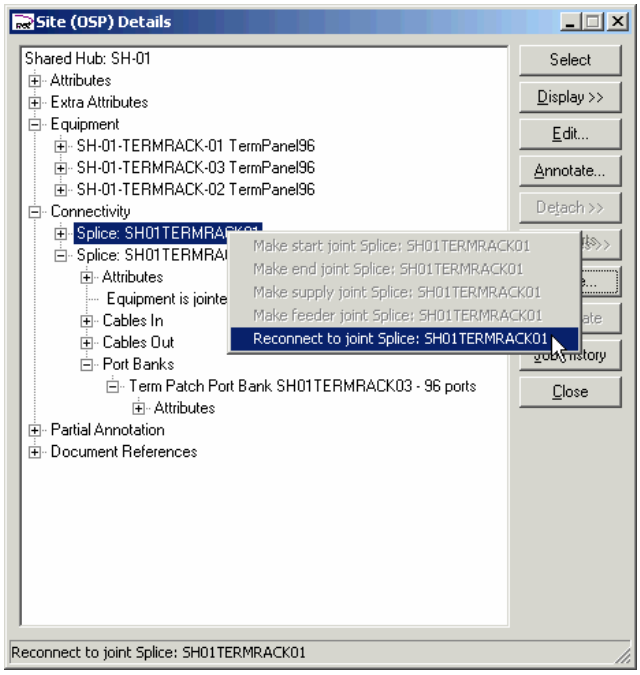


Figure 93 – Pop-up in Site (OSP) Details panel.

Step 32: Configure New HFC Node Type and Usage Codes

As with any type of record in SPATIAL*net* , before we can add a new instance of an HFC node, we have to make sure that the type of node we wish to add is configured in the Node Definitions dictionary. To configure a node we must:

- Specify an identifying name for the type of node, and
- Specify the number and types of fiber ports on this type of node (port types are defined by their "Usage Codes"—more on this below).

Let's suppose we wish to add a Generic type of node with two fiber receivers and one transmitter (for the return path signal). Before we actually go through the steps to configure the node type, let's first look at how we will tell SPATIAL*net* what each of its ports will be used for, and what kind of signal they will be using.

SPATIAL*net* FM provides a great deal of flexibility for tracking the various uses to which fibers in the network might be put. As with all other record types, this is done via a user-defined dictionary of "Usage Codes". Since our network will be using fibers for transmitting forward and return signals to an HFC node, we should define these usage codes in the system's dictionary. Let's do this now.

1. From the menu, run the command **SPATIALnet > Dictionaries > Other Definitions > Usage Code Definitions.** A dialog box will be displayed listing all of the usage codes currently defined.
2. Click on the **Add** button and fill in the field values as shown in the left dialog box below (for the forward path), then close this by clicking on its **Add** button. Repeat this step but this time, fill in the values (for the return signal) shown in the right dialog box in Figure 94, below.

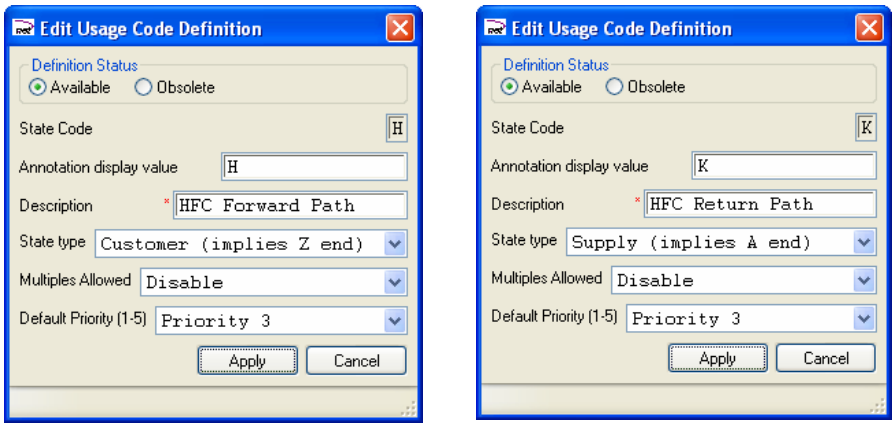


Figure 94 – New Usage Code Entry Forms

3. Close the **Usage Code Definitions** panel.

Note: You will only need to run items 1 to 3 above when you wish to add a new usage type that you wish to record. You will not need to do this each time you configure a new piece of equipment that makes use of a usage code.

Now that we have the usage code defined, we can configure the node type itself. To do this:

4. From the menu, run the command **SPATIALnet > Dictionaries > Fiber Definitions > Node Receiver Definitions.** A dialog box will be displayed listing all of the node types currently defined.
5. Click on the **Add** button and fill in the field values as shown in the dialog box below:

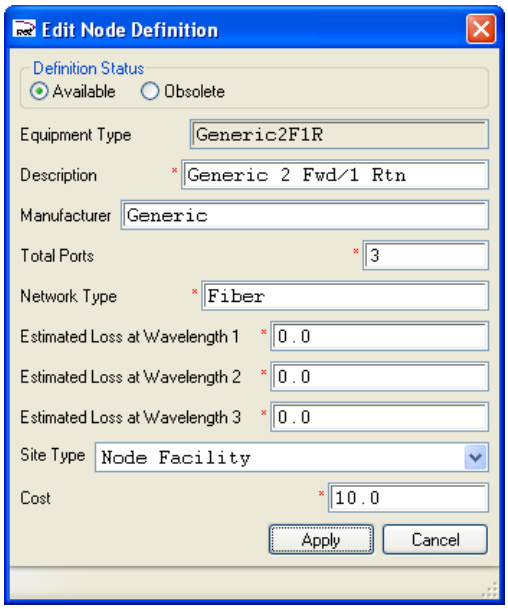


Figure 95 – Add Node Definition Dialog Box

6. Click on the **Add** button shown in Figure 95 to create the new node type. It should now be listed in the **Node Definitions** list.

To complete the definition of this node type, we now add and configure its fiber ports.

7. Ensure the **GenericNode2F1R** Node type is selected in the **Node Definitions** list.
8. Click on the **Ports** button. Since there are no ports defined for this node type, the **Ports** list will initially be empty.
9. Click the **Add** button on the **Ports** list panel. This will allow you to specify the details of each port you wish to add to the node type. We will add two ports of type **HFC Forward Path**, and one port of type **HFC Return Path**., as shown in Figure 96, below.
10. Fill in the fields of the **Add Port Definition** panels for the three ports, as shown below, and click the **Add** button when done for each port.

(Note that the panel for defining Port 1 is not shown, but its values can be seen in the list panel behind the two Add Port Definition panels visible for ports 2 and 3).

11. Close each of the panels shown in Figure 96, below when complete.

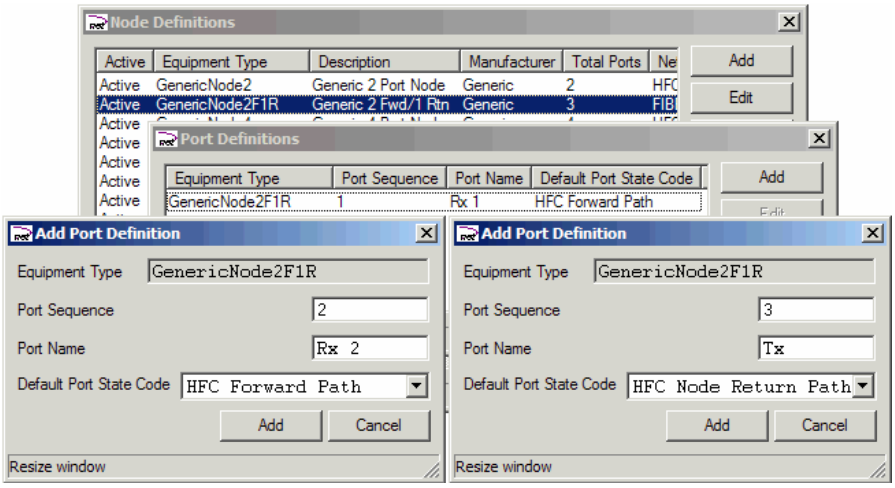


Figure 96 – Adding ports to a node type.

This completes configuration of the new node type, its ports, and the new usage codes we are adding to the network.

Step 33: Add New HFC Node

We will now add a new HFC node, using a process very similar to adding a site described in Step 18.

To add the node:

1. From the menu, run the command **Fiber > Add > Node Receiver...,** or click the **Add Node RX** button on the **Fiber** toolbar:

2. Fill in the fields of the **Optical Node Creation** Dialog Box with the values shown below:

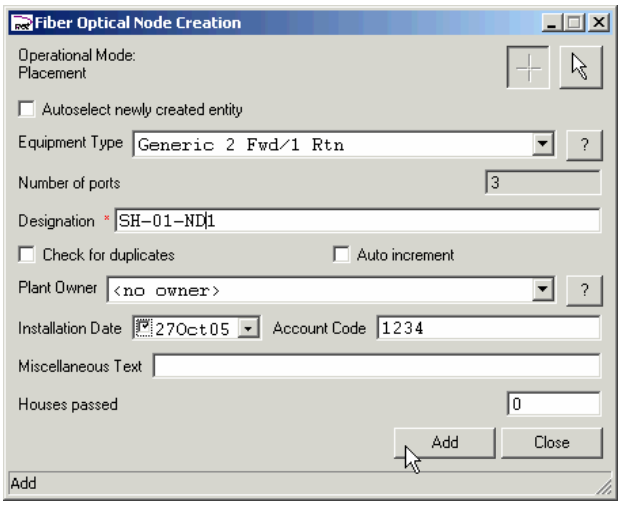


Figure 97 – HFC Optical Node Creation Dialog Box

3. Ensure that the "Placement" cross hair is selected in the **Optical Node Creation** Dialog Box, as shown in Figure 97, above, and also here:

4. Create the new HFC Optical Node at the specified location on the map by clicking on the location shown below.

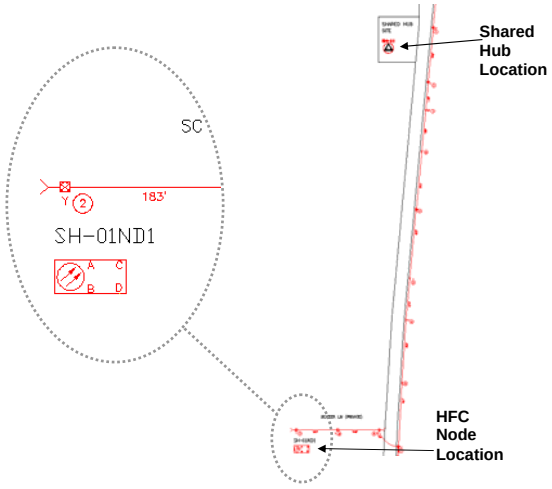


Figure 98 – Place new HFC Optical Node

Step 34: Add New Fiber/Buffer/Ribbon Configuration

In the next step, we shall define a new cable type, but before we do, we shall review a key configuration feature of fiber optic cables: the arrangement of fibers into buffers, ribbons, etc. In SPATIAL*net*, these are referred to as **Fiber Color Scheme Definitions**. To open the Fiber Color Scheme Definitions list:

1. Select the menu option **SPATIALnet > Dictionaries > Fiber Definitions > Fiber Color Scheme Definitions...**
2. A panel similar to that shown below is displayed.

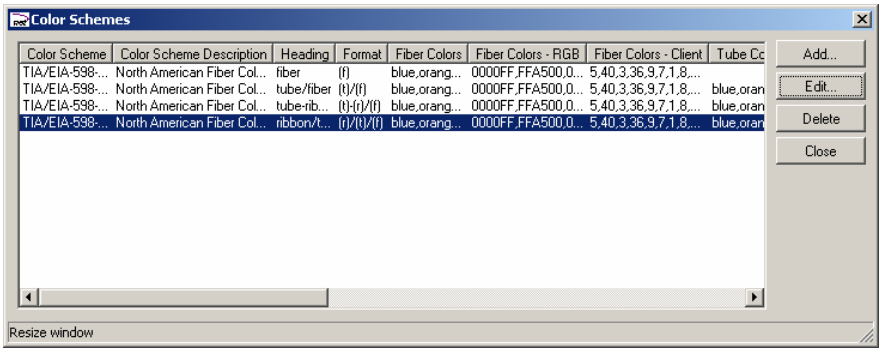


Figure 99 – Fiber Color Scheme Definitions Panel

3. Select one of the existing entries in the list and click the **Edit** button. A panel similar to that shown is then displayed.

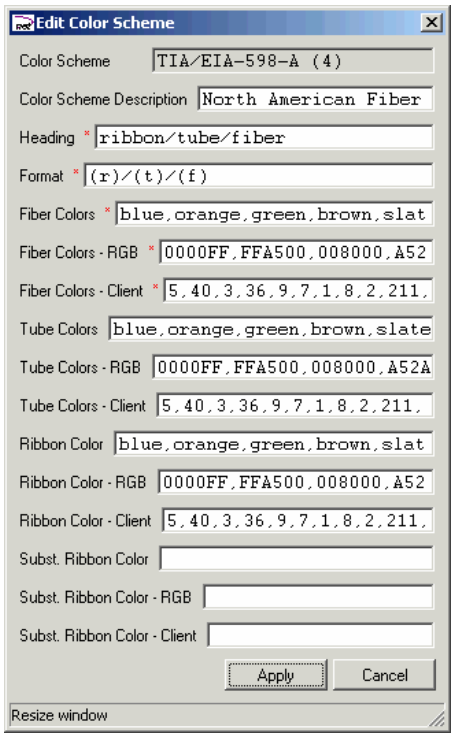


Figure 100 – Fiber Color Scheme Definitions Panel

The above panel configures information about the sequence of fiber colors, buffer colors and ribbon colors that are associated with the particular color scheme. In addition to recording the actual configuration to be used in the cable, the panel also has fields for configuring how these colors will be displayed in both graphical and textual representations of the fiber (for textual displays, a 6 character hexadecimal (base-16) number is used to represent the RGB-value of the display color. For graphical displays, the color index to be used by AutoCAD to draw the fiber is shown.

4. Click on the **Apply** button to commit any changes made to the fiber color scheme configuration to the database.

Step 35: Add New Fiber Cable Type

We have completed placing our new facility, and an HFC Node that it will feed. We are now ready to connect the two by creating a new fiber cable. As with our “Shared Hub” site type in Step 18, we shall assume that the type of cable we want to place is not currently in the Cable Type Dictionary, and that we need to create it. The process is similar to adding an entry to the other Type dictionaries, but cable types require an additional step to specify their linear symbology, as we shall see below.

To add the basic details of the new cable type:

1. From the menu, select **SPATIALnet > Dictionaries > Fiber Definitions > Cable Definitions...**. A dialog box will be displayed, listing all of the cable types currently defined.

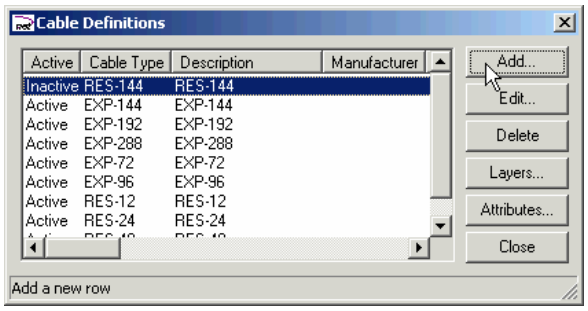


Figure 101 – Cable Definitions dialog box

2. Click on the **Add** button and fill in the field values as shown in the dialog box below.

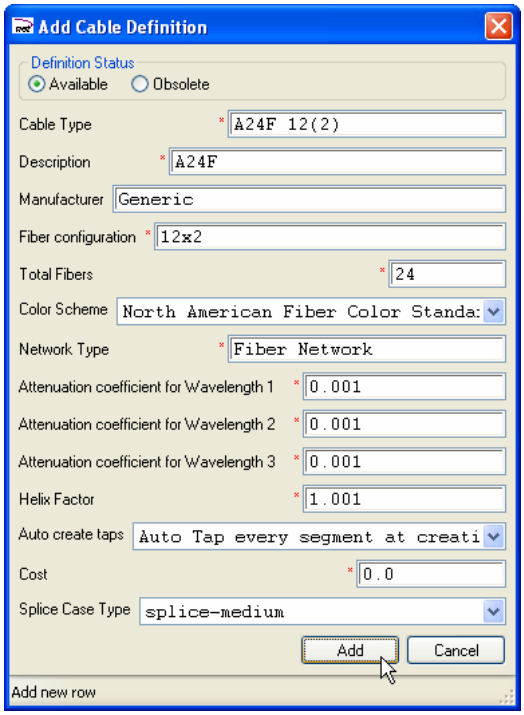


Figure 102 – New Cable Type Entry Form

3. Click on the **Add** button shown in Figure 102 to create the new CableType. It should now be listed in the **Cable Definitions** list.

Hint: The Description field is what is displayed when you choose the cable type for a new cable. Therefore, you should ensure that you include necessary information to identify the cable type, such as number of fibers, buffer organization, etc.

Because the symbology of cables can vary depending on the cable's construction status and build environment (Aerial, Underground, etc.), an additional step is required to define how the cable will appear in each of these situations. The display of a single cable type in these different situations is controlled by having the system render it into a different CAD layer in each case. You can then set the display properties of that layer (color, linetype, lineweight, etc.) as you wish in the template DWT file.

4. From the menu, select **SPATIALnet > Dictionaries > Fiber Definitions > Cable Definitions...**,then click on the **Layers...** button.

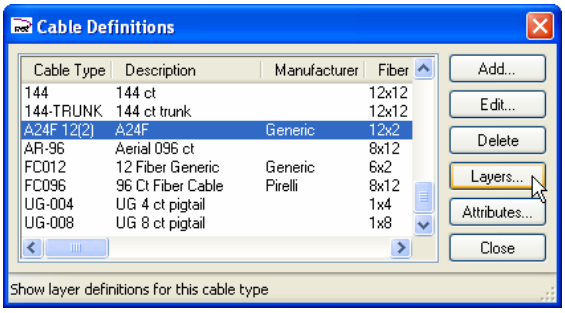


Figure 103 – Cable Definitions dialog box

5. A dialog box will be displayed, listing all of the assignments of cable types to CAD layers that are currently defined.

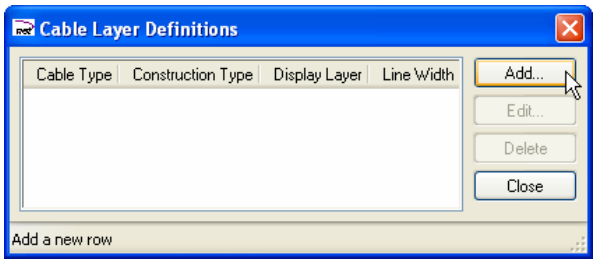


Figure 104 – Cable Layer Definitions dialog box

6. Click on the **Add** button and fill in the field values as shown in the left dialog box below, then close this by clicking on its **Add** button. Repeat this step but this time, fill in the values shown in the right dialog box in Figure 105, below.

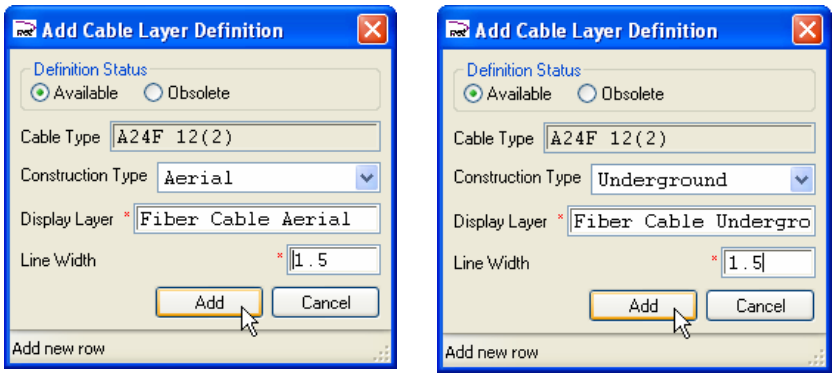


Figure 105 – New Cable Layer Entry Form

Step 36: Connect Site and Node with a New Fiber Cable

In this step, we connect the Node we placed in Step 33 to the Shared Hub we placed in Step 18 by creating a new fiber cable. Cable placement is one of the most sophisticated functions in SPATIAL*net*, allowing automated insertion of in-line equipment, automated creation of terminating equipment, and many other features. We will not cover these more advanced operations in this exercise.

To create the new cable, and connect it to both the Shared Hub and the HFC Node created in the earlier steps:

1. Zoom the AutoCAD view so that it is roughly centered around the Shared Hub, similar to the layout shown in Figure 61, above.

   Note: Ensure that the HFC Node we placed in Step 33 is also in the current window. If it is not, zoom out to an extent which includes both the Shared Hub and the Node, similar to Figure 98, and click the View Refresh button on the SPATIAL*net* View toolbar: 

2. Select the Shared Hub in the Map window by clicking on it. This will be the starting point for our new cable.
3. Start the Add Cable process by selecting **Fiber > Add > Cable** from the menu. Equivalently, click the Add **Add Cable** button on the SPATIAL*net* **Fiber** toolbar.

4. Select the fiber cable type **A24F fiber cable** that we created in Step 35 from the pulldown list, and fill in the other fields as shown below.

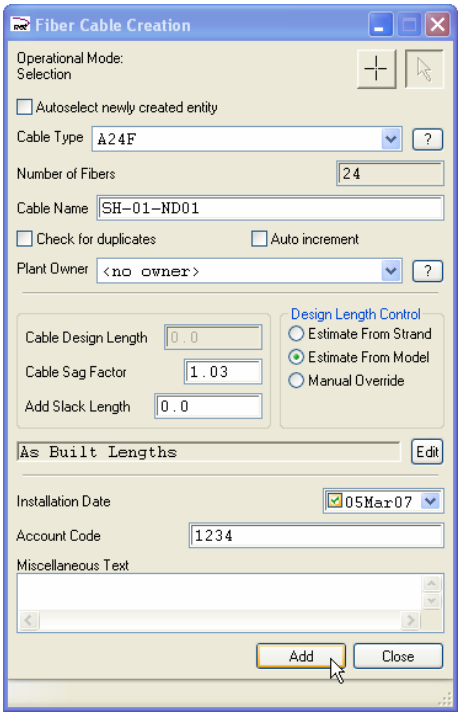


Figure 106 – Fiber Cable Creation Dialog box

5. Ensure that the "Selection" arrow is selected in the **Fiber Cable Creation** Dialog, as in Figure 106, above, and also shown here: 

6. When the fields in the dialog box have been filled in, click the **Add** button. Highlight the splice to connect to (SH01TERMRACK01) and press select. This will cause the **Line Capture Control** panel to become visible, and the cable line segment capture process to start.

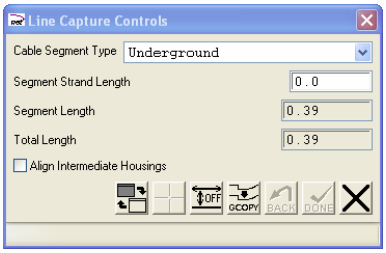


Figure 107 – Line Capture Controls Dialog box

In our hypothetical example, we will run an underground section of cable to a pole near the Shared Hub Site, and then run aerial cable to the node. To do this:

7. Ensure the **Cable Segment Type** in the **Line Capture Control** panel is set to **Underground**.
8. Right-click at a position near a pole as shown in the example below and select **Cable Riser** from the pop-up menu. This will automatically place a Cable Riser at this location when the cable is created, and transition the segment from Underground to Aerial (or vice-versa as appropriate).

Note: If a trench or conduit had already been included in the drawing, we could read its length automatically (as we shall do for the aerial spans, below). However, in this case, we shall use the system's measured length of the line segment to determine the length of the underground span.

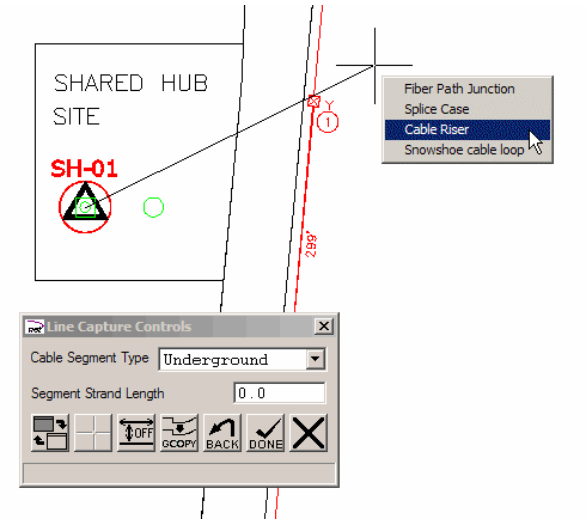


Figure 108 – Location of first line segment for new cable

9. Clicking will cause the **Capture loop attributes** dialog box to pop up (see 117, below). This is because the system is asking if you wish to add a storage loop

at the same location as the riser. If you do, enter the length of cable you wish to store into the loop **Cable Loop Length** field (if you do not, you can leave this field set to 0). If you also wish to record the as-built footage readings at the start and end of this segment, you may enter these into the **Sheath Reading at Segment Start** and **Sheath Reading at Segment End** fields. When you have finished entering data into the **Capture loop attributes** dialog, click **Continue**.

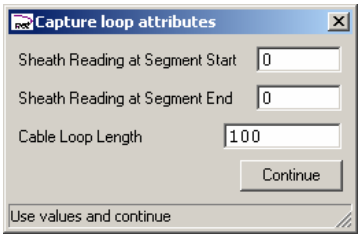


Figure 109 – Capture loop attributes Dialog box

10. We will now run the aerial spans of the fiber cable. Change the Cable Segment Type to Aerial in the Line Capture Control panel. When this is done, click at the position shown below to locate the next vertex of the cable.

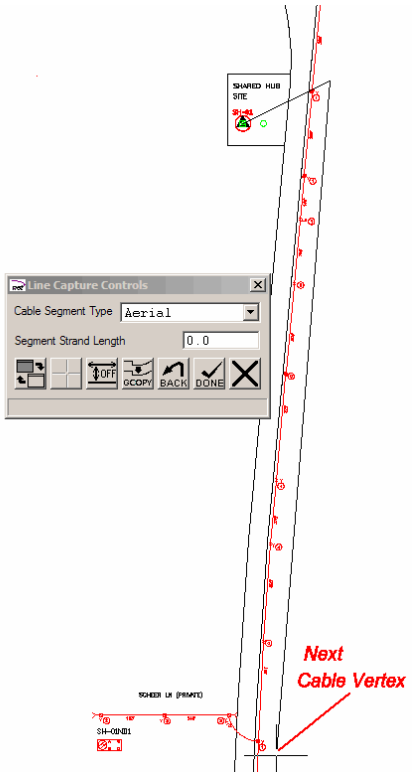


Figure 110 – Location of next line segment for new cable

11. Notice that the next strand segment leading to the pole is a slack span, represented by a curved arc. SPATIAL*net* can automatically trace this arc at a specified offset, using offset mode on the Line Capture Control panel. Before starting this, place one more vertex at a point which can be projected onto the arc segment we wish to trace. An example is shown below:

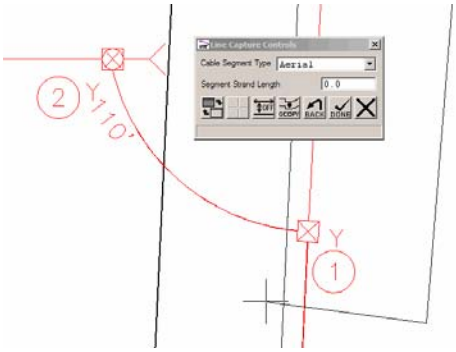


Figure 111 – About to start offset tracing the curved span.

12. Change to Offset capture mode by clicking the Offset button on the Line Capture Controls panel: 

13. Click on the line you will be offsetting from. This is the curved slack span segment. It should change color.

14. Click on a point to establish an offset side and distance for the offset segment. This is best achieved by clicking on the last point you captured in the line segment (i.e. click at the position of the cross-hairs in 119, above).

15. The system will now allow you to smoothly track around the arc drawing a curved segment at the specified offset distance. When you reach the position where you will begin entering straight segments again, click to add the end vertex of the curved segment, as shown below:

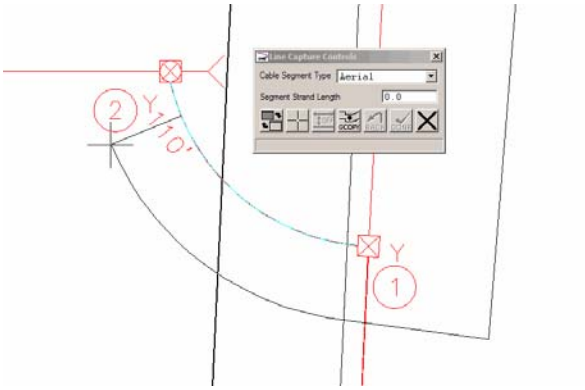


Figure 112 – Click to end the curved span.

16. To leave Offset Tracing mode, click the cross-hair button on the **Line Capture Control** panel. 

17. Up to this point, we have not captured the length of strand passed by this cable. Let's now do this by right-clicking on each strand annotation footage marker passed by the aerial segment over its entire length (alternatively you can right click on each footage as you're drafting past the strand segments). (If the system cannot unambiguously determine the strand footage marker and a pop-up is displayed, select the **RF_STRAND** option.) A running total will be displayed in the Line Capture Control panel as you add each footage.

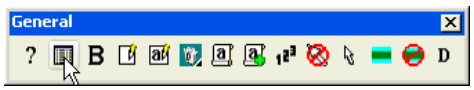
18. Draft the last section of cable, leading up to the Node. To connect the cable to the node, right click on the Node symbol and select **SH-01ND1** (the node name) from the pop-up menu.

19. We're done. Click the **Done** button on the Line Capture Control panel. This will close the panel and end line capture mode. Click **Close** on the **Fiber Cable Creation** dialog to end the Cable Creation function. 

Step 37: Insert "Tap Splice" into Existing Cable

In this step, we will insert an inline splice enclosure into the cable at the pole prior to the one containing the Node.

1. Select the cable we just created and display its details by clicking the **Details of Selected Entity** panel on the **General** toolbar.



2. On the Details panel select the cable (the topmost header of the browser tree), click the Insert button, and then select Tap Cable from the pop-up menu, as shown below. A "Scissors" tool will be displayed and will track along the cable allowing you to select the point at which you wish to insert the splice. Clicking at that point will insert a new splice enclosure.

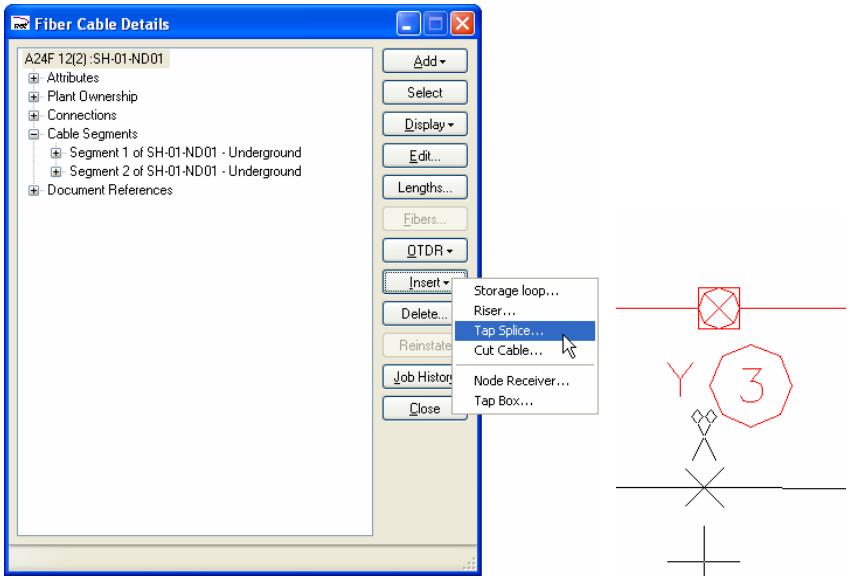


Figure 113 – Tap Cable and Scissor tool for inserting a splice case into an existing fiber cable.

3. When you select the Tap Splice… option from the pop-up menu shown in Figure 113 above, the **insert a tap splice dialog** is displayed (see Figure 114, below). This allows you to choose the type of splice you wish to insert, and to specify a name for both the splice housing (in the **Designation** field), and the splice itself (in the **Splice Name** field).

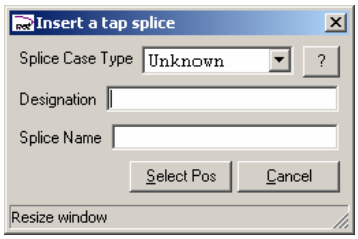


Figure 114 – Capture Tap Splice details panel

The completed network near the node should look similar to that shown below.

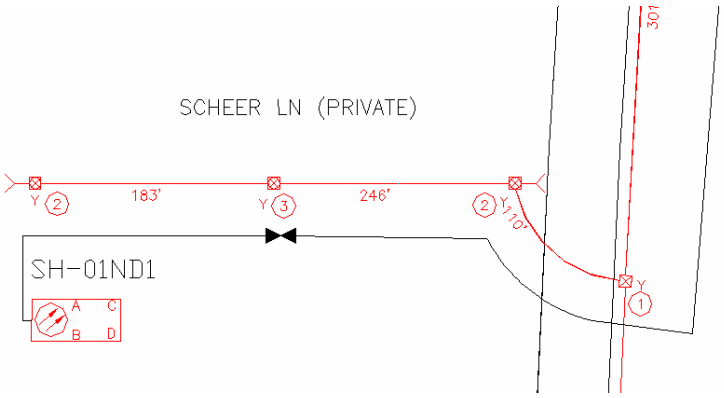
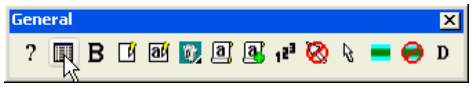


Figure 115 – Completed Cable segments showing inline splice and connection to Node.

Note: All the work you have done in this section has already been saved to the database. For non-CAD operations, SPATIALnet saves each change as you make it. You do not need to run the AutoCAD Save function.

To review the work we have just done, select the cable by clicking on it. When the cable is selected, a Move manipulator as shown in Figure 1 will be displayed at each vertex.

Click on the **Details of Selected Entity** button on the SPATIAL*net* **General** toolbar:



The details panel of the cable should display four entries under the **Cable Segments** branch, similar to that shown below:

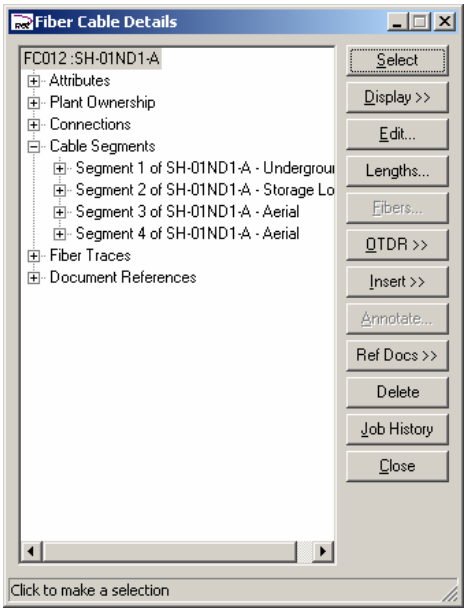


Figure 116 – Details browser of newly completed cable in Fiber Cable Details dialog box.

We shall study segments in more detail in Step 38, below. For now it is enough to observe that the four segments correspond to:

1. The Underground segment from the Shared Hub to the Riser.
2. The Storage Loop at the Riser (labeled as **Storage Loop** in Figure 116).
3. The Aerial segment between the Riser and the Inline Splice Case.
4. The Aerial segment between the Inline Splice Case and the Node.

Step 38: Editing Segments

We have just successfully placed a cable, and notice that different portions of the cable have different characteristics, in that:

- The portion of the cable is located above ground, or underground
- The portion of the cable begins or ends at a particular building, splice case, node, etc.
- The portion of the cable has a specific length
- Each of the fibers within that portion of the cable is utilized for a specific purpose (we will learn more about this in later steps).

In SPATIAL*net* each such portion of a fiber cable is referred to as a *Cable Segment* or just as a *Segment*.

Cable segments have the following properties:

- A construction type (Aerial, Underground, Buried, Storage-loop, Pigtail, etc)
- A distinct start and end location.
- Several different length parameters (Strand Length, Design Length, As-built Length, Map length, etc,)
- Start and end sheath readings (length from the beginning of the reel)
- A sequence number indicating the order of that segment in the cable (this is assigned and maintained automatically by SPATIAL*net*).

Every time one of these properties changes along a cable, SPATIAL*net* automatically creates a new cable segment.

Note: Segments *do not* have names of their own. The cable sheath determines the cable's name, and each segment is referred to as "Cable *XXX* – Segment *n*", or "Segment *n* of *XXX*" (where *XXX* is the cable name and *n* is the segment's sequence number). If you wish to change the name of a cable part way along its length, you must split the existing cable into an old and a new cable (see the Cut Cable operation).

In this section we are going to learn how to edit segment lengths and edit the segment construction type.

Editing Segment Lengths (Calculating Total from Segments)

1. Click on the fiber cable on the map, then on the **Details** button on the **General** toolbar.

Figure 117 – Details of Selected Entity button on General toolbar

2. This action will display the **Fiber Cable Details** panel, as shown below.

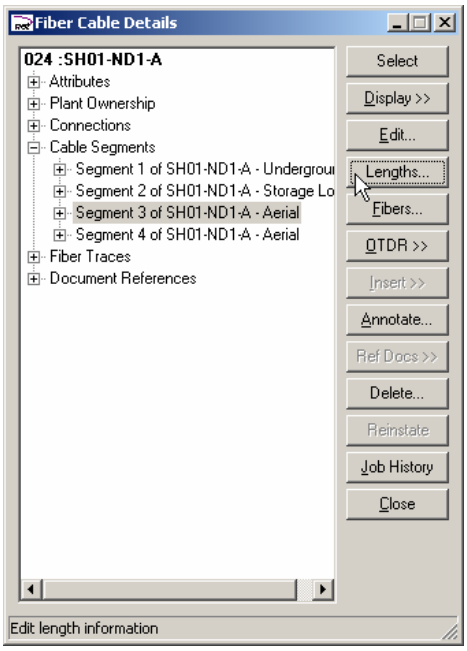


Figure 118 – Fiber Cable Details panel.

3. Click on the **Lengths** button to display the **Edit Linear Equipment Length(s)** panel.

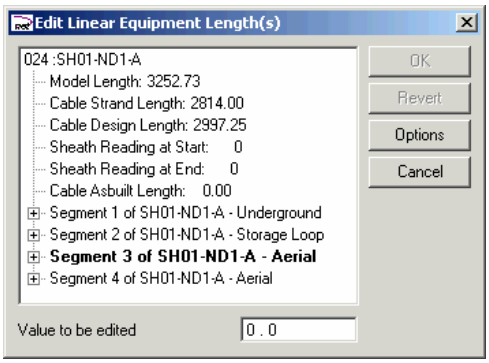


Figure 119 – Edit Linear Equipment Length(s) panel.

4. The **Edit Linear Equipment Length(s)** panel contains the following sub-headings.

Model Length:	Length of cable as determined by AutoCAD.
Cable Strand Length:	Length of support network with which the cable is associated. (It is the sum of all the segment strand lengths—information.)

Cable Design Length:	The estimated actual length of cable prior to construction.
Sheath Reading at Start:	Footage reading at the beginning of the cable.
Sheath Reading at End:	Footage reading at the end of the cable.
Cable Asbuilt Length:	Actual constructed length of cable.

5. Try clicking on each of the sub-headings, and watch the **Value to be Edited** box at the bottom of the panel. You will notice that for some sub-headings the box is grayed-out. This means that you cannot edit those options. (There is a way to override this feature, however we will return to this point in the next section.)

The values in the sub-headings above refer to the *total* cable. In this section we are going to edit the values of two of the cable *segments*, in that we are going to move 50 feet of cable from **Cable Segment 3** to **Cable Segment 4**. Notice that although the individual cable lengths will change, the overall design length will not be affected.

6. In the **Edit Linear Equipment Length(s)** panel, click on the **Segment 3** heading to display the sub-headings.
7. Click on the **Segment Design Length** sub-heading, and then enter the new value (subtract 50 feet) in the **Value to be edited** box. (In this case the new value will be 2347.0. However, the original value may be different in your version.)
8. Click the **OK** button. (The **OK** button will be grayed-out until you make a change in the **Value to be edited** window.)

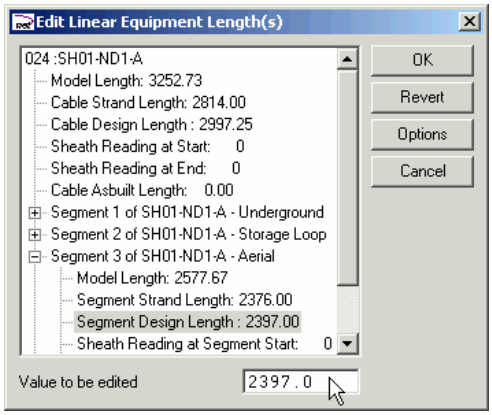


Figure 120 – Edit Linear Equipment Length(s) panel.

9. The new value should now be used everywhere the **Segment Design Length** of Segment 3 is displayed (this includes panels, any map annotation, etc.).

Now we are going to add the 50 feet of cable we just subtracted from Segment 3 to Segment 4.

1. Repeat steps 6-8, except this time edit **Segment 4**, and *add* 50 feet of cable to the original value.

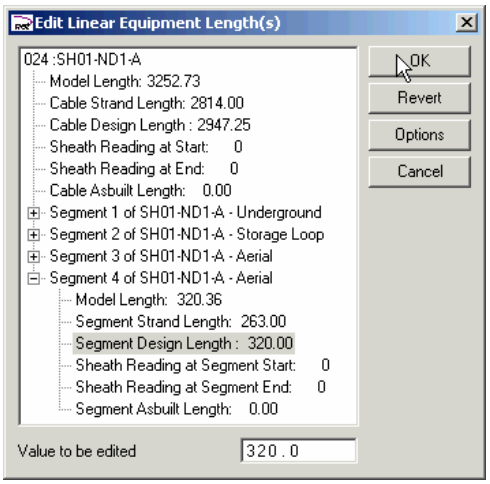


Figure 121 – Edit Linear Equipment Length(s) panel.

2. The new value should now be entered in the **Segment Design Length** of Segment 4.

Note: Clicking the OK **button commits the changes to the database. Clicking on the** Revert **button *before* clicking the** OK **button will restore the original values. To restore the original values *after* you have clicked the** OK **button you will need to select** SPATIALnet > Undo…**.**

Editing Segment Lengths (Manually Overriding Data)

In the above section, we changed the design length of the two individual segments. But notice that the *overall* design length did *not* change, as we merely subtracted 50 feet from one segment and added it to another. (Look at the **Cable Design Length** entry at the top section of the **Edit Linear Equipment Length(s)** panel.) If we had added 50 feet to both segments, the overall design length would have increased by 100 feet.

By default, SPATIAL*net* calculates the *overall* design length for a cable by adding up the design lengths of the *individual* segments. However, in some circumstances it may be necessary to edit the *total* design length without changing the data entered into the *individual* design segments. To edit the *total* design length without changing the *individual* design segments we will need to "manually override the data".

In step 5 of the above section we saw that it was not possible to edit the design length, as the **Value to be edited** window was grayed out. To override this:

1. Click on the **Options** button in the **Edit Linear Equipment Length(s)** panel.

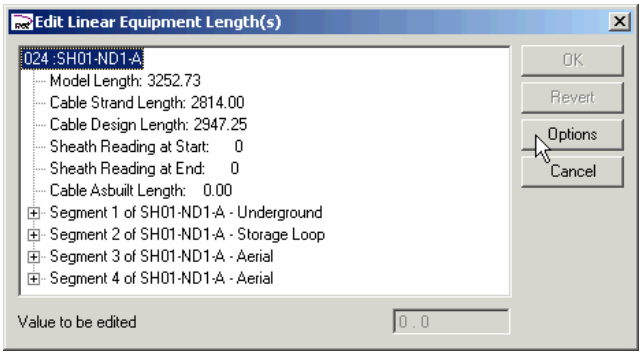


Figure 122 – Edit Linear Equipment Length(s) panel.

2. This will display the **Length Edit Options** panel, shown below.

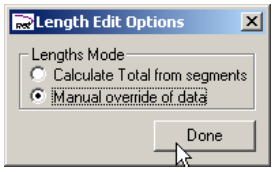


Figure 123 – Length Edit Options panel.

The **Length Edit Options** panel contains the following options:

Calculate total from segments:	Selecting this box results in SPATIAL*net* calculating the total cable length from the individual cable segments.
Manual override of data:	Selecting this box enables the user to edit the summary fields directly**.** **Note that this may mean that the values in the summary fields are no longer the totals of the corresponding values in the cable segments.**

3. Select the **Manual override of data** option, and click **Done**.
4. Now click again on the sub-headings in the **Edit Linear Equipment Length(s)** panel. Notice now that many of the options that were previously grayed-out in the **Value to be Edited** box are now shown with a white background. This means they can now be edited.
5. Change the value of the **Cable Design Length**, then click **OK**.

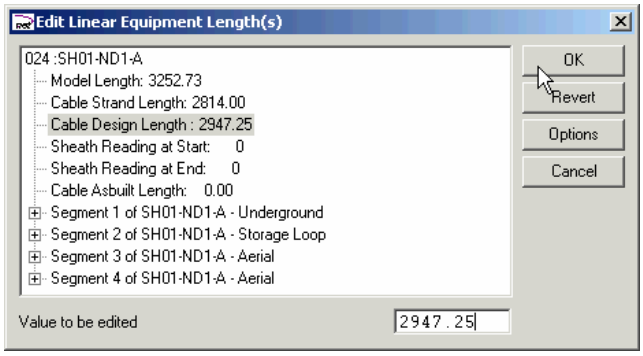


Figure 124 – Edit Linear Equipment Lengths(s) panel.

3. The new value should now be entered in the **Segment Design Length** field.

Note: Clicking the OK **button commits the changes to the database. Clicking on the** Revert **button *before* clicking the** OK **button will restore the original values. To restore the original values *after* you have clicked the** OK **button you will need to run the** SPATIAL*net* > Undo... **option from the main menu.**

Editing Segment Construction Type

Let us now turn to editing properties of segments other than their length.

To edit a segment construction type, choose one of the following options:

1. (a) On the map, click on *Segment 4.* (For the purposes of identification, it is shown highlighted, below.)

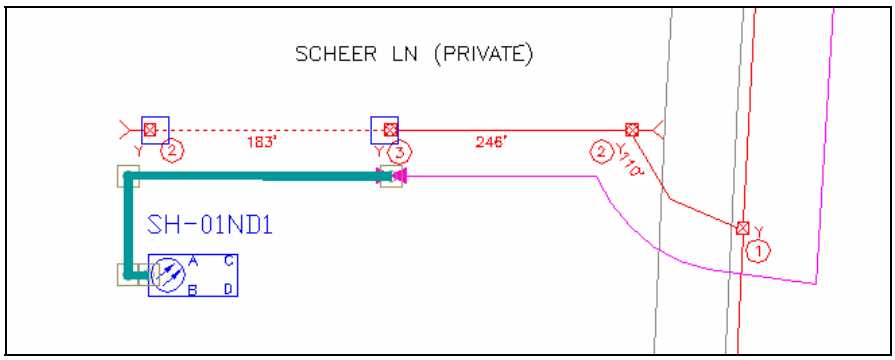
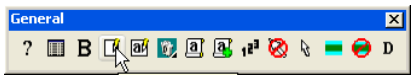


Figure 125 – Cable Segment 4 highlighted.

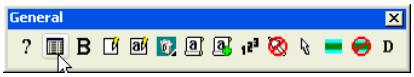
(b) Then click on the **Edit Selected Entity** button on the **General** toolbar.



Note: You do *not* need to highlight a segment to select it—it will be selected when you click on it.

or

(a) Click the **Details of Selected Entity** button on the **General** toolbar to display the **Fiber Cable Details** panel.



(b) On the **Fiber Cable Details** panel, highlight **Segment 4** then click the **Edit** button, as shown below.

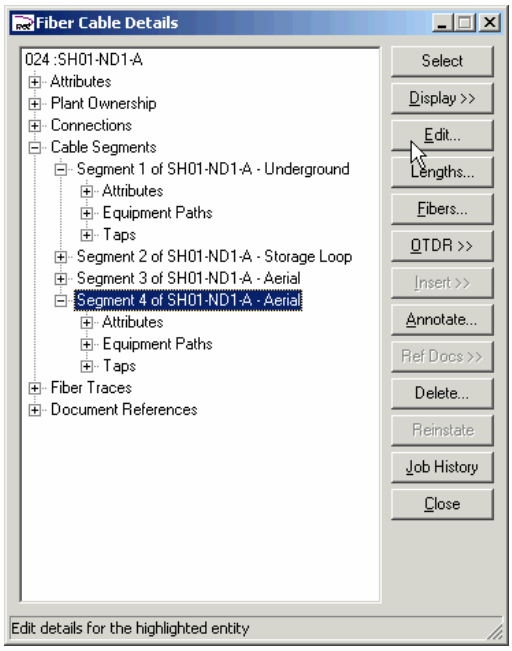


Figure 126 – Cable Segment 4 highlighted.

Note: It is often useful to be able to find the cable path in the map that corresponds to a given segment in the above list. You can do this by highlighting one of the segments displayed in the details panel. To do this, click on the segment, then either:
(a) Click the Display **button on the Details panel, and select Highlight from the pop-up, or**

(b) Click the Select button on the Details panel and then click the Highlight Selected Entity **button**  **on the** General **toolbar. To un-highlight it, click on the segment then click on the** Clear Highlight **button**  **on the** General **toolbar.**

2. Either of the above steps will display the **Segment Modification** panel. Click on the **Cable Segment Type** drop-down menu, and change the segment type from **Aerial** to **Underground**, as shown below.

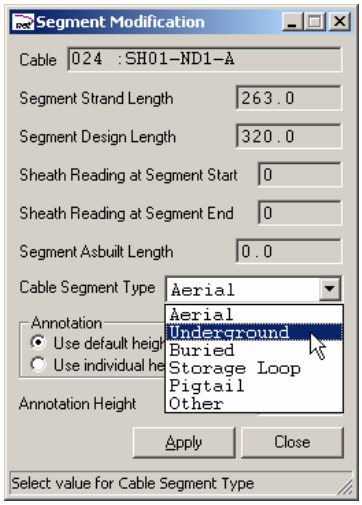


Figure 127 – Segment Modification panel.

3. Click the **Apply** button to save the change to the database.

Step 39: Cutting a Cable

In this step we are going to modify our design to accommodate another node, and change our cabling path to resemble the diagram shown below.

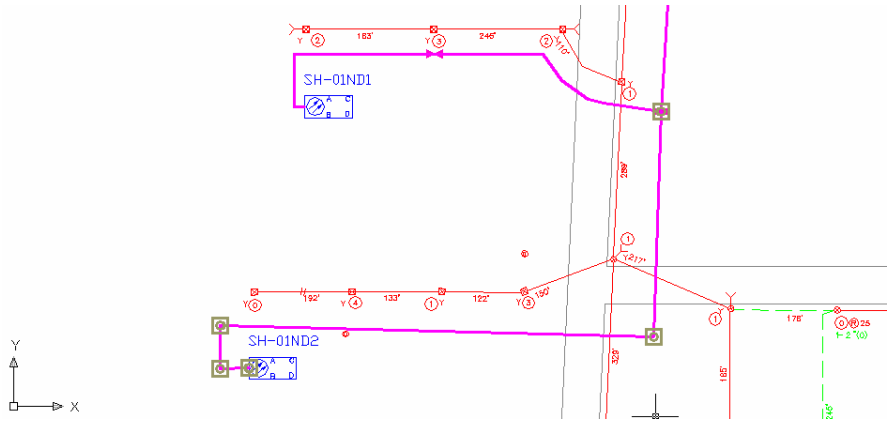


Figure 128 – Final Design Showing New Cable and Node.

We will begin this process by first placing a new node. We will use the "cut cable" function to create a new splice. And finally, we will run new cable to our new node, so that our finished design resembles Figure 128, above.

1. Place a new node called **SH-01ND2** in the place shown in Figure 128. (Refer to page 101 for details on how to add a new node.)
2. Click on the cable running to **SH-01ND1**, then click the **Details of Selected Entity** button on the **General** toolbar.

Figure 129 –Details of Selected Entity button on the General toolbar.

3. This will display the **Fiber Cable Details** panel, shown below. Click on the **Insert** button.

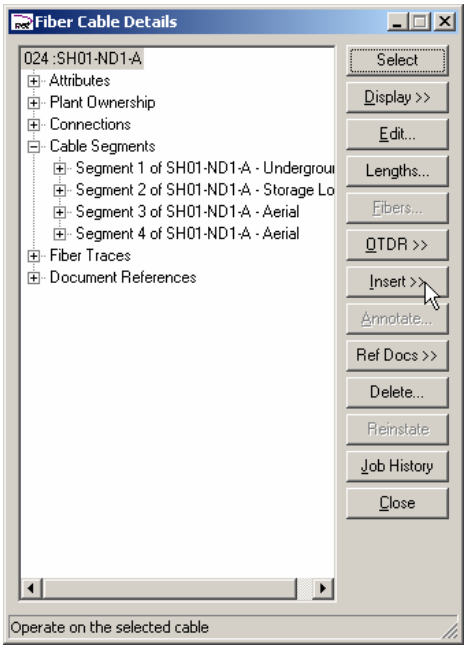


Figure 130 – Fiber Cable Details Panel.

4. Select the **Cut Cable** option from the dropdown menu.

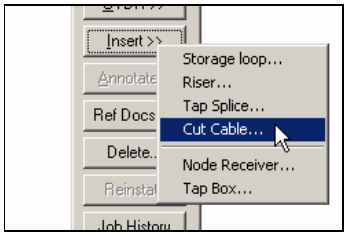


Figure 131 – Cut Cable... Option.

5. This will produce the **Cut Cable** panel shown below.

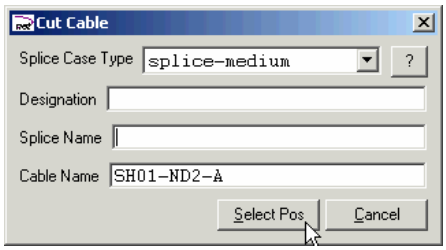


Figure 132 – Cut Cable panel

6. Enter the details into the fields, *including a new cable name*.

Note: Unlike the "Tap Splice" method (outlined on page 113) which merely creates a cable *segment*, the "cut cable" function creates a whole new cable. (See page 115 for the step on "Editing Segments" for more information.)

7. Click on the **Select Pos** button. This will create a pair of "scissors".

8. Place the cursor over the place on the cable you wish to "cut" and click on the mouse.

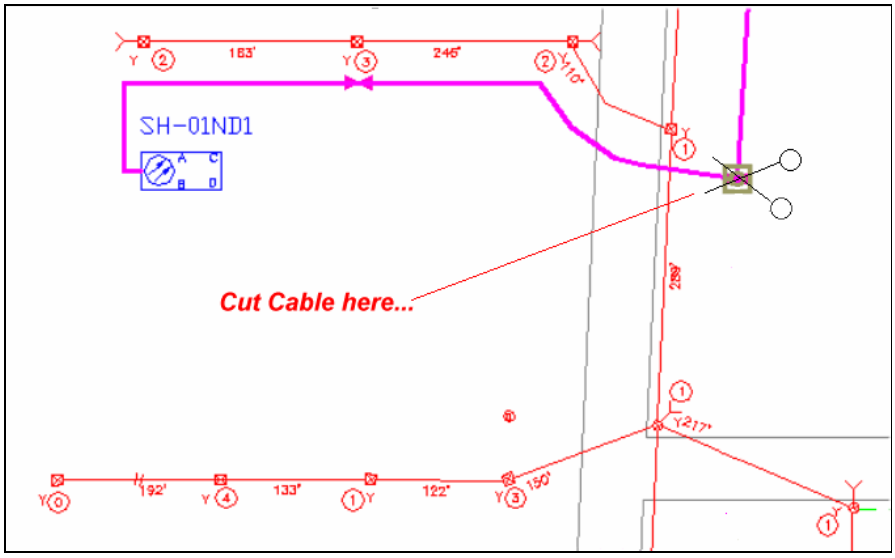


Figure 133 – Cutting the cable.

9. Now run new cable to node **SH-01ND2**, so your design resembles Figure 134, shown below.

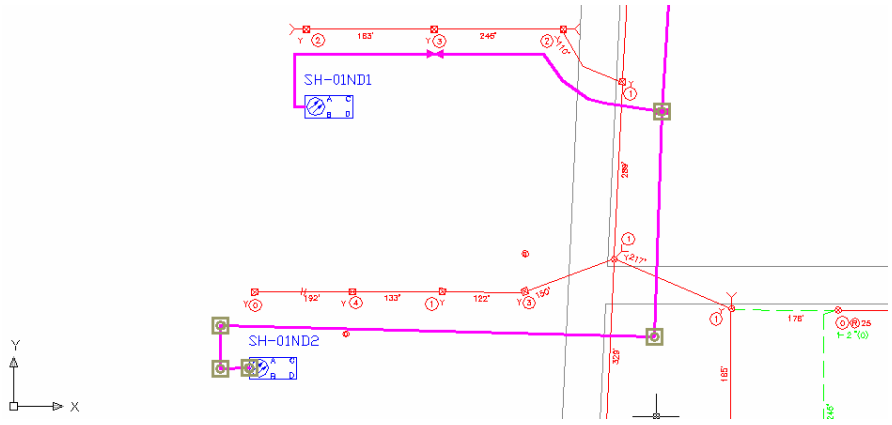


Figure 134 – New cable with new node.

Step 40: Reconnect Cable End by Dragging to New Site.

In this step we are going change the design once again; this time by adding another node, and then reconnecting the cable end from the second node to the third node. To complete the design we will then run another cable to the second node, so that the finished design looks something similar to this:

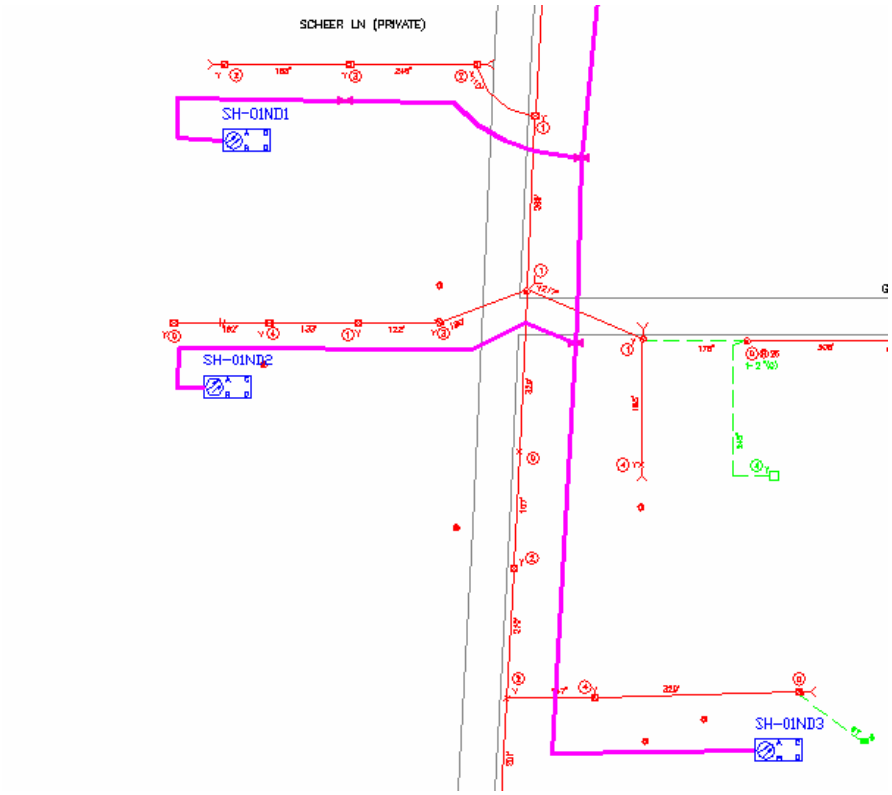


Figure 135 – Final design showing cable connected to 3 nodes.

1. Place a new node called **SH-01ND3** in the place shown in *Figure 135*. (Refer to page 101 for details on how to add a new node.)
2. Click on cable end joined to node **SH-01D2**, and drag the end to the node you just created. When the fiber is correctly placed, a pop-up box should appear, as shown below.

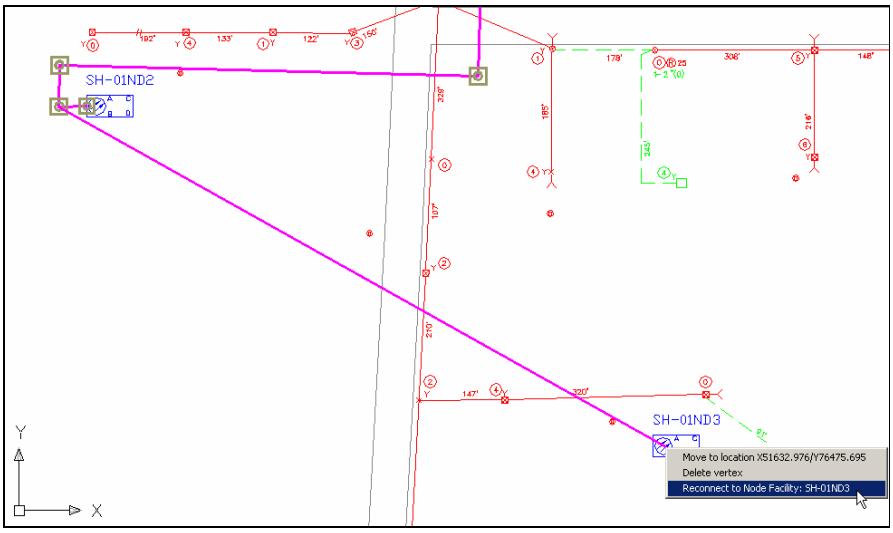


Figure 136 – Dragging cable end to a new node.

3. Highlight and click on the Reconnect to Node Facility: **SH-01D3**. The cable should now be connected to the new node.
4. Use the cable manipulators to realign the cable so it bypasses node **SH-01D2**. (Refer back to page 42 for instructions on using manipulators.)

Now we have just one last step to do, and that is to run a new cable to our second node.

5. Run a new cable to node **SH-01D2**, using either the "Cut Cable" method outlined in the previous step on page 123, or the "Tap Splice" method outlined on page 113.
6. Use the cable manipulators to realign the cable and tidy up your design, checking that it looks similar to the final design shown in Figure 135, above.

Step 41: Configure Optical Coupler

In this step, we shall configure an optical coupler, that is, a passive device that splits (or combines) the optical signals to and from multiple fibers.

Optical couplers are modeled as a type of optical node, with the special property that a signal path exists between one or more pairs of ports within the node.

To configure an optical coupler:

1. Run the menu option **SPATIALnet > Dictionaries > Fiber Definitions > Node Receiver Definitions...**
2. When the **Node Receiver** list panel is displayed, click the Add button. A panel similar to that shown below will be displayed. Fill in the values as shown below.

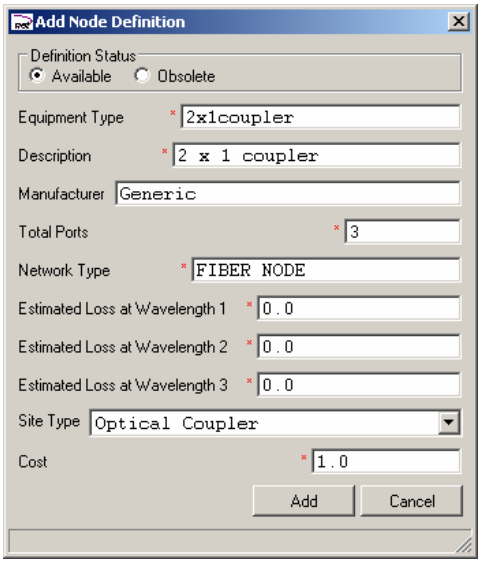


Figure 137 – Node Definition Creation panel

3. Click the **Add** button to add the new coupler definition to the system.
4. Now the ports on the coupler must be defined, along with the signal path connecting them. To do this, Click on the **Ports** button of the **Node Definitions** list panel. Click the **Add** button to create 3 ports called IN, OUT1 and OUT2, so the result appears as shown in the panel below.

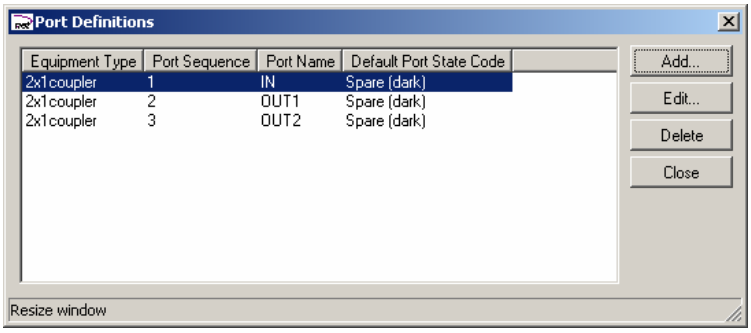


Figure 138 – Ports to create for coupler

5. Click the **Close** button to dismiss the **Port Definitions** list.
6. To define the signal path between the ports, click the **Mappings** button on the **Node Definitions** panel. This will open a list panel displaying all the port-to-port paths through the device (it should currently have no entries). Click the **Add** button to add the first port mapping.
7. A panel similar to that show will be displayed. Fill in the values as shown to map the IN port to the OUT1 port.

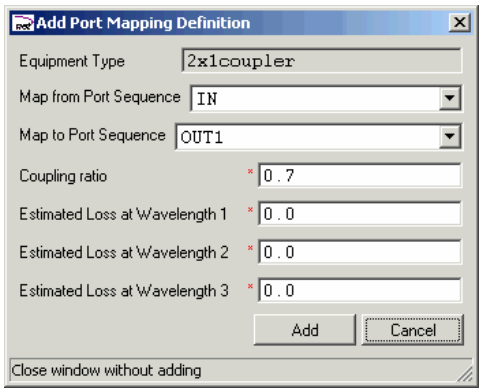


Figure 139 – Port Mapping panel

8. Click the Add button to save the port mapping to the system.
9. Add another port mapping to connect IN to OUT2.

Step 42: Add Fiber Annotation

It is often useful to place textual annotation on the map to provide further information about fiber cables and splices. SPATIAL*net* FM largely automates this process so that once annotation is placed, the system will ensure that the values displayed are always kept up to date and reflect the current state of the network.

In this exercise, we will add annotation to the fiber segment connected to the inline splice we placed in page 107 and annotate the splice case itself. To do this:

1. Zoom the window to the area around the inline splice, similar to that shown in Figure 115, above.
2. Click on the fiber cable line between the inline splice and the HFC Node to select the cable segment.
3. Click the **Annotate** button on the SPATIAL*net* **General** toolbar, as shown:

4. The panel shown below will be displayed. Select Fiber Count from the **Manual Annotations** pulldown list. This specifies the type of annotation to be added.

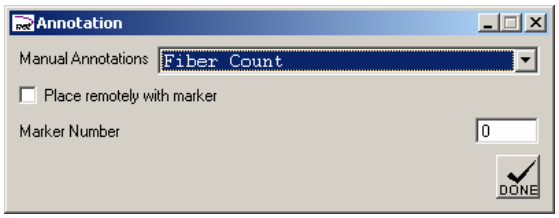


Figure 140 – Annotation Position and Type Control Panel.

5. Click at a location near the cable to be annotated. An annotation block similar to that shown below will be created at the specified position.

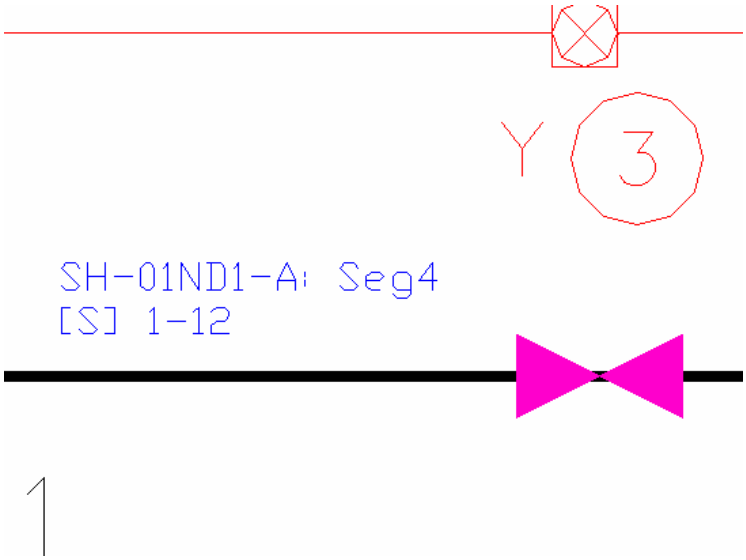


Figure 141 –Example of fiber count annotation block.

6. Click at a different location to add another copy of the annotation block. Then click the **Done** button shown in Figure 140, above to exit Annotation mode.
7. Splices can be annotated in a similar way. The only difference is that when a splice is selected, clicking the **Annotate** button as in point 3 above, will open the splice case's details panel. This is because a single splice case may contain several splices, and the system would like you to choose which one you would like to annotate. You can choose this by selecting the relevant splice in the details panel, and then click on that panel's **Annotate...** button.

Step 43: Splice Fibers

We have finished laying out the physical plant, and establishing physical connectivity between the cable and the various devices. We now need to connect optical paths and assign fibers to specific usages. In this exercise, we will:

- Splice specific fibers in our new cable to input ports of the HFC Node.
- Splice all fibers in our new cable to the output ports of the term panel in the Shared Hub.
- Modify splicing at the inline splice enclosure.

Splicing is very simple and intuitive in SPATIAL*net* FM. Regardless of the type of optical connection (fiber to fiber, fiber to device, fiber to term panel port, etc.), a single toolbar button and a single panel is always used. We begin by splicing the cable we just created to the HFC Node. To do this:

1. Select the HFC Node by clicking on it.
2. Click the **Splicing** button on the SPATIAL*net* **Network Tools** toolbar.

3. The splicing panel for the node will be displayed, with each of the fibers in the cable listed on the left, and each of the ports in the node listed on the right.
4. To connect one or more fibers to ports, select the number of fibers you wish to connect (hold down the **Shift** key to select a contiguous range, or use the **Ctrl** key to add individual items to the current selection).
5. When the same number of objects are selected on both sides, you can click on the Connect button. This will form an optical connection between each selected objects on the left, and a matching selected object on the right. (Take note of the changes to the **Usage** and **Priority** information assigned to the fibers when they are connected to ports on the node. Also, take another look at the annotation you placed in Step 42

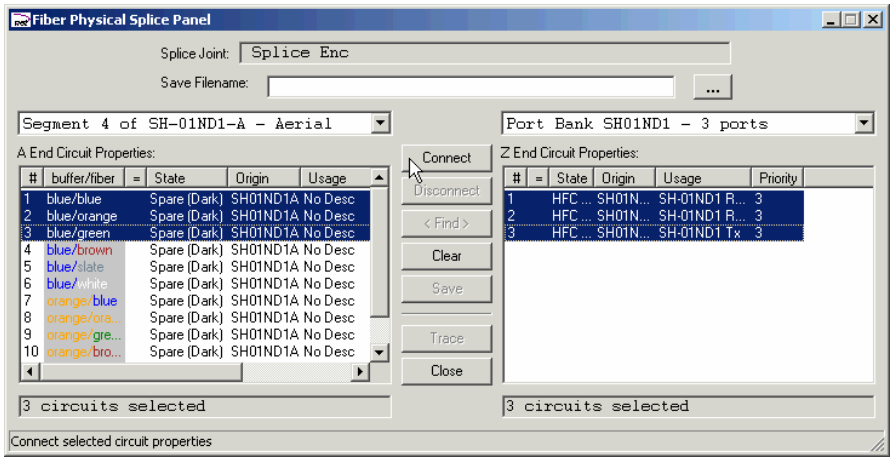


Figure 142 – Splicing panel showing the connection of fibers 1-3 to ports 1-3.

A similar process should be carried out at the Shared Hub. This time, the fibers will be on the right and the ports on the left. But the splicing operation is carried out the same way. Again, take note of changes to the **Origin** of the fibers once they are connected to the term panel in the Shared Hub. This change will ripple down all fibers and equipment on the optically connected path.

As a last exercise for this step, try to de-splice (or disconnect) the unused fibers (numbers 4-12) at the inline splice near Node SH-01ND1. The process for this is very similar to the splicing operations we have just done, with the exception that we highlight only one side, and click the **Disconnect** function, rather than the **Connect** function. When the fibers are disconnected, note the changes to the splicing annotation.

Step 44: Trace Fibers

To verify that the optical path has been spliced correctly you should run SPATIAL*net* FM's Trace functionality. This allows you to start at any point in an optical path and follow all connections:

- Upstream.
- Downstream.
- In both directions.

Trace results are shown on the map by highlighting cable sheaths and equipment, and several textual report formats are produced. Traces can also be saved in the database for later reporting, or export to third-party tools.

We will run the trace function on fiber 1 of the sheath we created in Step 36, assuming that it was spliced as shown in Step 43. To do this:

1. Select the line representing the fiber sheath by clicking on it in the map view. When the cable is selected, a Move manipulator as shown in Figure 1 will be displayed at each vertex.
2. Click on the **Fibers/Ports list** detail button on the SPATIAL*net* **Network Tools** toolbar:

3. Select **Fiber #1** in the **Fiber List** panel, and click the **Trace** button, as shown below.

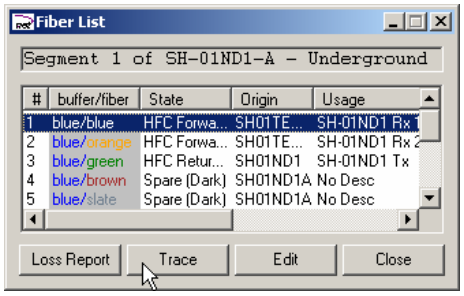


Figure 143 –Fiber List dialog box.

4. Set the fields of the Trace Configuration panel as shown below.

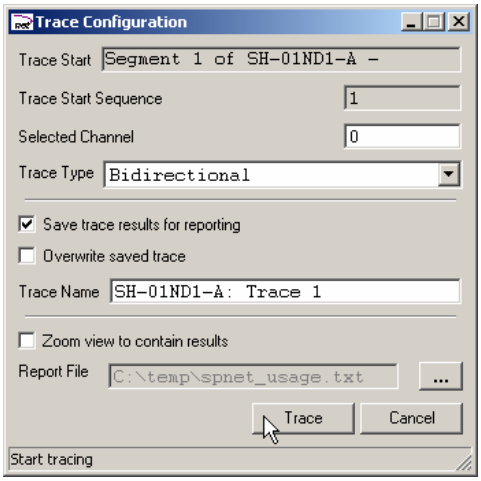


Figure 144 –Configuration for a Bidirectional Trace of Fiber 1 of cable SH-01ND1-A.

Note that the Save trace results for reporting **check-box must be checked, or the listing and reporting functions below will differ from those shown below.**

5. Click the **Trace** button in Figure 144. The system will trace optical connectivity both upstream and downstream. The trace will highlight any cable sheaths it passes through, and should begin at port 1 of the term panel we placed in the Shared Hub, and end at port 1 of the HFC Node.
6. The trace results will initially be displayed in a tree similar to that shown below. If you expand the entries under the **Next** branch, you can walk along the entirety of the traced network.

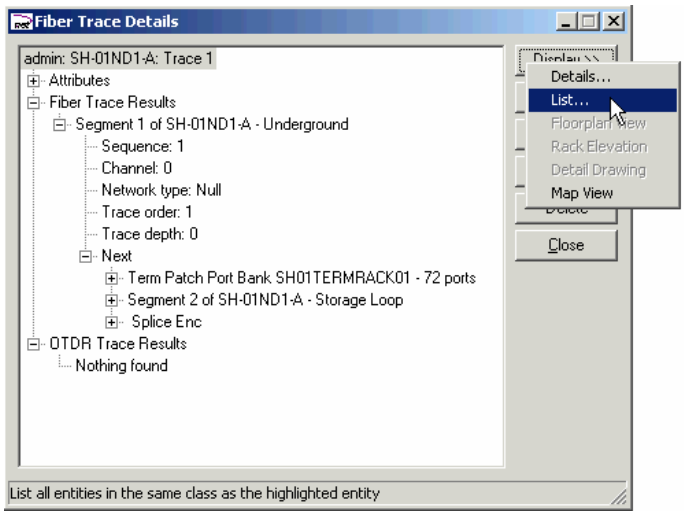


Figure 145 – Trace Results tree.

7. To view the trace results in other formats, click the **Display > >** button and select **List...** from the pop-up as shown in Figure 145. This will display a list of all saved traces in the system.
8. Select **SH-01ND1-A: Trace 1**, the trace we just created and click the **Grouplist** button. This will display the trace results in sequence in a tabular format. The **Expand** and **Contract** button let you show or hide network segments associated with different logical circuits (specifically, contiguous segments with the same **Network Type**). Since we have not created any logical circuits in this exercise, these buttons will expand and contract the entire network.

Step 45: Generate Link Loss Report

As the final step in our tutorial, let us generate a Link Loss report for the fiber we traced in Step 44 To do this:

1. Select the line representing the fiber sheath by clicking on it in the map view.
2. From the pull down SPATIAL*net* **Fiber** menu, select **Reports > Link Loss Report...**. The **Fiber Link Loss Report** panel will appear. This allows you to control where and how the link loss report is generated. Use the default values assigned by the system. They should be similar to those shown below:

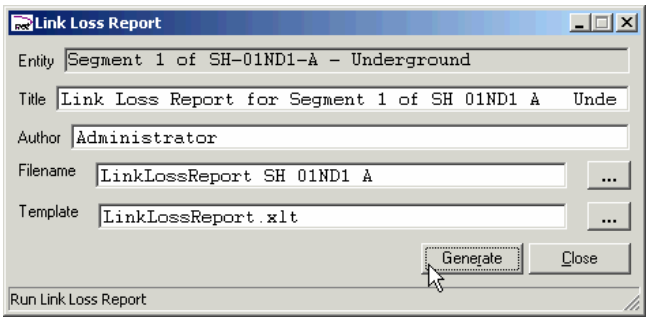


Figure 146 – Fiber Link Loss Report creation panel.

3. Click the Generate button in Figure 146. This will create a Microsoft Excel spreadsheet with loss information, similar to that shown below:

Project Name:		**Release Date:**		
Project Number:		**Revision Date:**		
Serving Hub:		**Design By:**		
		QC Check By:		
Statistics for Fiber Hub to Node Receiver				
Start Node	**End Node**	**Distance to Node**	**Estimated Loss**	**Actual Loss**
SH-01ND1 Node4in	SH-01-TERMRACK-01 Rack 72	5611.8	5.6	0.0

Figure 147 – Fiber Link Loss Report output.

Note that Estimated Loss is calculated by the system using the length of the cable multiplied by the linear **Attenuation Coefficient** entered into the fiber specification (see Step 33, Figure 102), along with any splice losses.

The following chapter covers topics specific to Fiber To the Home/Premises/Curb/User networks (collectively referred to as FTTX networks). The assumed network architecture runs multi-channel fibers to a FTTX cabinet. In the cabinet, each channel on each feeder fiber is passively split onto its own home-run distribution fiber, which connects an individual home network interface device to an output port in the cabinet. Note that this architecture has been chosen simply to provide a concrete example.

Any other FTTX architecture can be managed by configuring the various equipment dictionaries appropriately.

In the exercises, you will:

- Extend the landbase.
- Configure and Create a FTTX distribution cabinet site.
- Create new strand network.
- Record new addresses.
- Run fibers to connect the addresses to the FTTX cabinet.

Chapter 5

FTTx Networks

This chapter introduces SPATIALnet FM functions developed specifically for designing fiber to the home/curb networks (collectively referred to as FTTx). The previous chapter presents pre-requisite knowledge required in this chapter and should be completed before commencing the exercises below.

Step 46 : Configure FTTX Distribution Cabinet Site

The FTTX Distribution Cabinet will be modeled as a site, just like the Shared Hub we configured in Step 18. Re-run Step 18, but use the values shown in the panel, below:

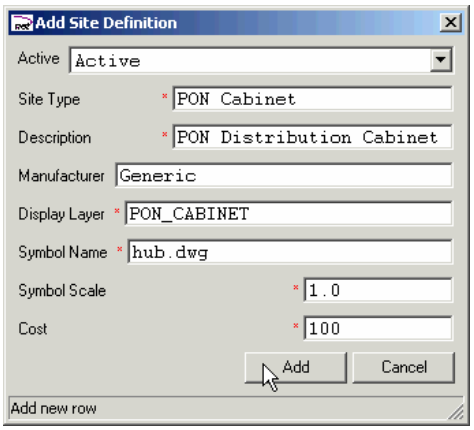


Figure 148 – New FTTX Cabinet Site Type.

Note that we have used **hub.dwg** as the symbol to represent the FTTX Cabinet. You may wish to change this to a new symbol that you create.

Recall from Step 18 that to display attributes as text annotation associated with the symbol, you need to link the attribute tags in the symbol to fields stored in the database. We would like the name of the cabinet to be displayed on the map, so create an **Attribute** entry called **NAME** for FTTX Cabinet as shown below (see Step 18, page 62, for more details on adding attributes). If you have used your own symbol, you will need to use the attribute tag that you defined, instead of **NAME**.

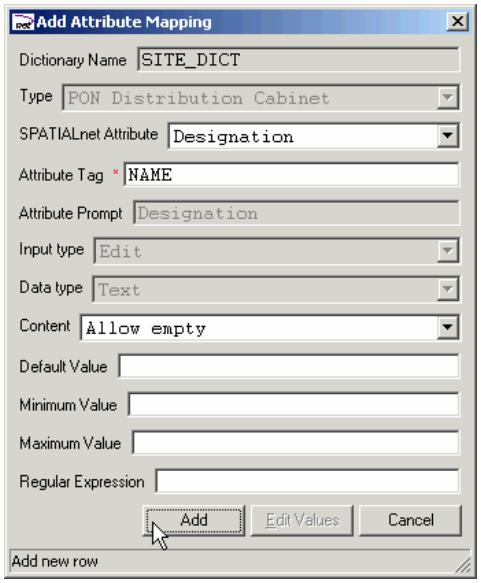


Figure 149 – Define Attribute for FTTX Cabinet Site Type.

Step 47: Add New FTTX Distribution Cabinet Site

Now that we have configured the type of site we wish to add (see Step 46, above), we can add it to the newly developed area we are designing. Usually, you will be able to Clone an existing site, using the **Clone** button on a site's **Details** panel. This will eliminate the need to have to create the internals of each site from scratch every time. However, you will have to set up at least one site's internal structure, so that is what we will do here.

By following the same procedure as in Step 19 (but choosing **FTTX Distribution Cabinet** instead of **Shared Hub** as shown below),we can place our new site at the location shown in Figure .

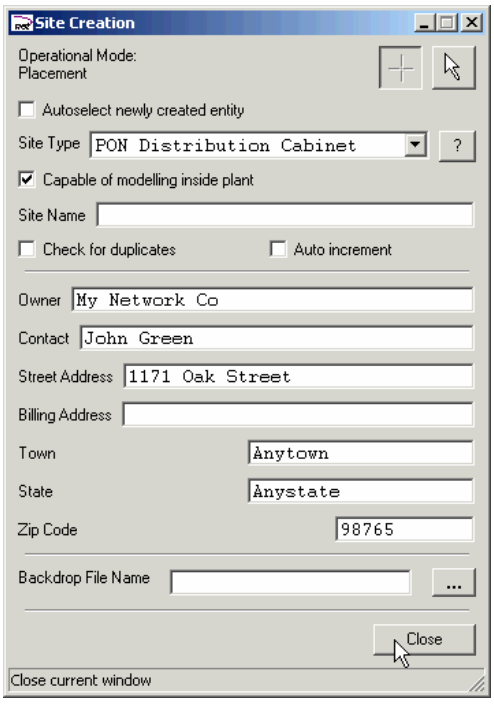


Figure 150 – Define Attribute for FTTX Cabinet Site Type.

Note that this time, the **Capable of modeling inside plant** check box is set. This is important, as failing to do this will prevent you from placing the site's internal equipment into the FTTX cabinet.

Once the site has been created, we will need to begin specifying its internal layout. Because sites are modeled using SPATIALnet FM's inside plant functions, all sites must have at least one "floor". To add a floor to the new site:

1. Select the site
2. Open its **Details** Panel
3. Click the **Add** button
4. Select **Add a Floor to the highlighted site.**

After you have created the floor, you must add a rack or frame within which the OSP cables can be terminated, and equipment can be mounted. To do this:

5. Select the floor you just added in the same details panel.
6. Click the Add button again.
7. Select **Add a rack to the highlighted site floor**.
8. Supply values in the **Rack Creation** panel to add the new rack. (See *: Adding* Racks in Part B on page 209 for more information about adding racks to the system).

Step 48: Cloning a FTTX Distribution Cabinet Site

In this step we are going to add another FTTX distribution cabinet site. However, as ISP buildings such as these can be complex structures, it is often cumbersome to create them from scratch, especially if a building very similar to the one being created is already in existence. In this situation it is much easier to tell the system to copy, or "clone" an existing building—including all the ISP characteristics.

To "clone" a new ISP site:

1. Select the site to be cloned. In this example, we are going to clone **PON-1**, shown below.

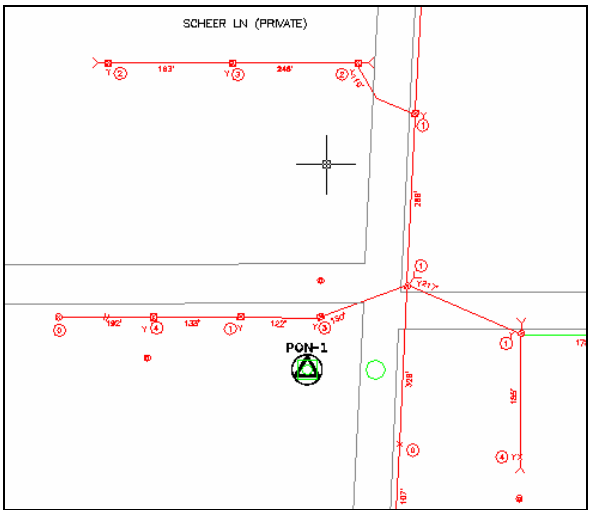


Figure 151 – Location of PON-1 site.

2. Open its **Details** Panel, and click on the **Clone...** button.

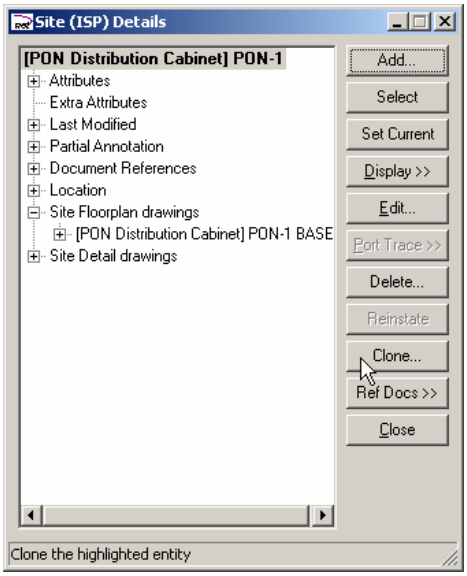


Figure 152 – Clone button on PON-1 Details panel.

3. This will display the **Site (ISP) Clone** box. We now need to give our new site its own name. Change the site name from **PON-1** to **PON-2**.

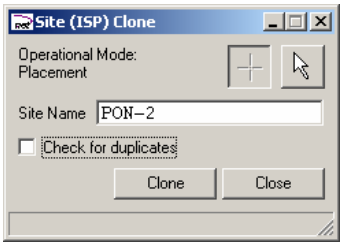


Figure 153 – Clone button on PON-1 Details panel.

4. Place the cursor over the place on the map where you wish to place the newly cloned site (see map below), and click the mouse.

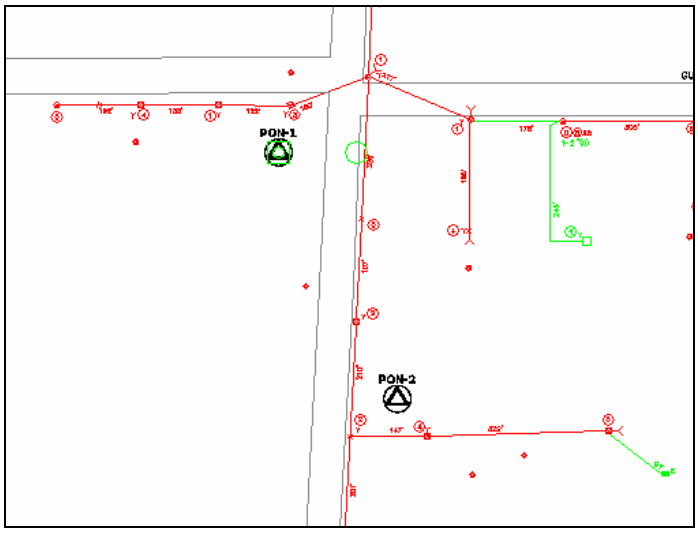


Figure 154 – Location of new site.

5. **The Entities being cloned** panel will appear. This lists all of the individual objects that will be replicated when the site is cloned. Click on the **Confirm** button to place the site, as shown below.

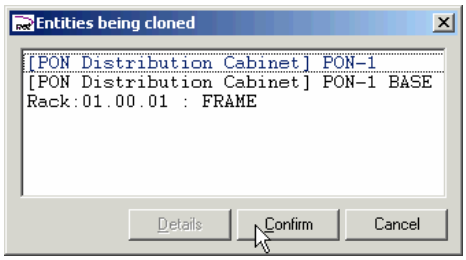


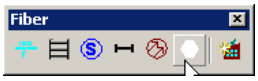
Figure 155 – Entities being cloned panel.

Step 49: Add Fiber Taps

Now that the poles, strand, and addresses have been entered, we can place taps and create drops to the addresses (the system will automate much of this). Taps are configured in the same way as other equipment (see, for example, *: Configure New HFC Node Type and Usage* Codes on page 98). We will assume that the system's **Tap Box Definitions** dictionary already contains the taps you wish to add. If not, you should be able to define the equipment types you need.

To add Fiber Taps to the system,

1. Run the menu command **Fiber > Add > Tap Box...**
   or
   Click the **Add Tap Box** button on the **Fiber** toolbar



2. A panel similar to the one below will appear. Choose the Tap Box Type and enter a name.

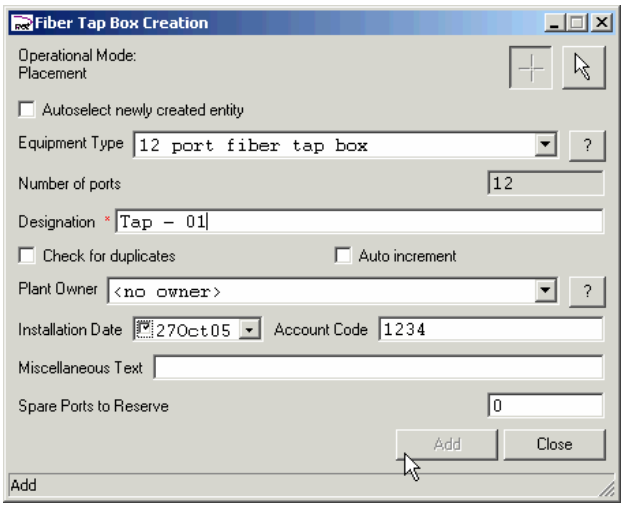


Figure 156 – Fiber Tap Box Creation dialog box

3. Click on the AutoCAD map area once to transfer the input focus, and then click at the location of each new tap. They should be drawn close to the poles so that the system's automatic attachment process can assign the tap to the right pole. (We will run this process later. The distance used by the system to determine if equipment is attached to a structure is specified in **Attachment Tolerance** field of the configuration panel for the pole type. See **Error! Reference source not found.**, above).

When done, the new network should look similar to that shown below:

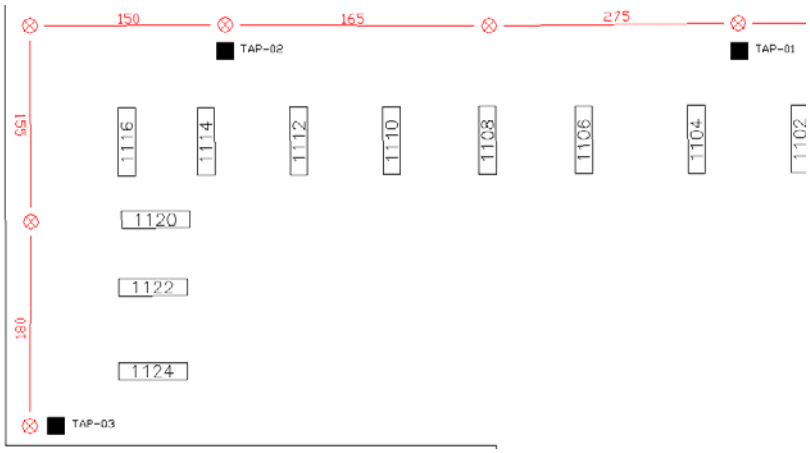


Figure 157 – New Taps

Step 50: Associate Taps with Poles

The system will automatically associate the taps you placed in Step 49 with the poles near to which they are placed. There is also a manual process for doing this.

The automated process requires that you first draw a "fence" (a polygon) around the area within which you want the system to search for taps which need to be attached to poles. A polygonal area of any type will do, so you could select the boundary you created in Step 7. Alternatively you can create a new boundary around the area of the job you are currently working on, and use that instead.

To have the system automatically use proximity to associate the taps to the poles to which they are attached:

1. Select the boundary that encloses the area you wish to scan.
2. Click the **Attach Equipment by Polygon** button on the **Construction** toolbar.

Check that the association was made by clicking on one of the poles with a tap attached to it. The tap should be shown as attached to the pole, as in the area highlighted below.

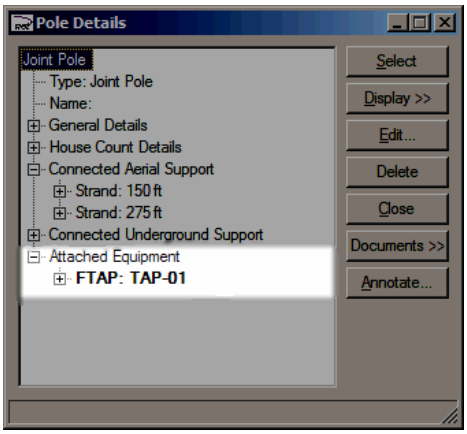


Figure 158 – Tap attached to pole after automatic attachment by proximity function run.

Step 51: Create Drops

The last step in creating the infrastructure needed to connect addresses to the network is to create drops. This process is also significantly automated, via a "wizard" that does most of the work for you.

To run the drop wizard:

1. Select a tap box by clicking on it in the map.
2. Click the **Drop Tool** button on the **Network Tools** toolbar.

3. A panel similar to that shown in Figure 159, below will be displayed.
4. The drop tool always operates on a single tap box. You must dismiss the tool and re-start it if you wish to change tap boxes. While you have a single tap box selected, the tool works as follows:
   - Click on any support structure (pole, pedestal, vault, etc.) and any drops to be created will be run from that structure to the address they serve.
   - Click on any group of addresses. These will appear in the address list window and be assigned to the next sequence of spare ports in the tap box. If there are no spare ports, the system will display an error message.
   - Click on a strand/trench footage. This will be added to the length of drop cable required when the structure from which a drop is connected to the address is not the structure to which the tap is attached.

5. When you click the **Apply** button, drops will automatically be created between any new addresses assigned to the tap box, from the currently selected structure. Each new address will be assigned to the next available port in the tap box, while there are still unused ports available. Addresses already connected to the tap box will be shown in the address list, but will not be affected when the **Apply** button is clicked.

The results of running this process on the area we are building are shown below.

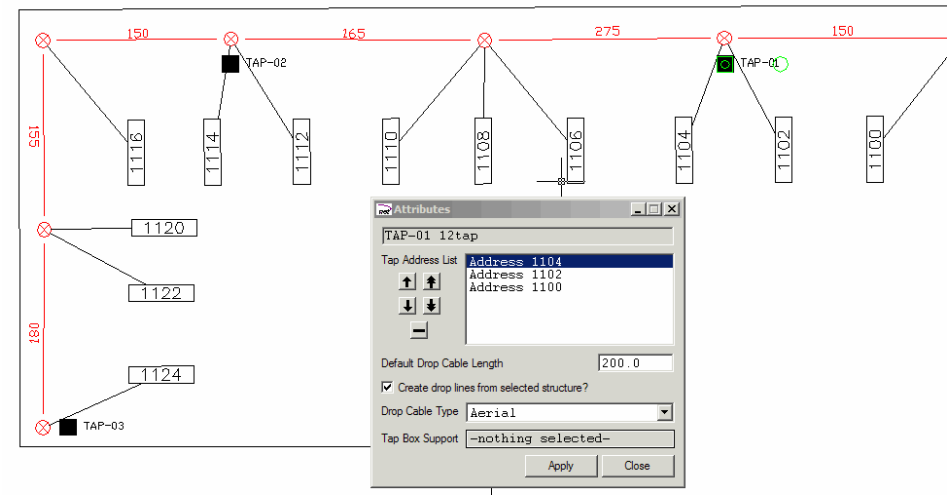


Figure 159 – Drop Tool showing auto-created drops and assignments of addresses to tap ports.

The automatic assignment of the addresses to the ports in the tap box can be seen by showing the port details of a selected tap box (via the **Fibers / Ports List** button on the **Network Tools** toolbar).

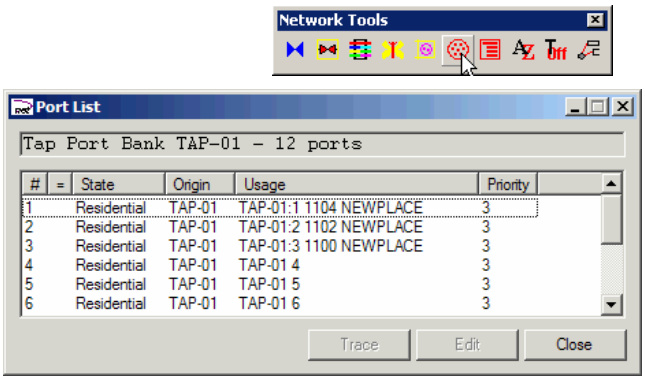


Figure 160 – Addresses assigned to Ports after Drop Tool run.

Note that you can edit the assignments to ports manually, including deleting an assignment, by selecting a port and clicking the **Edit** button in Figure 160.

Step 52: Configure FTTX Fiber Cable Types

We saw how to configure fiber cable types, and define their display layers in Step 35. FTTX cables have an additional feature you should be aware of, so let's revisit this topic briefly, before adding a fiber cable to finish off our network.

SPATIAL*net* FM allows you to run a cable through several spliceable locations by right-clicking as you define the cable path, but it does not always make the cable accessible for splicing at those locations. In FTTX networks, the suggested procedure for connecting a fiber cable to the taps along its length is exactly this right-clicking process, but the fibers obviously have to be accessible for splicing in each tap box.

To ensure this is the case, all cables to be used for FTTX distribution should be created with the **Auto Create Taps** field set to the **Auto Tap every segment at creation**.

To configure a cable type like this, run the menu function **SPATIAL*net* > Dictionaries > Fiber Definitions > Cable Definitions**, click the **Add** button to bring up the **Add Cable Definition** dialog box, and supply the values shown in the panel below (or similar).

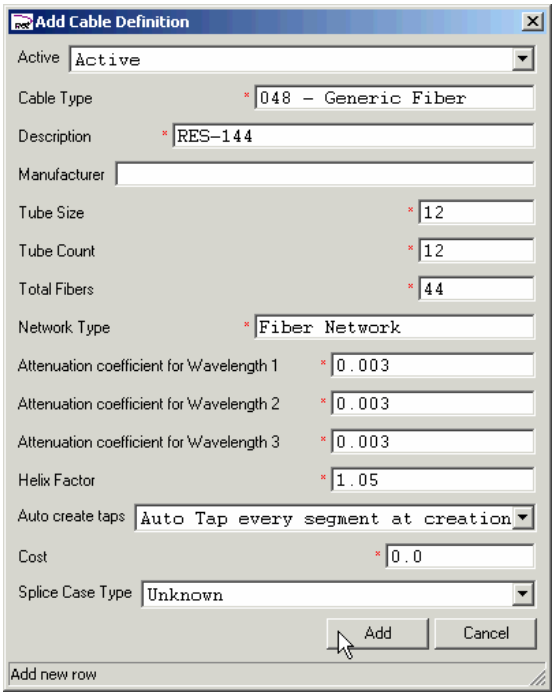


Figure 161 – Cable Type Definition for FTTX cables, showing Auto Tap every segment at creation *option.*

Step 53: Add FTTX Cable

Let us now add a fiber cable connecting each of the taps (and their associated addresses) to the FTTX cabinet we placed earlier. We shall tap the cable as we go and, in the next step, run a tool which automatically splices the required fibers to the used ports in each tap box. That will complete the job.

Adding an FTTX cable is identical to adding any other fiber cable, as was described in Step 36. You should review this now. The only difference is that FTTX distribution fibers should be tapped at each tap box they pass through, so you go past a tap box, right-click on it and select its name from the pop-up. Remember that the cable type chosen for the FTTX distribution cable must have **Auto Tap every segment at creation** set (see Step 52, above).

The network after a distribution cable has been run from the FTTX Cabinet through each of the taps is shown below. Note that annotation has also been added (see Step 42).

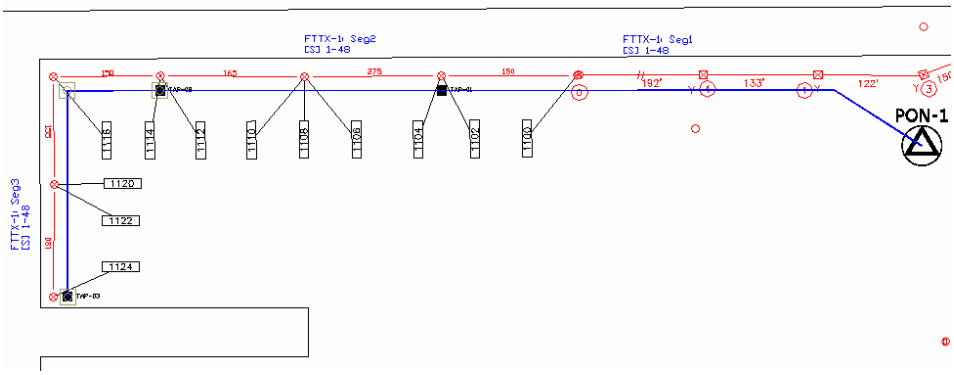


Figure 162 – Cable Added to FTTX network, to connect FTTX cabinet to taps.

Step 54: Assign Fibers to Tap Ports ("Tail Off" cable)

The last step in our job is to splice fibers to the tap ports which were associated to addresses by the Drop Wizard in Step 51, above. This is a largely automated process, which requires only that you select the sheath you wish to "tail off", as this process is called. The system will then locate all the taps along the sheath, find all the used ports in each tap, and splice the next available fiber in the cable to that port. If there are not enough fibers in the cable to connect to all of the used ports, an error message is displayed.

To run the Tail-Off tool:

1. Select the fiber cable you wish to tail off (select the one you just placed in Step 53, above).
2. Click the **Tail Off Cable** button on the **Network Tools** toolbar.

3. The Tail off panel will be displayed, as shown in the panel below.

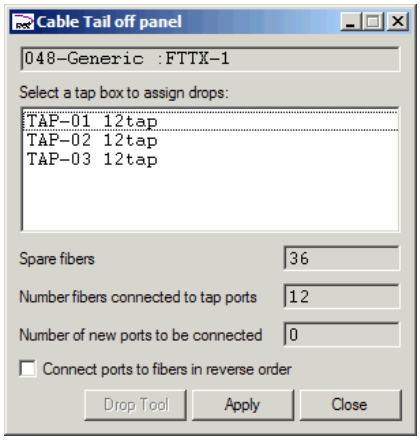


Figure 163 – Cable Tail off panel

4. Click the **Apply** button. This will automatically splice the next available fiber in the sheath to the next used port. Note that the annotation readings will automatically update as the Tail Off tool processes each tap.

Alternatively, fibers may be spliced to ports manually, by selecting the tap and using the same process as was described in Step 43, above.

This completes Part A of the tutorial. Further more detailed information about all of the topics covered in this section is available in the *SPATIALnet FM User Manual*.

Chapter 6

Outside Plant RF

In this chapter, we shall go through the process of capturing the base, addressing, support network, and RF design for the service area of a new HFC node. The steps involved in this include:

- Configuring equipment types and design frequencies in the RF dictionary
- Configuring a new RF Design Profile
- Capturing addresses and land parcels
- Drafting poles, aerial strand and underground support network structures
- Adding RF plant
- Associating RF taps with addresses

Step 55: Set Design Frequencies

The first thing we will do is specify the frequencies at which the various RF signal levels will be tracked throughout the system. To do this:

1. From the menu, select **SPATIALnet > Dictionaries > RF Definitions > Frequency Definitions...**. A dialog box will be displayed, listing the values currently specified for each of the design frequencies. The panel shown below will be displayed.

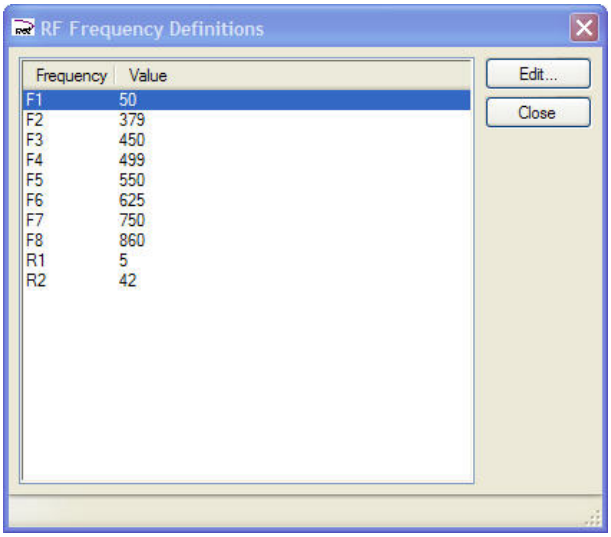


Figure 164 – Frequency Definitions

2. The Frequencies listed above are those at which design can be performed. As we shall see, not all frequencies are necessary in any given project, but the above list shows all those which are available. Any of these may be edited so that a different value is used and, if this is done, that value will be changed throughout the entire system.

Step 56: Add New RF Cable Type Definition

Now we will add the definition of a new type of RF cable to the RF cable dictionary. To do this:

1. From the menu, select **SPATIALnet > Dictionaries > RF Definitions > Cable Definitions..**. A dialog box will be displayed, listing all the cable types currently defined. Click the **Add** button to add a new cable type to the dictionary, as shown below.

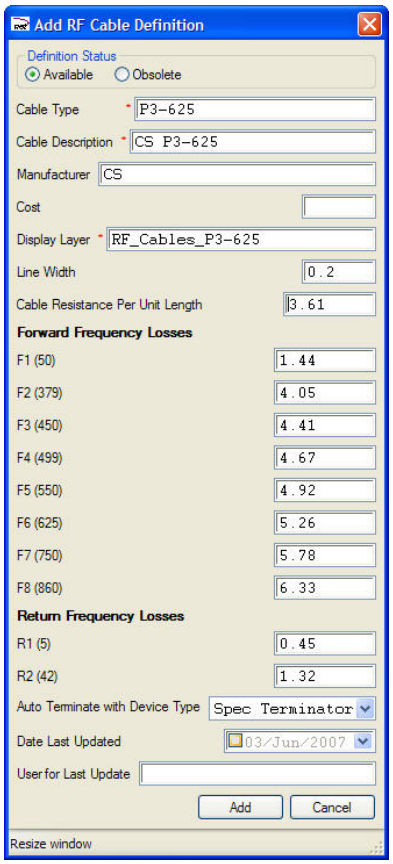


Figure 165 – Add RF Cable Type Definition

2. Fill in the field values and click on the **Add** button.

Step 57: Add New Active Equipment Definition

Let's now add a definition of a new type of active device to RF Active Devices dictionary. To do this:

1. From the menu, select **SPATIALnet > Dictionaries > RF Definitions > Active Definitions...**. A dialog box will be displayed, listing all the active device types currently defined.

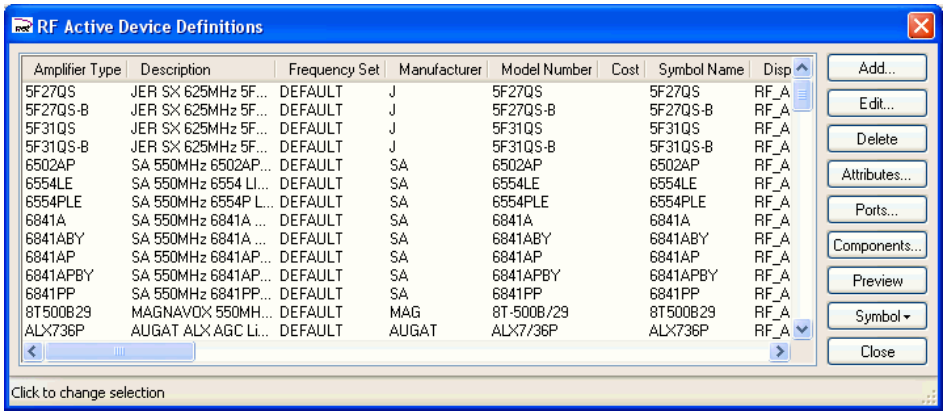


Figure 166 – RF Active Device Definitions Panel

2. Click the **Add** button to add a new active device type. A panel similar to that shown below will be displayed.

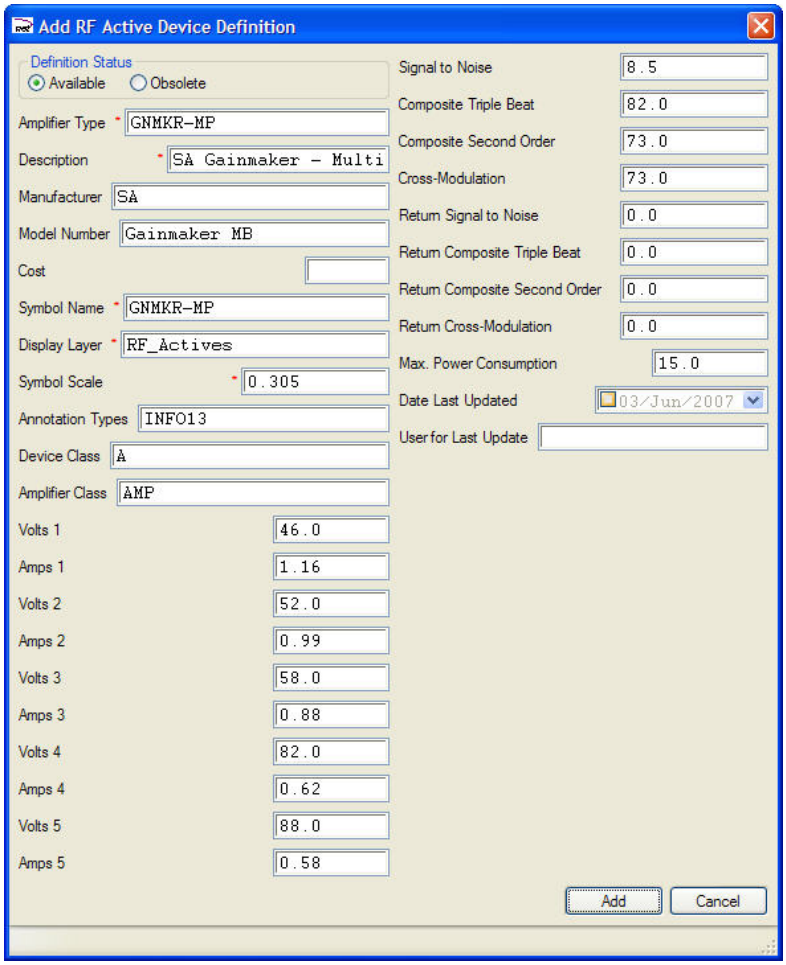


Figure 167 – Add RF Active Device Definition Panel

3. Fill in the field values and click on the **Add** button.
4. Once RF devices are defined, their port configurations must be specified. To do this:
   - Open the **RF Active Device Definitions** panel as described in procedure 1 above (if it is not still open).
   - Highlight the RF device just configured, and click the **Ports** button, as shown below.

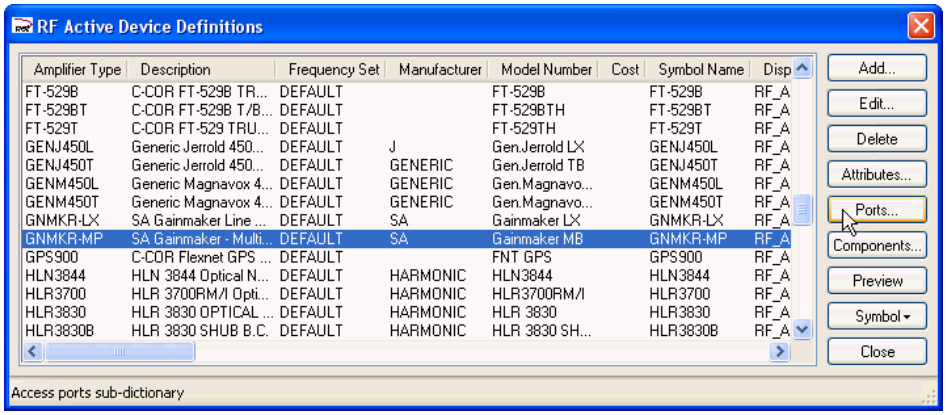


Figure 168 – Ports Button on RF Active Device Definitions Panel

5. A panel will be displayed with no entries.

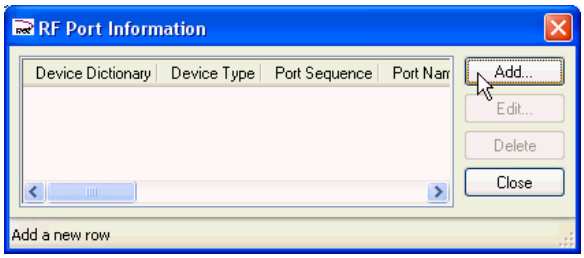


Figure 169 – RF Port Information Panel

6. Click the **Add** button to display the **Add RF Port Information** panel.

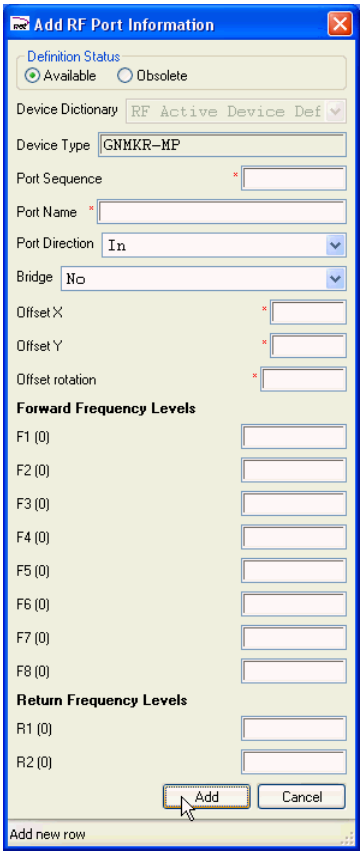


Figure 170 –Add RF Port Information Panel

7. Add an entry for each port the active device has, as shown in the example below.

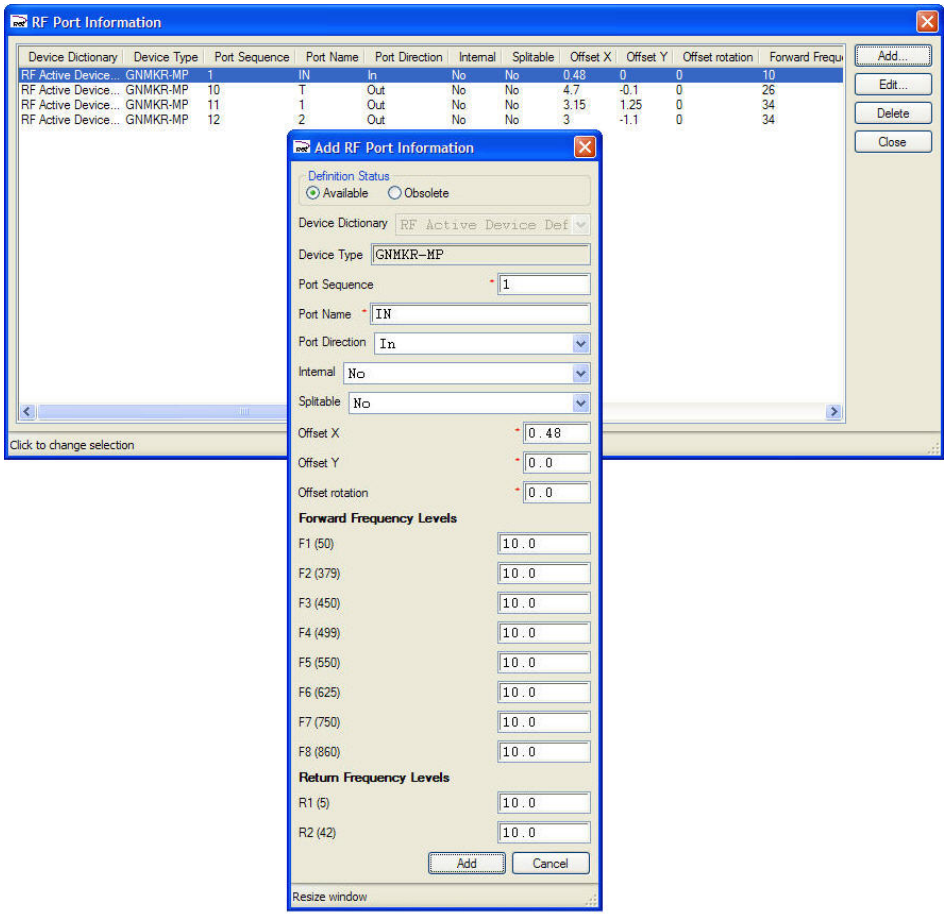


Figure 171 – Configure Active Equipment Ports

8. In the case of some multi-output amplifiers it may also be necessary to configure components for the ports. To do this
   - Click the **Components** button on the **RF Active Device Definitions** list panel. A dialog box will be displayed with no entries.
   - Highlight the components you want to associate with this active device and click select. Creation of device components is covered in step 6.

     Note: Ports must be configured as "internal" and "splittable" to be able to associate internal components such as splitters and jumpers.

Step 58: Add New Passive Equipment Definition

Let's now add a definition of a new type of passive device to RF Passive Devices dictionary. To do this:

1. From the menu, select **SPATIALnet > Dictionaries > RF Definitions > Passive Definitions..**. A dialog box will be displayed, listing all the passive device types currently defined.

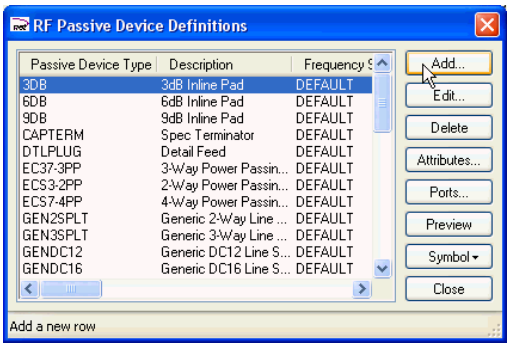


Figure 172 – RF Passive Device Definition

2. Click on the **Add** button. A panel similar to that shown below will be displayed.

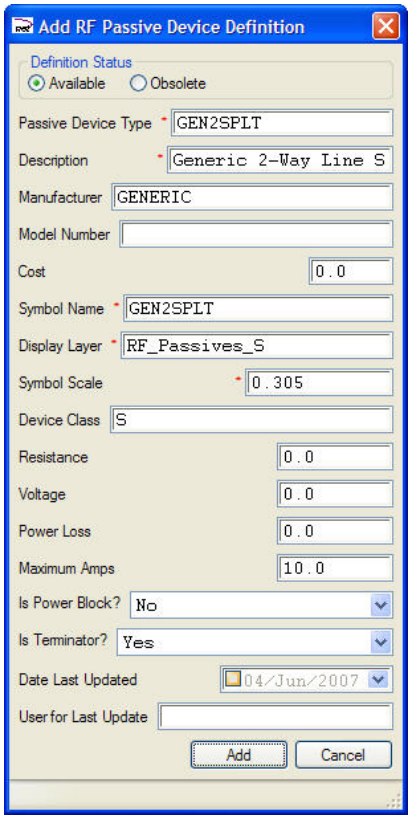


Figure 173 – Add RF Passive Device Type Definition Panel

3. Fill in the values, and click the **Add** Button.
4. In a manner similar to the active device configured in the previous step, once a passive device has been defined, its ports must be configured. To do this:
   - Open the **RF Passive Device Definitions** panel as described in procedure 1 above (if it is not still open).
   - Highlight the RF device just configured, and click the **Ports** button, as shown below.

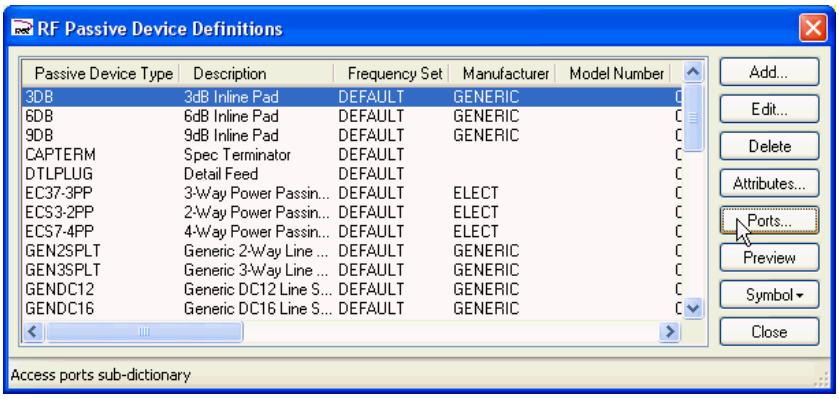


Figure 174 – Ports Button on RF Passive Device Definitions Panel

9. A panel will be displayed with no entries. Click the **Add** button to display the **Add RF Port Information** panel, and add an entry for each port the RF device has, similar to that shown below. Note that in this case, the levels associated with each port are the loss incurred by signals traveling from the input of the device to that port.

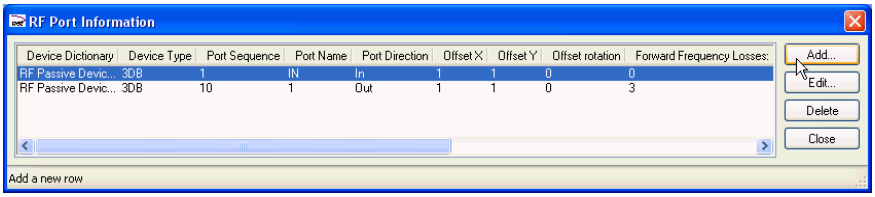


Figure 175 –RF Port Information Panel

Step 59: Add New RF Tap Definition

Let's now add a definition of a new type of RF Tap to RF Tap dictionary. Before adding the tap itself, let's review the way that tap values are configured. SPATIAL*net* keeps a separate dictionary of tap values, an entry of which is then assigned to each tap configuration (this simplifies changing face plates, etc.) To add or update a Tap Value Dictionary entry:

1. From the menu, select **SPATIALnet > Dictionaries > RF Definitions > Tap Value Definitions..**. A dialog box will be displayed, listing all the active tap value definitions. Click the **Add** button. A panel will be displayed, similar to that shown below.

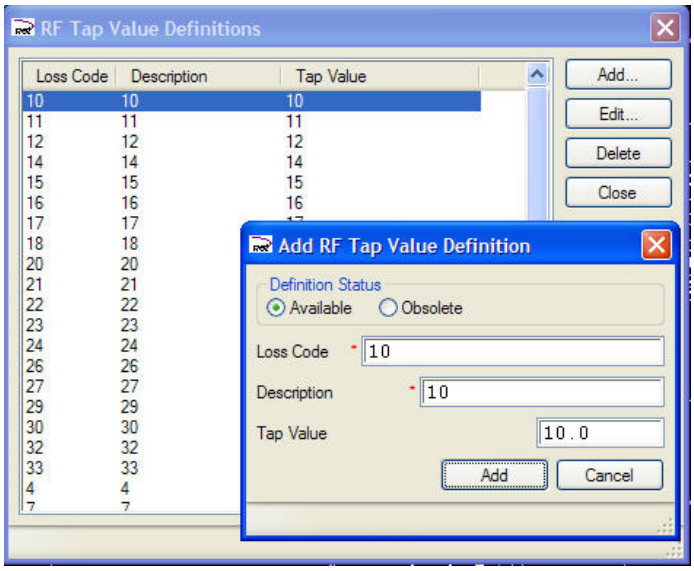


Figure 176 – Tap Value Definitions

2. To define a new tap, select **SPATIALnet > Dictionaries > RF Definitions > Tap Definitions..**. A dialog box will be displayed, listing all the taps currently defined.
3. Click on the **Add** button and fill in the field values, similar to that shown below.

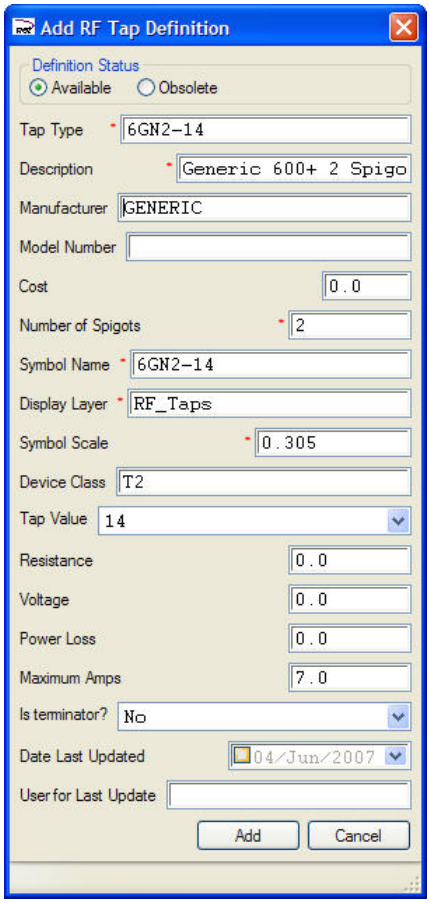


Figure 177 – Tap Definition

4. When the values for each field have been entered, click the **Ports** button. A dialog box will be displayed with no entries. Click the **Add** button and add an entry for each port the tap has, similar to the panel shown below.

Note that the ports on the tap refer to the tap insertion loss – loss to tap spigots is a function of the tap value.

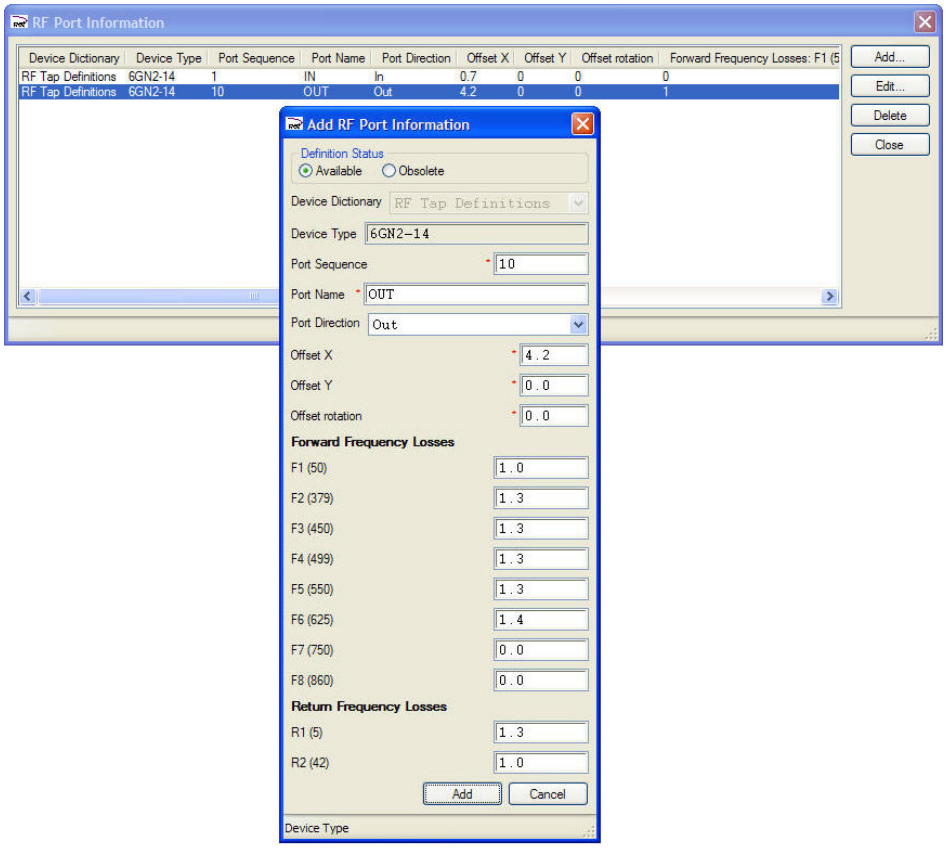


Figure 178 – Tap Port Definition

Step 60: Add New Device Component Definition

Let's now add a definition of a new type of Active Component to RF Active Component dictionary. To do this:

1. From the menu, select **SPATIALnet > Dictionaries > RF Definitions > Device Component Definitions...**. A dialog box will be displayed, listing all the active device component types currently defined.
2. Click on the **Add** button and fill in the field values.

Note: some components such as pads and equalizers only have losses at the given frequencies; other components such as internal splitters will require losses to be specified through creating ports in much the same manner as on active and passive devices.

Step 61: Create New Design Profile and Assign Equipment and Frequencies

The RF equipment dictionaries contain the specifications for all RF devices used anywhere in the system. If all of these were displayed each time you wished to pick a plant item to add to the database, the equipment lists could be unmanageably long. In addition, it is sometimes necessary to design and manage different parts of the system utilizing different equipment and different frequencies.

To remedy this, a Design Profile can be created and used to specify the subset of equipment available in the job, to nominate which design frequencies are to be used, to specify which frequencies will be used to fill in annotation values, to specify maximum amplifier and tap cascades, and to specify tap output parameters.

To create a new Design Profile:

1. From the menu, select **SPATIALnet > Dictionaries > Other Definitions > RF Design Profile Definitions..**. A dialog box will be displayed, listing all the active design profiles currently defined.

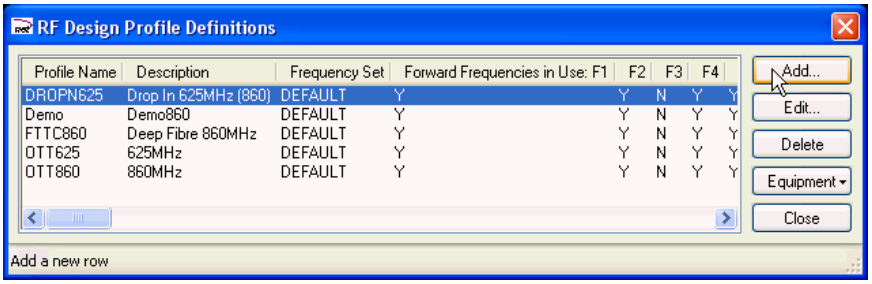


Figure 179 – RF Design Profile Definitions Panel

2. Click on the **Add** button and dialog similar to the following will be displayed:

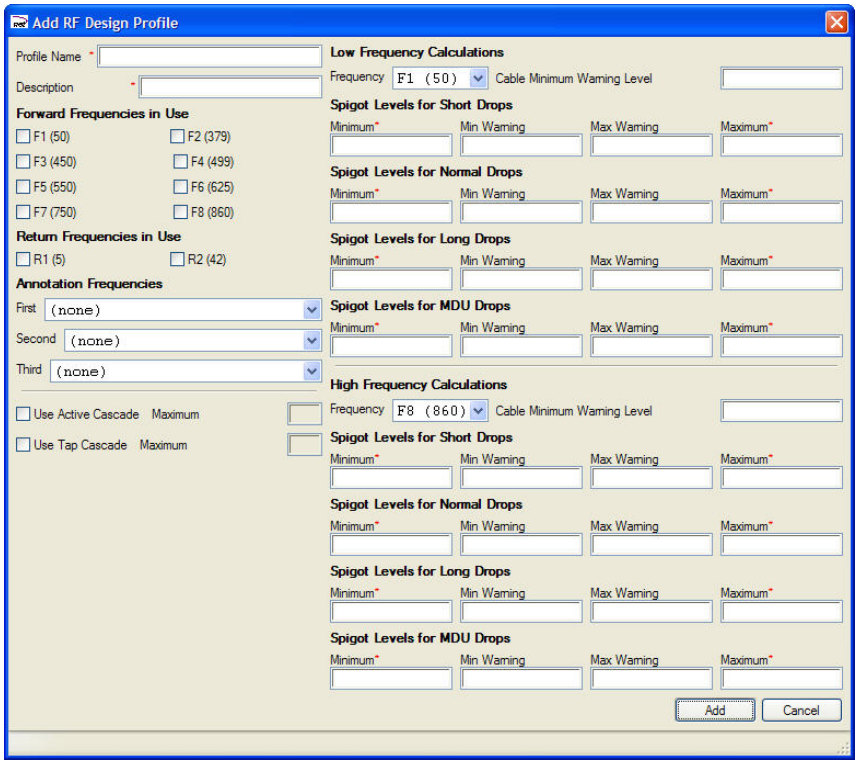


Figure 180 – Add RF Design Profile Panel

3. Fill in the field values, enter a name for the new design profile and select which of the design frequencies will be used in that profile. Click the **Add** button.
4. Once the design profile has been created, select it in the list and click on the **Equipment** button category that will be available in the Design profile. A drop-down panel will appear displaying the equipment list.

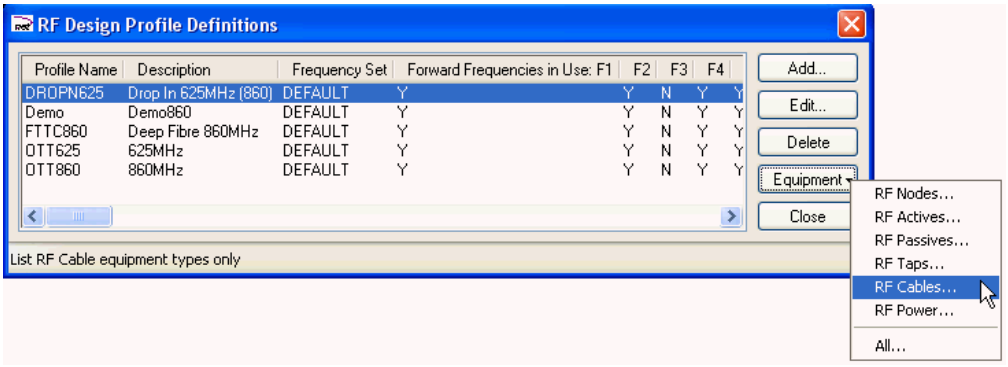


Figure 181 – RF Design Profile Definitions Panel

5. Select the equipment type from the drop-down menu, and click. A panel listing the RF design profile equipment will appear. Highlight the appropriate equipment type from the list, and click **Add**.

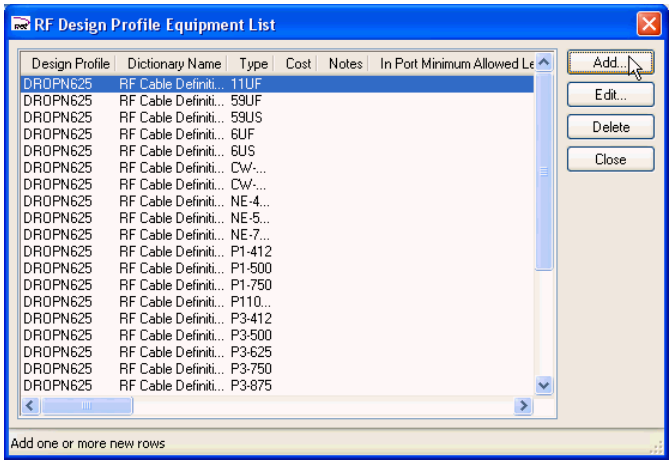


Figure 182 – RF Design Profile Equipment List Panel

6. The relevant RF equipment definitions panel should now be displayed. Highlight the definition to be added to the design profile and click **Select**.

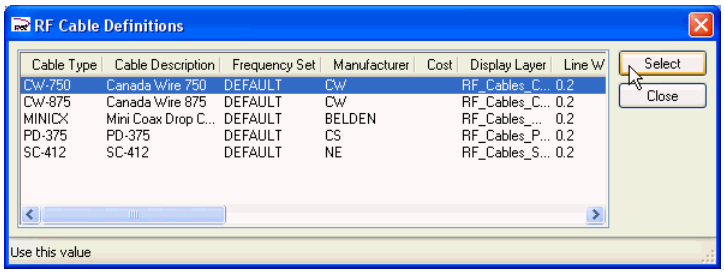


Figure 183 –RF Cable Definitions Panel

The selected definition should now be visible in the **RF Design Profile Equipment List** panel.

Step 62: Assign Design Profile to Job

Let's now assign the RF Design profile we just created to the job we are working on. To do this,

1. Run the menu command **SPATIALnet > Administration > Change Profile of Current Job…** The **Change of Profiles for Job** panel will be displayed.

Figure 184 – Change Profiles for Job Panel

2. Select the desired profile from the drop down list.

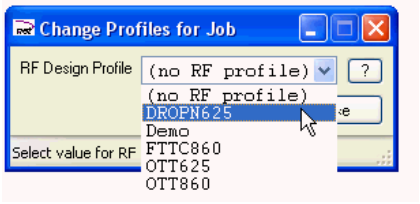


Figure 185 – Change Profiles for Job Panel

3. Click **Apply** and close the **Change of Profiles for Job** panel.

Step 63: Add RF Module to Existing HFC Node

Let's now begin adding the RF plant itself, beginning with the HFC Node. In SPATIAL*net*, nodes are modeled as a housing (or Site) into which RF modules and optical modules are placed. These can then be connected, to connect RF legs with optical transmitters and receivers, or to couple optical ports together.

There are two ways an RF module can be added to the system:

- Add an RF module to an existing site in which a fiber receiver has already been placed
- Add a new site, and place the RF module directly with in it.

In this step, we'll take the first approach. In the next one, we'll take the second.

To add an RF Node Receiver to an existing node site, we must first find an existing Node (with a fiber module installed in it). To do this:

1. Run the menu command **Fiber > Find By Matching > Node Receiver…** The following panel will be displayed, listing the fiber node modules returned by the search. Use the query filter fields above each column to narrow the search conditions if necessary.

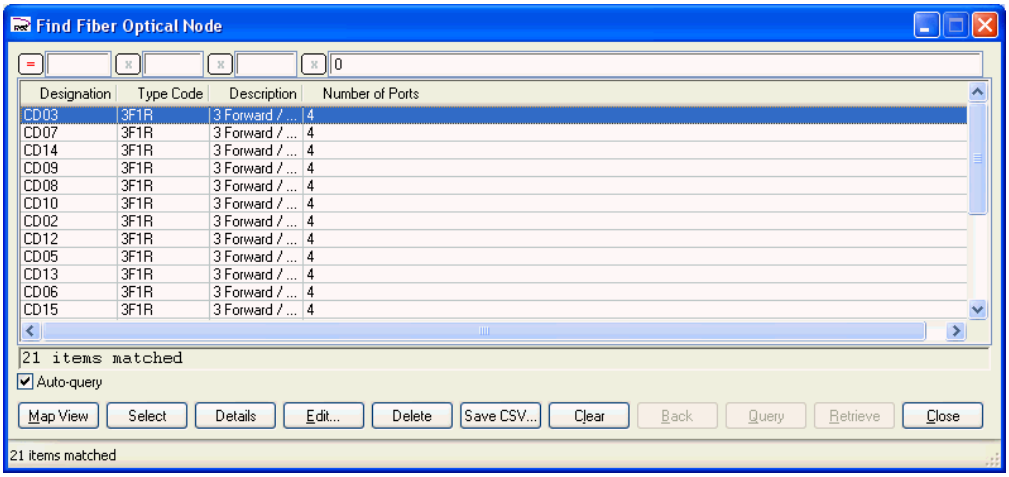


Figure 186 – Find Fiber Optical Node panel

2. Select the optical node you wish to view in the result list and click on the **Map View** button. The system will zoom the map to the selected node's location.
3. Click on the chosen Node in the map to select it and then click on the **Details** button (on the **General** toolbar) to display the node's details. A box similar to the one shown below should appear.

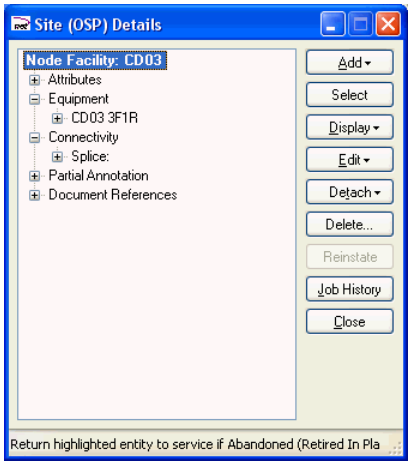


Figure 187 – Site (PSP) Details panel

Note that the Node Site currently contains a fiber splice and an optical receiver/transmitter. Let's now add the RF module to this site.

4. Make sure the Node Site is selected (shown in Bold in the **Site (OSP) Details** panel as above. If it is not, click on it in the Details panel and click the **Select** button).
5. From the **RF Tools** toolbar, click the **Add Node** button, as shown below.

6. A panel similar to that shown below will be displayed. Select the type of Node to add, fill in the remainder of the fields similar to those shown below.

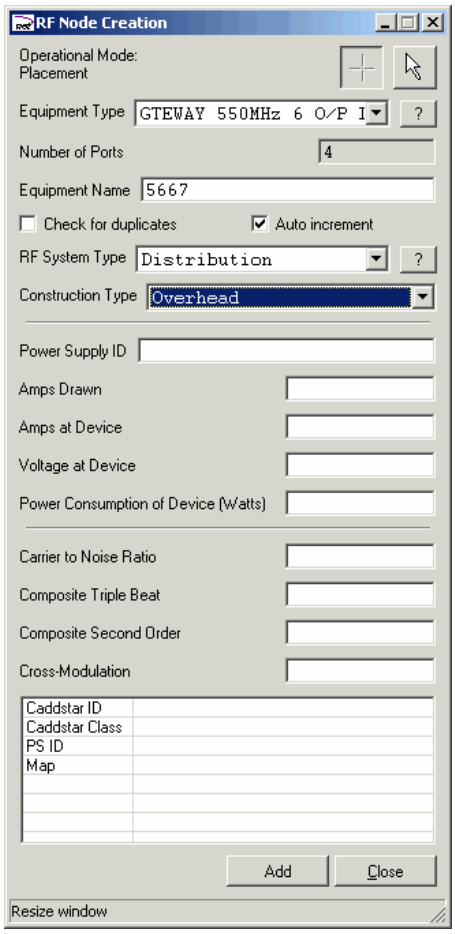


Figure 188 – RF Node Creation Panel

7. Click the Add button to add the RF module to the existing node. The details panel will show this as illustrated below.

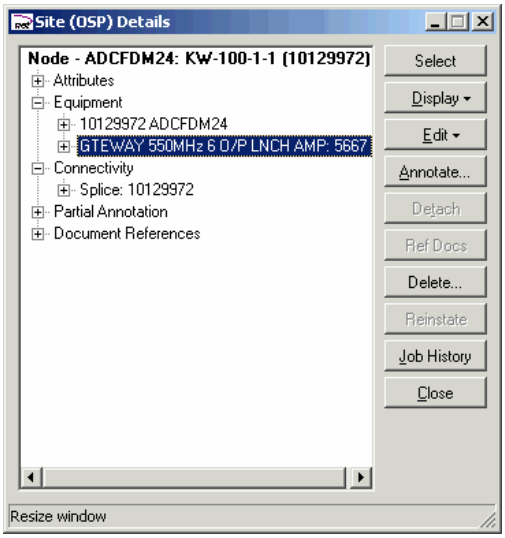


Figure 189 – Site (OSP) Details Panel

Step 64: Add RF Node Directly to Map

In the previous step, we added an RF module to an existing HFC node housing that already contains a fiber receiver/transmitter. In this step, we shall add a new node directly to the map, where none existed previously, and automatically place an RF module in it.

To do this,

1. Zoom the map to the location at which you wish to create the new HFC node.
2. Make sure there is no currently selected entity (otherwise, the system may try to place the new RF node as a module inside that entity). To do this, you can clear the selection on the **General** toolbar.
3. From the **RF Tools** toolbar, click the **Add Node** button, as shown below.

4. A panel similar to that shown below will be displayed.

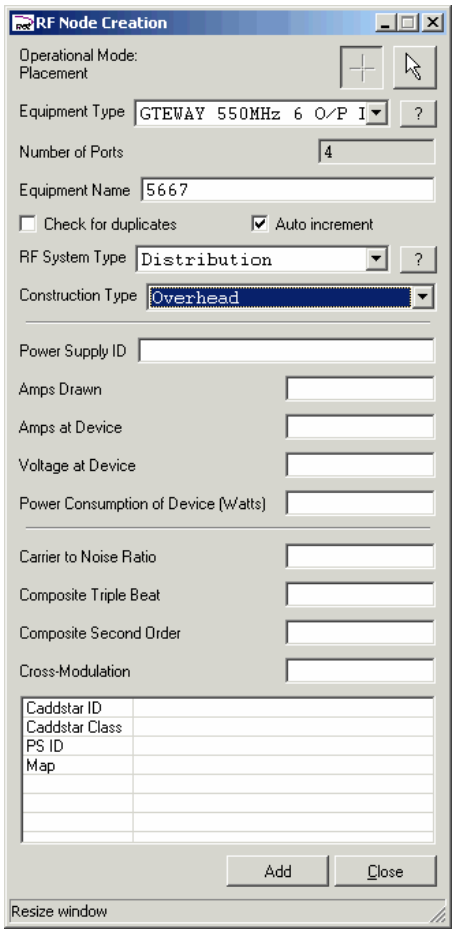


Figure 190 - RF Node Creation Panel

5. Click on the AutoCAD map window once to activate it. Click again at the location at which you wish to place the new node. A new node will be created at that location.

We will now add the fiber module into the new node.

6. Select the new node housing by clicking on it in the map.
7. Click on the **Add Node Receiver** button on the **Fiber** toolbar to add a new optical module to the node. A panel similar to that shown below will be displayed.

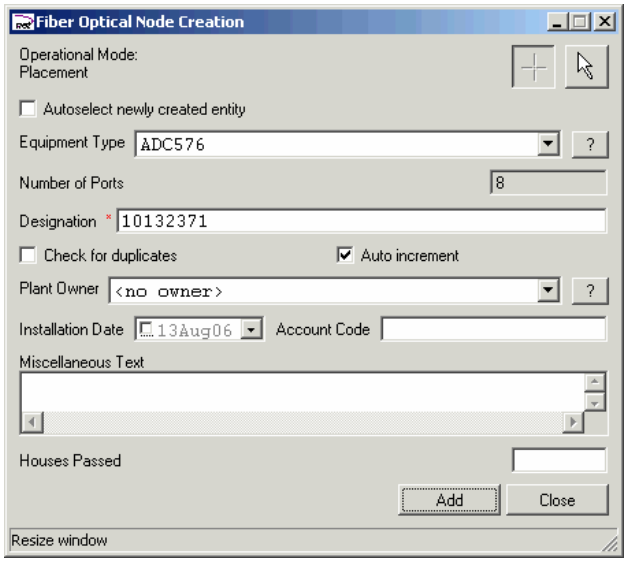


Figure 191 – Fiber Optical Node Creation Panel

8. Click the **Add** button. This will add the fiber optic module to the selected node housing.

Step 65: Connect Optical and RF Ports in HFC Node

Once the optical and RF components have been installed in the node housing, they can be connected together, creating a traceable circuit from the appropriate fiber ports in the node to the RF outputs associated with them. This requires the creation of an "interface table" for the housing. Once created, the ports which are to be connected are assigned to the interface, and the connections (or port mappings) are made.

To create the interface at the node site:

1. Select the node and open its Details panel.
2. Click on the entry showing the RF module and click the **Edit** button, as shown below.

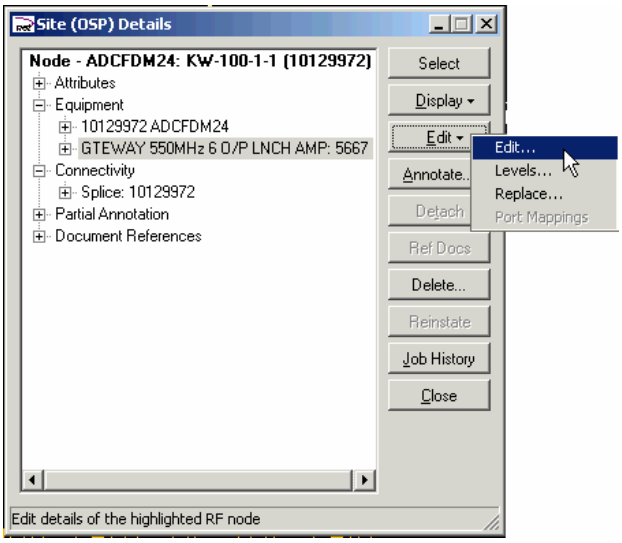


Figure 192 – Site (OSP) Details Panel

3. A panel similar to that shown will be displayed. Click on the **Interface** field, as shown below.

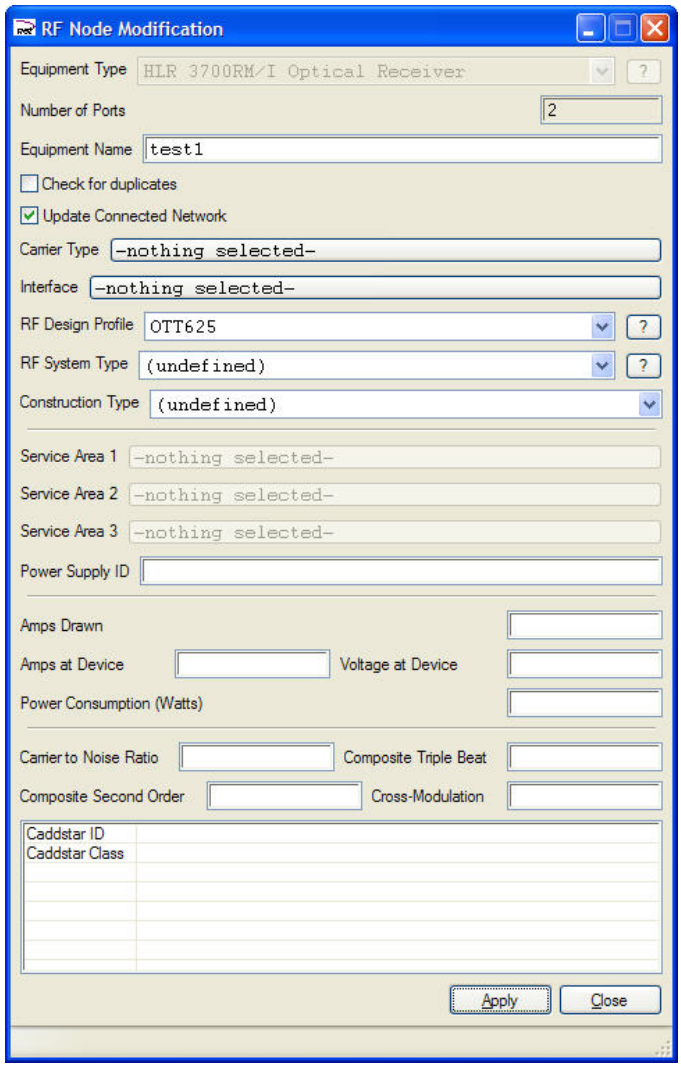


Figure 193 – RF Node Modification Panel

4. A pop-up will be displayed. Click on the **Select Current Site** option.
5. A panel similar to that shown below will be displayed. Click the **New** button to create the new interface. In the **Create Interface** panel, give it the same name as the Node Housing. Also, ensure that the Instance Based Mapping option is checked in the panel, as shown below.

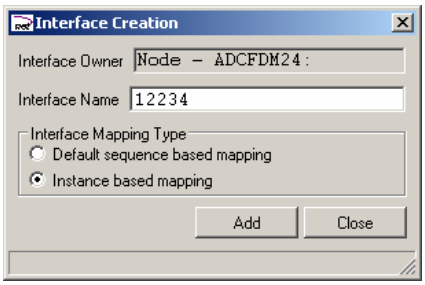


Figure 194 – Interface Creation Panel

6. Click the **Add** button to create the interface.

This completes the first part of the process. The HFC Node how has an interface table associated with it which will allow ports from the various modules in the node to be mapped together. Because we created the interface from the RF module, that module is already associated with the interface table.

The next part of the process requires that we associate the fiber ports with the interface table. To do this:

7. Select the fiber module in the HFC node details panel and expand the branch under it. Click on the **Port Bank** entry and click the **Edit** button, as shown below.

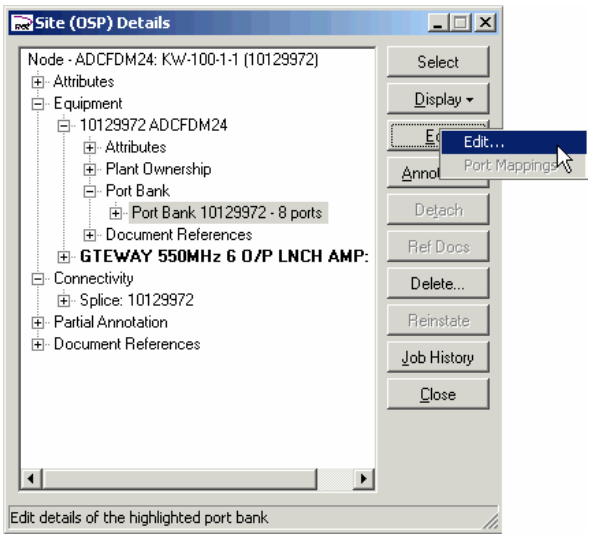


Figure 195 – Edit Optical Port Banks

8. A panel allowing the port bank to be associated with the interface we created above will be displayed. To do this, click on the **Interface** button and select the interface from the pop-up, as shown below.

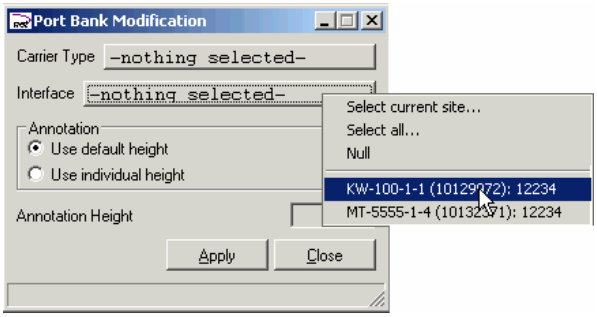


Figure 196 – Assign Fiber Optic ports to interface

9. Click the **Apply** button to save the change.
10. To complete the final step for mapping the RF ports to fiber optic ports, now that both the fiber and RF ports are present in the interface table, we can specify which ports are connected together.
11. To do this, select either the RF module, or the Fiber Port Bank in the **Site (OSP) Details** panel again, click the **Edit** button, and select **Port Mappings** from the pop-up, as shown below.

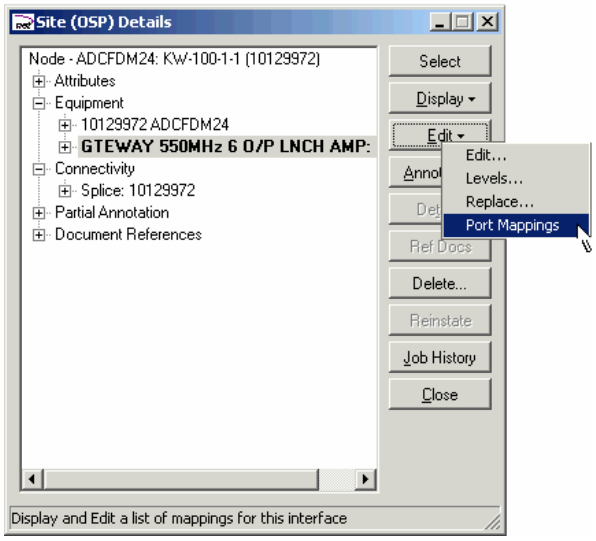


Figure 197 – Edit Port Mappings

12. When the Port Mappings panel is displayed, for each connection to be made:

- Select the device from which the ports are to be mapped on the left of the panel
- Selected source port on the left of the panel,
- Select the device to which the ports are to be mapped on the right of the panel

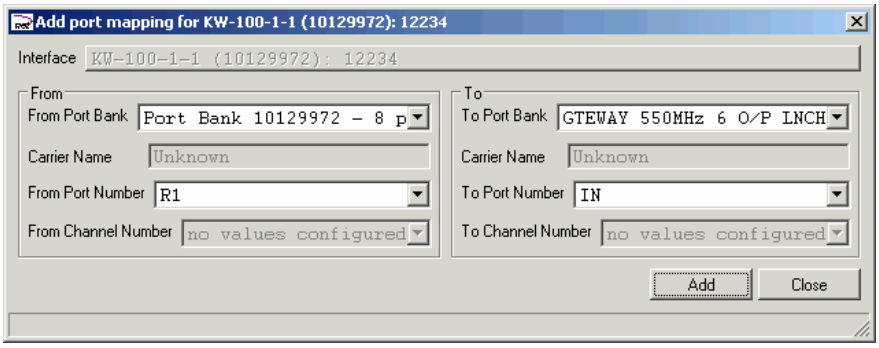


Figure 198 – Create Port Mappings

Step 66: Connect RF Cable to RF Node

Note: The following 12 steps highlight the functionality used to place "as-built" plant. To learn how to do RF design in SPATIALnet please see the steps beginning with Step 78 : Designing RF Plant – Cable and Taps***.***

Let's now connect a cable to the RF node receiver we just placed and configured. To do this

1. Select the Node to which you wish to connect the new RF cable.
2. Click the **Add Cable** button on the **RF Tools** toolbar, as shown below.

3. Specify the details of the RF cable to be added by filling in the fields shown in the **RF Cable Creation** dialog box.
4. Click the **Add** button. If no specific ports are selected, a list of available ports to connect the cable to will be displayed. Select the port to which you wish to connect the cable.
5. Capture the path of the cable as required.

Step 67: Add RF Cable in Free Space

As an alternative to attaching an RF cable to an existing device, a cable may be drawn in free space by clicking at the desired start location and drawing the cable line along the desired path. The steps are similar to connecting a cable to an existing device, as described below:

1. Click the **Add Cable** button on the **RF Tools** toolbar, as shown below.

2. Specify the details of the RF cable to be added by filling in the fields shown in the **RF Cable Creation** dialog box.
3. Click on the AutoCAD window to activate it. Then click at the location in space at which you wish to begin the cable.
4. Capture the path of the cable as required.

Step 68: Add RF Power Supply

Let us now add a power supply associated with the node (although power supplies do not need to be associated with nodes). To do this:

1. Click the **Add Power Supply** button on the **RF Tools** toolbar, as shown below:

2. The **RF Power Supply Creation** panel, similar to that displayed below should appear. Fill in the details.

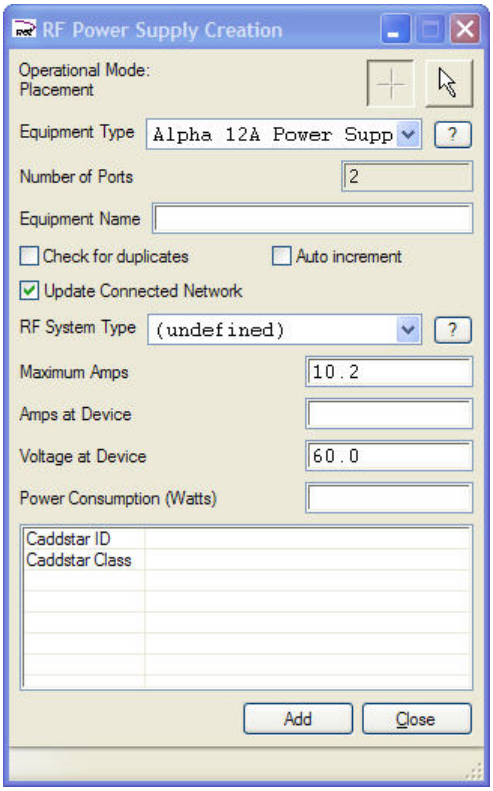


Figure 199 – RF Power Supply Creation Panel

3. Click on the AutoCAD map window once to activate it. Then click at the location at which the power supply is to be placed. This will add the new power supply to the system.

Step 69: Add RF Power Inserter

Let's now add a Power Inserter to the network, connect it to the power supply we placed in the previous step, and reconnect the cables so that it is connected to the network we have placed so far.

A Power Inserter is classed as a Passive RF device, so to add one:

1. Click the **Add Passive** button on the **RF Tools** toolbar, as shown below.

2. The **RF Passive Device Creation** panel is shown. Select the Lindsay Power Inserter from the **Equipment Type** pulldown menu.

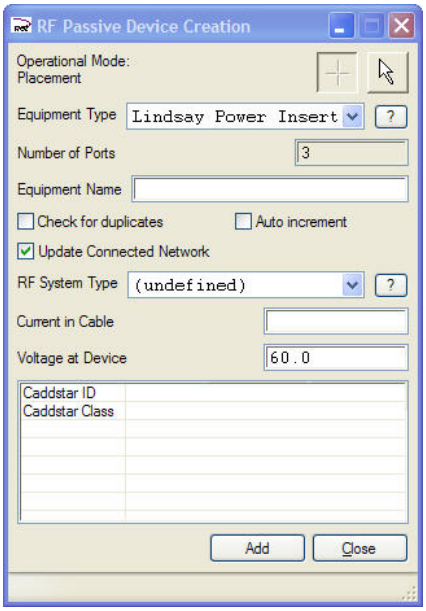


Figure 200 – RF Passive Device Creation Panel

3. Click once on the AutoCAD window to activate it, then click on the map at the location at which the power inserter is to be placed.
4. Once the power inserter has been placed, click on it. Each of the ports will be shown as a circle, as shown below.

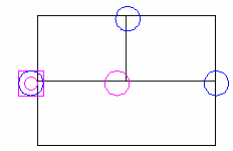


Figure 201 – Location of Ports shown on Power Inserter

5. When a new device or cable is to be connected to an existing device or cable, the port to be connected should be selected by clicking on it. Click on the Power Supply port (the topmost one) to indicate that it will be the starting point for the new cable.
6. Click the **Add Cable** button on the **RF Tools** toolbar as shown below.

7. Select the RF cable to add, and draw the path from the power inserter to the power supply.
8. To connect the cable to the power supply, right click on the power supply symbol and select the power supply name from the pop-up, as shown below.

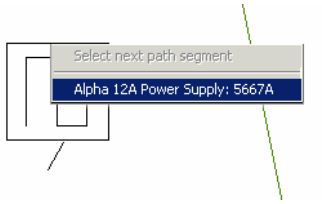


Figure 202 – Connect cable to Power Inserter

Step 70: Add RF Active Device

To add an RF Active Device:

1. Click the **Add Active** button on the **RF Tools** toolbar, as shown below.

This will display the **RF Active Device Creation** dialog box.

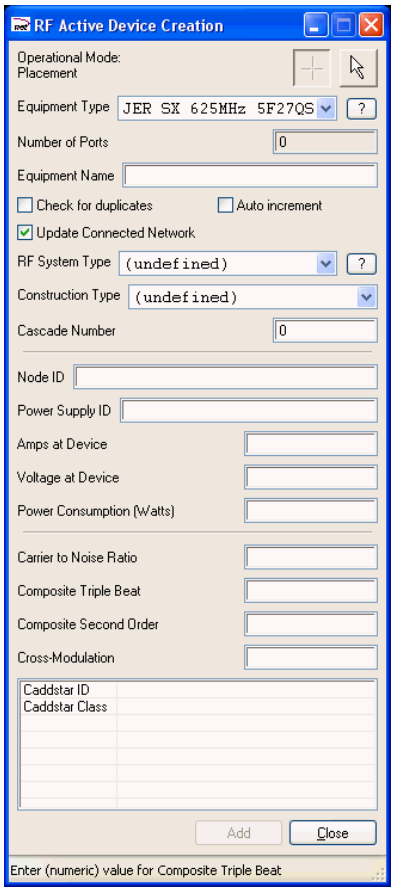


Figure 203 – RF Active Device Creation Panel

2. Select the equipment, system and construction type from the pulldown menus, and fill in the details of the RF Active type in the panel.
3. Click the location on the screen at which you wish to place the new Device.

Step 71: Add RF Passive Device

To add an RF Passive Device:

1. Click the **Add Passive** button on the **RF Tools** toolbar, as shown below.

This will display the **RF Passive Device Creation** panel.

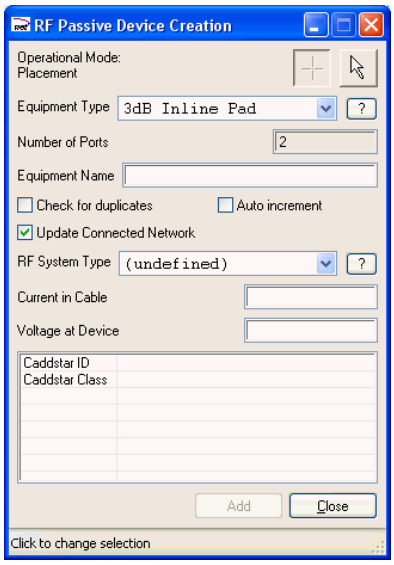


Figure 204 – RF Passive Device Creation Panel

2. Fill in the details of the RF node and click the location on the screen at which you wish to place the new Device.

Step 72: Add RF Tap Device

To add an RF Tap Device:

1. Click the **Add Tap** button on the **RF Tools** toolbar, as shown below.

This will display the **RF Tap Creation** dialog box.

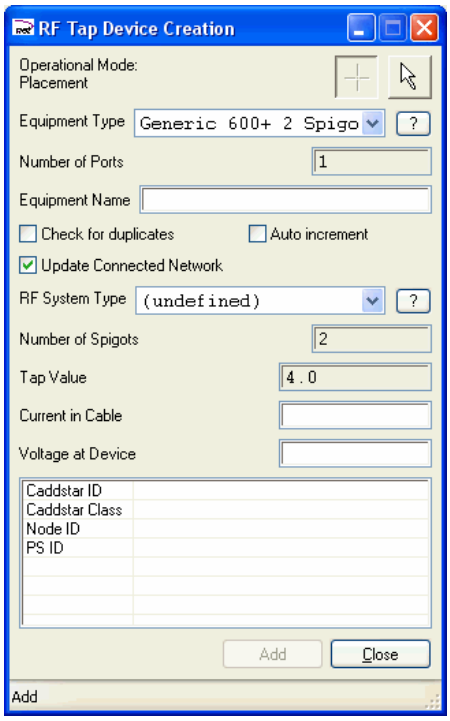


Figure 205 – RF Tap Device Creation Panel

2. Fill in the details of the RF Tap, and click the location on the screen at which you wish to place the new Device.

Step 73: Associate Taps with Addresses

The last step in creating the infrastructure needed to connect addresses to the network is to create drops, that is, assign tap ports to addresses. This process is also significantly automated, via a "wizard" that does most of the work for you.

To run the drop wizard:

1. Select a tap box by clicking on it in the map.
2. Click the **Drop Tool** button on the **Network Tools** toolbar.

3. A panel similar to that shown in Figure 206, below will be displayed.
4. The drop tool always operates on a single tap. While you have a single tap box selected, the tool works as follows:
    - Click on any support structure (pole, pedestal, vault, etc.) and any drops to be created will be run from that structure to the address they serve.
    - Click on any group of addresses. These will appear in the address list window and be assigned to the next sequence of spare ports in the tap box. If there are no spare ports, the system will display an error message.
    - Click on a strand/trench footage. This will be added to the length of drop cable required when the structure from which a drop is connected to the address is not the structure to which the tap is attached.
5. When you click the **Apply** button, drops will automatically be created between any new addresses assigned to the tap box, from the currently selected structure. Each new address will be assigned to the next available port in the tap box, while there are still unused ports available. Addresses already connected to the tap box will be shown in the address list, but will not be affected when the **Apply** button is clicked.

The results of running this process on the area we are building are shown below.

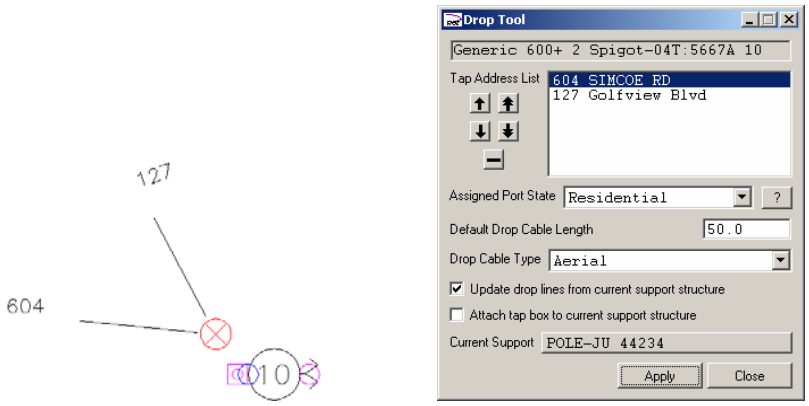


Figure 206 – Drop Tool showing auto-created drops and assignments of addresses to tap ports.

The automatic assignment of the addresses to the ports in the tap box can be seen by showing the Details panel of a selected tap.

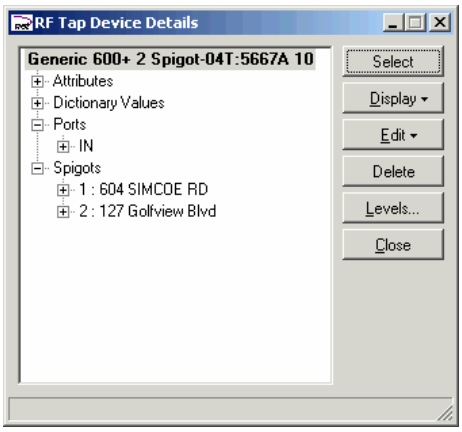


Figure 207 – Addresses assigned to Ports after Drop Tool run.

Step 74: Edit Signal Levels

To edit the signal levels on any RF device:

1. Select that device and open its **Details** panel.
2. Click the Levels button on the Details panel. A panel similar to that shown below will be displayed.
3.

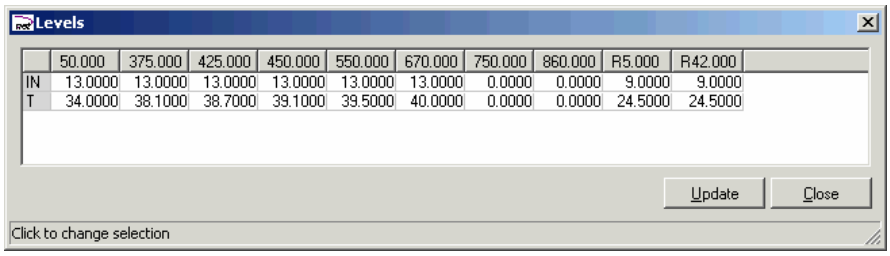


Figure 208 – RF Levels at a device

4. To update any of the levels at any of the ports, double-click on the desired cell and change the value.
5. Click **Update** to save the changes.

Step 75: Reconnect Cables or Devices

When a cable or device is connected to another device, you can disconnect or reconnect the cable or device by clicking on the manipulator representing the cable end, or device port.

If you drag the manipulator onto a new device port, the cable or device will be reconnected to that port.

Step 76: Connect RF Equipment Directly to Other RF Equipment

It is possible to "chain" RF devices together by connecting their ports in sequence without intervening cables. To do this, select the port to which the new device is to be added prior to adding the device.

The new device will be added and connected directly to the selected port.

Step 77: Creating an RF Cable Leg with Inline Equipment

Up to now, we have been placing RF devices, and then connecting them with cables. However, it is possible to place RF devices "inline" as a leg of RF is being entered into the system. This is done as a variation on the "Create Cable" process. To do this,

1. Begin the "Create Cable" process using either method described in Step 66 or Step 67.

2. When you reach a location at which a device is to be placed (e.g. a tap, a passive or an active device), right click during cable creation. A pop-up menu will be displayed, as shown below.

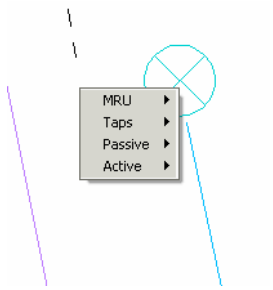


Figure 209 – RF Levels at a device

3. Select the category of device you wish to place from the first pop-up, and then the particular device from the follow-on pop-up. Note that only equipment assigned to the current profile will be displayed.
4. When you finish drafting the cable path, any in-line devices will be automatically created along the cable, with their ports wired in the correct direction.

Step 78: Designing RF Plant – Cable and Taps

SPATIALnet has the capability to do "on the fly" RF design as the cables and equipment are placed. The system will automatically suggest tap values, keep track of signal attenuation / amplification, calculate pads and equalizers at amplifiers, and perform power calculations. If you've gone through the steps above placing as-built RF plant, you will notice that actually designing the plant is not much different – the major exception being it is necessary to capture the cable distance between devices in order to perform the design calculations.

Let's begin by starting a design from the RF Node we placed in step 10. We'll begin by attaching a cable to one of the available output ports on the node. *Tip: it can be helpful to have a relatively large section of landbase and route refreshed then zoom in closer to add equipment so that it is not necessary to refresh every time you need to pan to add design.*

1. Click on the Node symbol to make sure it is selected. There are two ways to connect to a specific port on an RF device: select the symbol, press the add cable button and select from the available port dialog that appears; or select the symbol, select a specific port (you'll know it's selected because it changes

colors), and then press the add cable button. Now, select Add RF Cable from the RF toolbar:

2. Select a cable type and a RF system type, enter other desired values. In most cases if you have strand distances available to you for design purposes, you'll also want the Design Length Control set to "Estimate From Strand", but the system has the flexibility to manually enter distances or estimate from the model length (the AutoCAD determined length).
3. Choose the port you'd like to connect to, if you didn't select a specific port, and press Add.
4. You've now begun the design process! Note the addition of the Design Details to the standard line capture control dialog box:

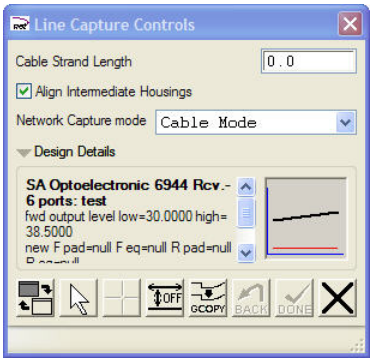


Figure 210 – Line Capture Controls with Design Details

The design details show the current signal levels at the fwd high and low frequencies as well as a slope graph that can help to track the signal levels and slope as you design. Note, that the black line in the graph displays the slope and will flatten out as you design farther away from and active or other equalizing device. The design details can be collapsed by pressing the arrow next to the title "Design Details". Click the Align Intermediate Housings checkbox to rotate connected devices without cable in between and set the Network Capture mode to Cable Mode if there will be cable in between devices or Cluster mode if there will be multiple pieces of equipment placed with no cable between.

5. At this point, you have some design decisions to make. For instance, is there a housecount at the same location as the node and do you want to feed the tap from this port? Or, if you're designing a Trunk section of the network you may not want to place any taps at this time. Let's assume this is a feeder leg and you do want the tap on this port – right click on the housecount or pole symbol and a dialog similar to following will appear:

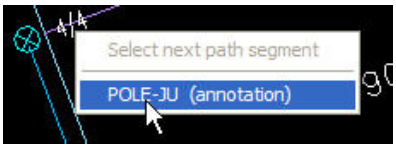


Figure 203 – Select pole annotation

Select the pole / ped annotation to register the housecount and the system will only display taps with the correct number of ports in the Auto suggest function.

6. Next right click on the map where you'd like to insert the tap and hold the curser over the Auto Suggest Taps option. A list of taps that meet the design criteria is then displayed, select the desired tap and the section of cable and equipment will be displayed on the screen in a temporary fashion (when the design leg is complete and you click done all design entities will then be displayed properly).

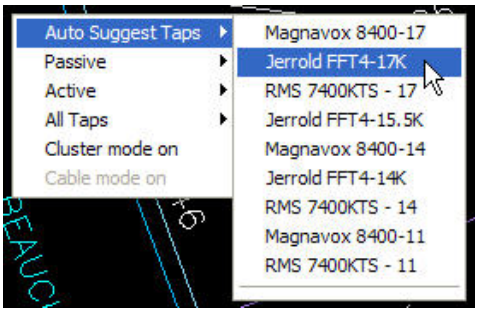


Figure 204 – Select Auto Suggested Tap

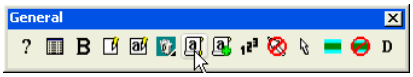
7. Let's now continue placing taps along this run. Right click on the next strand distance and a box similar to the one displayed when selecting housecounts will appear. Select the "Aerial Footage" distance (or other distance attribute) to enter the actual strand distance into the system for this section of cable.

   Note: When placing Trunk or Express cable it will be necessary to right click on all distances along a run for the system to perform the design calculations correctly.

8. Right click on the next housecount, right click to place an auto-suggested tap, and continue in this fashion. Note, that as you add cable and taps, the slope graph begins to flatten out and the signal levels decrease due to the cable attenuation and tap insertion losses. At some point, the Auto Suggest Tap options will have a gray line between two groups of taps: the upper group is non-self terminating taps and the lower group consists only of self terminating taps. Also, at some point the Auto Suggest Taps option will be completely blank, this is because there is not enough signal to place a tap that is within specification. It's now time to amplify the signal. For this exercise, however, let's place a couple of taps, then terminate and place the end of line annotation block.

9. If you've selected a self-terminating tap as your last tap, the system will automatically end the Cable Capture control and place the equipment. If you've selected a non-self terminating tap as the last one, you'll need to switch to cluster mode and select a terminator from the passives menu and then press done.

10. The final step is to place the end of line annotation block. To do this simply select the tap on the screen, press the **Annotate Selected Entity** icon on the general toolbar:



Then select the appropriate end of line annotation block from the drop down list in the dialog that appears and click on the screen where you want it inserted. Note that all levels are filled in automatically.

Step 79: Designing RF Plant – Splitters, Taps, and Amplifiers

In this exercise we'll use the same methods described in Step 24, but we'll go a little further and add an amplifier and maybe a splitter or two.

1. Let's once again start at the node placed in Step 10 and we'll design off one of the other available ports. Before starting, ensure that you have a section of landbase and strand or trench large enough to need at least 2 splits and at least 1 line extender.
2. Proceed now as in Step 24 – select the node, hit the Add RF cable button and begin to design.
3. When you come to the first split point right click and pick passive instead of Auto Suggest tap, then select the split you would like to use. Note that if there is also a tap at that location, you may need to switch into Cluster Mode.
4. When you are running low on signal, right click and select and amplifier from the Active menu.

It is encouraged that you take your time on this step and attempt to design numerous different scenarios so that you become accustomed to the tools and comfortable with their use.

Step 80: Designing RF Plant – Powering

In this section we'll add a new Power Supply to power the plant we've just built. Depending on your corporate policies it may or may not be necessary to place a power block everyplace the power needs to be stopped. For instance, some companies require a power block on output of an amplifier if there are no other amplifiers cascaded afterwards; other companies simply need an "IN" flag in the amplifier annotation block.

1. If company policies dictate, place power stops on the outputs of all "last in cascade amplifiers. Select the output port at the amplifier, then click on the **Add Passive** button on the RF toolbar:

2. Select a power block from the drop down list of available passive devices and click **Add**.
3. Next, determine the desired location for the power inserter. Like a power block, select the device port where you'd like to place the power inserter, click on the **Add Passive** on the RF toolbar and select a power inserter.

   Note: Typically power inserters are defined with 3 ports – an in port, an out port, and a power port. The insertion loss is calculated between the in and out port while the power port is the port actually connected to the power supply.

4. Once the power inserter is in place it's time to place the power supply and hook it up to the network. Select **Add Power Supply** from the **RF Tools** toolbar:

   Select the desired power supply and click **Add**, then connect the power port on the power inserter to the power supply with the desired cable type.

5. After placing the power supply and ensuring that power blocks are in place, click on the **RF Power Tool** button:

   A trace result window similar to the one pictured below will appear showing equipment that is powered by the selected power supply and showing all devices (if any) with warning messages.

Figure 205 – RF Power Trace Results

Step 81: Designing RF Plant – RF Network Reporting

At this point, the network should be designed and working. To be sure, you can select a leg at a time and verify through the use of the RF Network Reporting tool. Press the RF Network Reporting button on the RF toolbar:

A window similar to the following will be displayed:

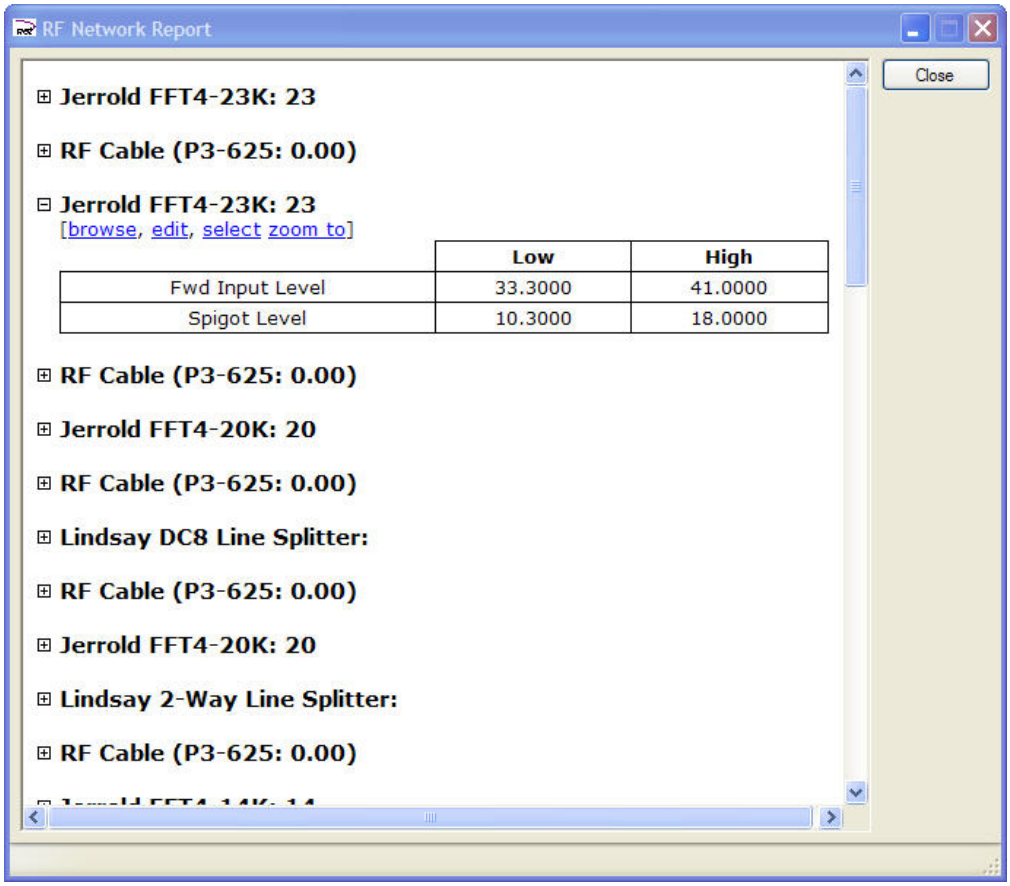


Figure 206 – RF Network Reporting

The information contained in this report will provide a good amount of detail about the state of your network, allow you to view equipment details without having to navigate to each piece of equipment and allow you to zoom directly to certain devices or areas.

Step 82: MDU Detail Drawings

MDUs (Multiple Dwelling Units) pose a particular challenge in outside plant mapping, since they require a lot of design detail to be represented in a small space. Often, they also include a significant vertical layout component, such as in high-rise apartment buildings.

To support MDUs, SPATIALnet includes the ability to create MDU detail drawings. These are separate drawing spaces “internal” to the MDU which allow internal diagrams and plans of MDUs to be stored in the database and connected to the network.

To create an MDU detail drawing:

1. Create an MDU by clicking the Create Buildings and Addresses button on the Landbase toolbar, as shown below.

2. Specify the details of the MDU and place it on the map by clicking at the desired location.
3. Select the new MDU by clicking on it.
4. Add a new MDU Detail drawing to the MDU by clicking on the Add MDU Detail drawing button on the MDU Detail Drawing toolbar, as shown below.

5. Specify the name you wish to give to the drawing and click the Add button to save it.
6. To verify that the new detail drawing was added, open the MDU's details panel. The new drawing should be listed similar to that shown below.

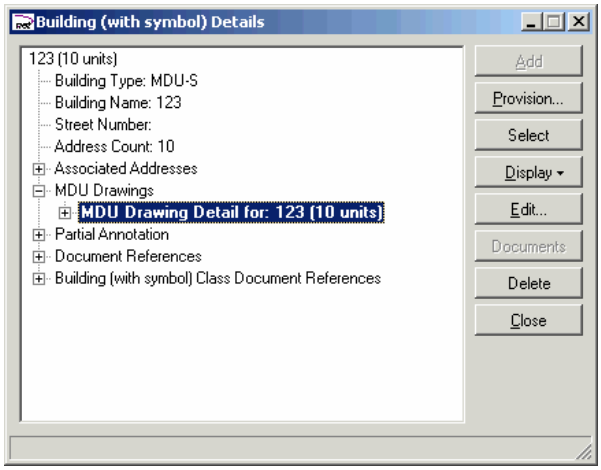


Figure 2117 – Building Details Panel showing MDU Details

7. To load the graphical appearance of the detail drawing, you must first open it in an AutoCAD window and then either draft the details you wish to record, or copy the contents of an existing DWG file into them. The CAD-scrape process for MDU detail drawings works in an identical fashion to the process for outside plant maps.
8. Adding modeled network features to an MDU detail drawing is identical to the process used for entering features into the outside plant maps.

Step 83: MDU Detail Drawings Demarcation Points

To allow the outside plant map representation of the network to connect to the network represented in MDU detail drawings, special symbols called Demarcation Points or “demarks” are used. When an MDU with a detail drawing is placed, a Demark should be placed in the detail drawing, and a matching Demark in the outside plant map. In this situation, the system will assume that the point in the network represented by the outside plant representation of the Demark is the same as the point represented by the Demark in the detail drawing.

To add a Demark to a Detail Drawing:

1. Open the Detail Drawing by selecting the MDU, opening its Details panel, selecting the detail drawing to be shown, and Selecting **Display > View Drawing**, as shown below.

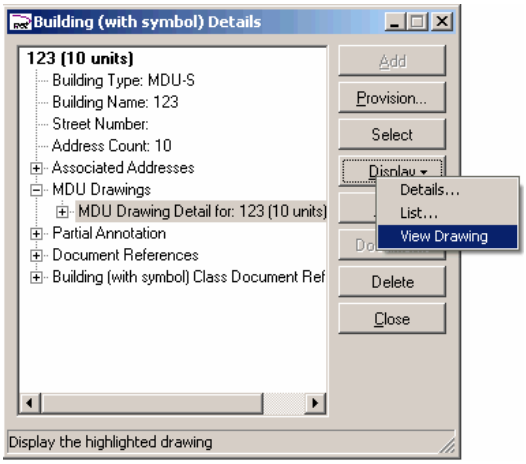


Figure 2128 – Building Details Panel showing procedure for displaying Detail Drawing

2. To place the Demark in the drawing, click the Add MDU Demark button on the MDU toolbar, as shown below.

3. Give the Demark point a name and click on the location at which you wish to place the Demark point.
4. To place the corresponding Demark on the outside plant map, repeat 2 and 3 above, but place the Demark symbol at the location on the outside plant map at which you wish to display it (typically close to the MDU symbol).

5. The details panel of the MDU should display both Demark points, as shown below.

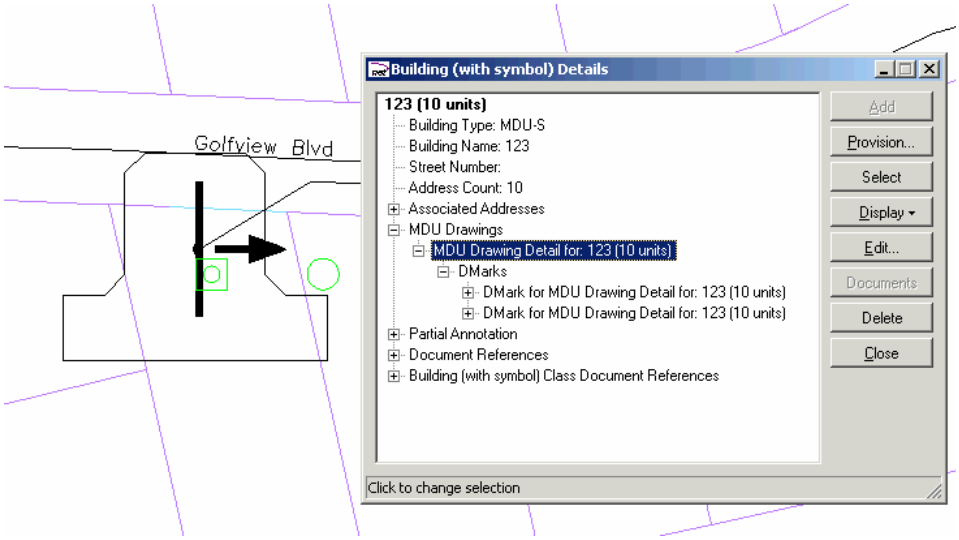


Figure 2139 – Demarks added to MDU, and shown on outside plant map

6. Demarks can be connected to the network in the same way as other plant. That is, when adding a cable, right-click on the Demark to form the connection. The cable connected to either side of matching Demarks will be treated as a single cable by the system.

Chapter 7

Inside Plant

This part of the tutorial walks you through the inside plant engineering and management functions of SPATIAL*net* FM. The tutorial is divided into two sections. The first of these deals only with managing inside plant at the physical level (that is, equipment locations and attributes, connections between equipment, etc.). The second section is more advanced and introduces you to SPATIAL*net* FM's logical network modeling functions. If you are not planning to use SPATIAL*net* FM to manage circuits, CWDM, DWDM or other routing/multiplexing schemes, then you should stop at the end of Section 1.

Section 1. Physical Plant Engineering and Management

In this section's example, you will perform an end-to-end job for fitting out a new network point of presence (POP) with equipment, including storing a floorplan and generating rack elevations and schematic views of the connectivity. You will also add a customer site, and connect this to the POP via some outside plant fiber. We will not go through the details of adding a realistic outside plant network in this part of the tutorial, since that was the subject of Part A. So let's assume that the customer and POP are next door to one another and require only minimal outside plant to connect them. We don't want to dispense with outside plant altogether in this part of the exercise, however, as one of the important topics we'll cover is the interface between the outside and inside plant networks, allowing you to seamlessly trace connectivity and circuits between them.

The sequence of steps is organized so that system configuration is not addressed until the end of the section. This is because inside plant equipment requires more specification information that outside plant, much of this being graphical. So, rather than interrupting the flow by inserting system configuration steps into the natural inside plant workflow, equipment specification is deferred until the last three steps.

When Section 1 is complete, you may wish to work through Section 2, where we define logical network services and route these through the physical plant we created in Section 1.

Step 84: Logging On and Selecting a Job

This step is identical to Step 1 in Part A. Please refer to the steps on page 26.

Step 85: Find a Map Location

To record inside plant, Sites do not have to have a location on a map. You can quite easily use SPATIAL*net* FM to manage sites as "aspatial" entities (that is, objects whose geographical location is not being modeled) and and never enter a single map into the system.

However, in most cases, you will probably want to integrate inside plant management with your outside plant network so let's take that approach here. So, to begin with, we will need a map location at which to place our POP and the customer. If you already have a map in the system, feel free to use that. If you do not, run *: CAD-Scrape Landbase and* Strand from Part A of the tutorial (page 28) to import some reference landbase. Alternatively, just create a map view in empty space as follows:

1. Make a drawing window is open.
2. Run the menu command **SPATIALnet > View > Settings**...
3. Make sure the **View Type** field is set to **All Networks (Seamless)**, and that the **Select Area** check box is clear.
4. Click the **Apply** button. This will create a drawing window with a view onto the database at the coordinates of the original drawing window. Most likely, it will be empty. If it isn't it doesn't matter because we will not be interacting with any existing map objects in the remaining steps.
5. Dismiss the **View Settings** panel by clicking the **Close** button.

Step 86: Add New ISP Site

All inside plant in SPATIAL*net* FM must be contained within a site. So, now that you have logged on and have a map view open, the first real step in this exercise is to add a new site. Before we can add a new site, however, we must ensure that the type of site we wish to add is in the system's Site Dictionary. For Part B of the tutorial, we shall only use site types which are supplied in the default configuration of SPATIAL*net* FM. *: Add New Site* Type in Part A (page 61) walks you through adding a new site type should you wish to review this.

Sites have both outside plant (OSP) and inside plant (ISP) characteristics, so they appear on both the **FM OSP** and **FM ISP** menus. However, the configuration of site types is treated as an OSP function (which is why we covered it in Part A).

Note: Although they refer to the same object, the OSP and ISP menus use slightly different language to refer to a site. Adding a site is found under FM OSP > Add > Facility..., and also under FM ISP > Add > Building... Both menu options run the same command, however.

To add our new Point of Presence and a neighboring customer site:

1. Run the menu command **FM ISP > Add > Building**... (you could equally well run **FM OSP > Add > Facility...** or click the **Add Facility** button on the **Add** toolbar as shown here , since they all do exactly the same thing).

2. Fill in the fields of the Building Creation dialog with the values shown in Figure 214, below (Note: for this exercise, it is important to ensure that the **Capable of modeling inside plant** check box is checked for both buildings)

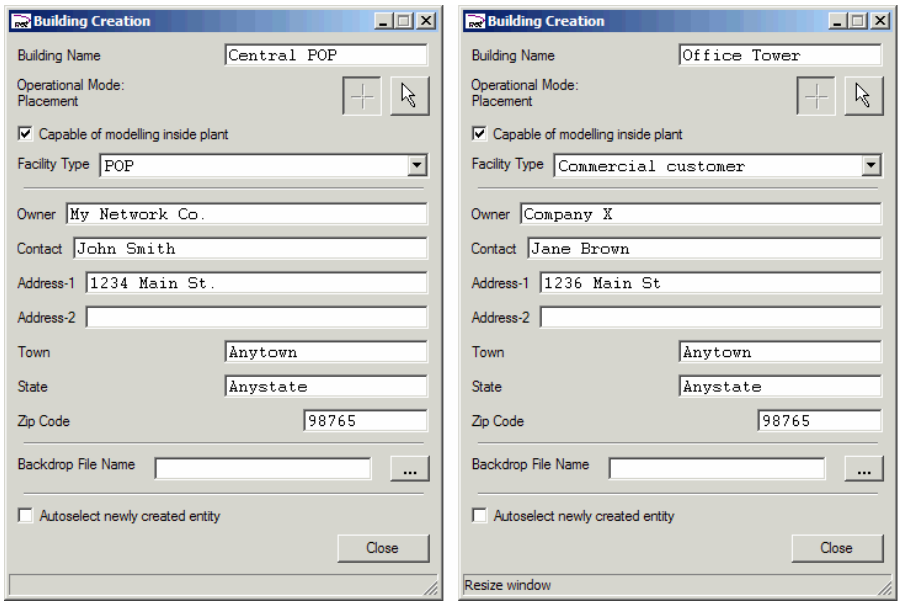


Figure 214 – ISP Building Creation Dialogs for POP (left) and Customer (right)

3. Ensure that the "Placement" cross hair is selected in the **Building Creation** dialog, as shown in Figure 214, above, and also here: 

4. Place the new building on the map by clicking twice in the drawing window (first click to set input focus back to the drawing window, second click to place the new entity).

5. When you have added the POP, repeat items 2 to 4 above for the Customer building. To simplify linking up the two buildings later, place the customer site close to the POP, similar to the example shown below. (Of course, if you would like to run a geographically realistic outside plant network between the two buildings using the steps covered in Part A, then by all means do so!)

Figure 215 – Example locations of POP and Office Tower

6. When you have added both buildings, click the **Close** button shown in Figure 214 to dismiss the **Building Creation** panel.

Note: The relative sizes of the POP and Customer symbols may appear different to that shown in Figure 215, above. If so, you may change the symbol scale of either site type from the FM OSP > System Configuration > Site Definitions... menu by editing the Symbol Scale property of the chosen Site Type.

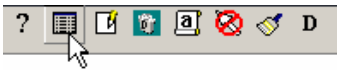
Step 87: Search for and Select Sites

All inside plant equipmenet in SPATIAL*net* FM must be assigned to a floor in the building inside which it is located. So, before we can add any equipment to a building, we have to add at least one floor. But before we can add a floor to a building, we must be able to find the building, select it, and display its details.

The steps below provide several alternative options for doing this. You should try using each of them.

Option 1: Select from the map

1. Select one of the buildings by clicking on its map symbol in the drawing window, as shown in Figure 215, above.
2. Click the **Details** button on the **General** toolbar, as shown:



Option 2: Search by Name

1. Run the menu command **FM ISP > Find > Find by Name > Building...** The panel shown in Figure 216, below will be displayed. This panel provides a number of search operators, such as **equals**, **like**, etc.
2. Fill in the values for each of the fields shown in Figure 216, below for each of the buildings and click the **OK** button.

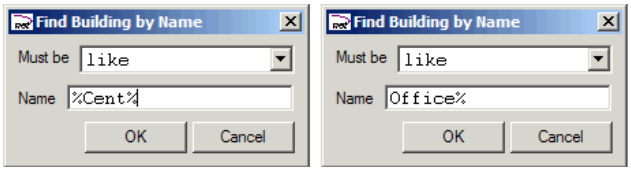


*Figure 216 –Wildcard searches for all Buildings whose name includes **Cent** (left) and whose name begins with **Office** (right)*

3. When the search is complete, the details of the building will be displayed.

Notes:

1. **Using a like search lets you include wildcards (represented by the % character).**
2. **The search is case sensitive.**

Option 3: List Buildings

1. Run the menu command **FM ISP > List > Building...** This will display a panel listing all buildings by name.
2. Click on the row containing the building you wish to work with.
3. Click the **Display >>** button and select **Details** from the pop-up menu.

After completing steps in any of the above options, the details panels for each of the buildings should be visible, similar to those shown below.

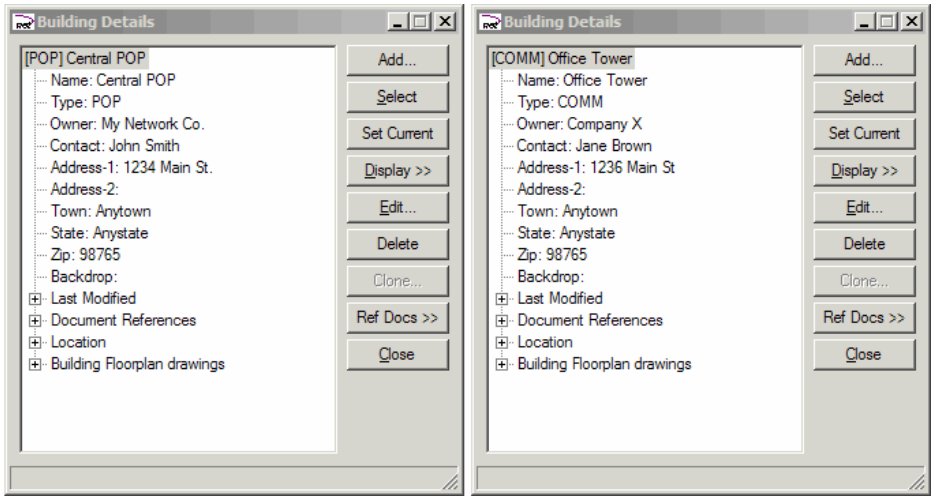


*Figure 217 –Details panels for **Central POP** and **Office Tower** buildings.*

Step 88: Add Floor to Sites

If you completed Step 87 you should have Details panels open for each of the **Central POP** and **Office Tower** buildings, as shown in Figure 217, above (if you haven't, you should do this now).

We will now add a floor to each of these buildngs. After doing that, we can choose to import floorplan drawings, or to draft a floorplan directly into AutoCAD. Neither of these are mandatory, but to illustrate the process, we shall import a floorplan in Step 89, below.

To add a floor to each of our buildings, repeat the following steps for each building:

1. Click the **Add...** button on the **Building Details** panel, and select **Add a floor to the highlighted building** from the pop-up menu.
2. Fill in the values for each of the fields shown in Figure 218, below for each of the buildings, and click the **Add** button.

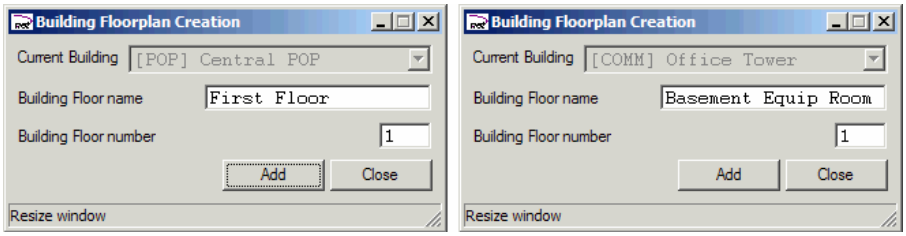


*Figure 218 – Floor details to be added for **Central POP** (left) and **Office Tower** (right)*

3. Dismiss each of the **Building Floorplan Creation** panels with the Close button.
4. Expand the **Building Floorplan drawings** branch of the tree view in each of the **Building Details** panels. The new floors should be visible, as shown highlighted in Figure 219, below.

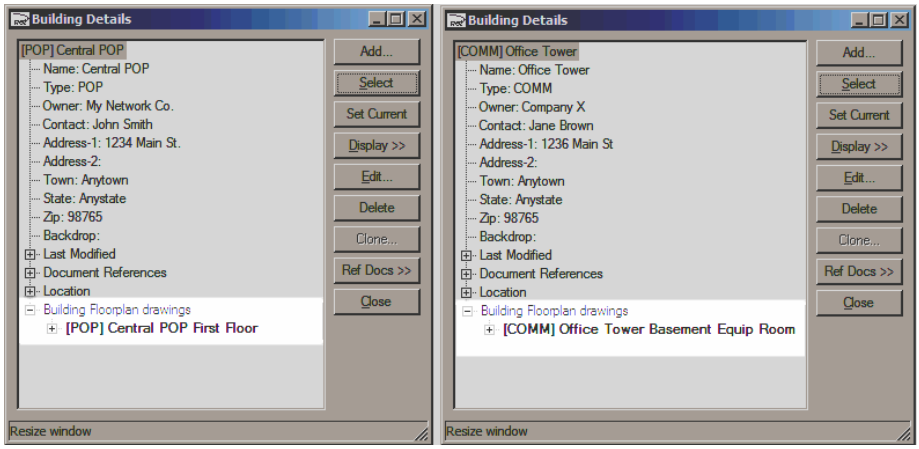


*Figure 219 – Floors added to **Central POP** (left) and **Office Tower** (right)*

Step 89: Import Floorplan Drawing

While it is not necessary to import a floorplan drawing, if you have one, it is certainly helpful to use the information in it to accurately locate inside plant equipment. For this exercise, a sample floorplan drawing supplied with the documentation will be used (you can substitute this for a drawing of your own if you wish).

Existing floorplan drawings are imported into the system using CAD-Scrape in a way that is completely analogous to the import of the base map in *: CAD-Scrape Landbase and* Strand of Part A (see page 28).

To import a floorplan drawing:

1. Run the AutoCAD **File > Open** command.

2. Navigate to the directory containing your SPATIAL*net* documentation (usually **Program Files/SPATIALnet/doc**)
3. Open the AutoCAD file **Floorplan.dwg**.
4. When the file is opened, SPATIAL*net* will display the following panel:

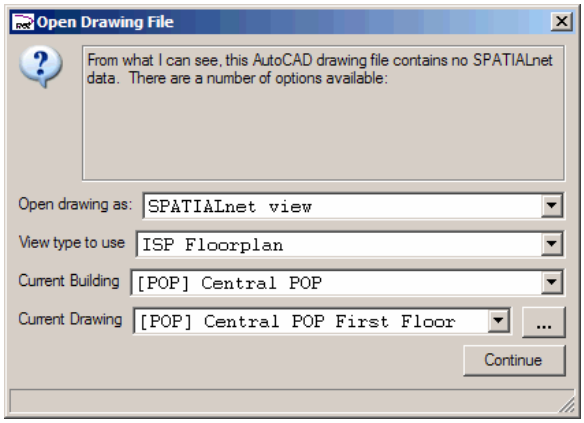


Figure 220 – SPATIALnet Open Drawing File Dialog

5. Select **SPATIALnet view** from the **Open drawing as** pulldown list and make sure the other fields are filled in as shown in Figure 220 above. These fields determine which building and which floor the floorplan will be associated with.
6. When the field values are correct, click on the **Continue** button.
7. When the graphics are visible in the AutoCAD window and the **Open Drawing File** dialog has closed, run the **File > Save** AutoCAD command to import the graphics into the SPATIAL*net* database (running the **Save** command will automatically initiate the CAD-Scrape process. The Scrape process may take a minute or two. See Chapter 10 for further information about CAD-Scrape).

When this process is complete, the imported graphics will be displayed whenever you open the floorplan view for the specific building and floor.

Notes:

1. **You can also CAD-Scrape any AutoCAD objects created during the session using normal AutoCAD drawing commands. You don't always have to open a DWG file. You do have to have a SPATIALnet view open, however (not a native CAD file).**
2. **Any blocks scraped in must be defined either in a DWG file on the AutoCAD Support File path (the usual method), or in the view template DWT file. CAD-Scrape only records block names; it does not read the block definition into the database.**
3. **Properties of CAD Layers must be defined in the view template DWT file. If the system renders a CAD object into a layer which is not**

defined in the view template DWT file, the default color, linetype, lineweight, etc. will be used.

4. **Because of points 2 and 3, the objects rendered back into AutoCAD after being imported using CAD-Scrape may appear different from the original objects. This can be fixed by making the Layer and block definitions available as described in points 2 and 3.**

To validate that the floorplan is indeed stored in the database, close the window from which you ran the CAD-Scrape, and open the details panel for the **Central POP** building. Clicking the **Display>>** button and selecting **Floorplan View** from the pop-up menu will retrieve the floorplan from the database and display it in AutoCAD.

Note: If you do not import a floorplan, then displaying a floorplan view will create an empty window. It is still possible to place equipment into this window, and to draft and Scrape CAD objects at any later time.

Step 90: Adding Racks

Let us now add a pair of racks to each of the buildings we created in Step 86, above. Repeat the following steps for each of the buildings:

1. Select the building and display its **Details** panel (see Step 87, above).
2. Click the **Display>>** button and select **Floorplan View** from the pop-up menu. The floorplan view of the building will be displayed in a new drawing window.
3. Run the Add Rack function by one of the following methods:
    a. Run the **FM ISP > Add > Rack...** menu command.
    b. Click the **Add Rack** button on the **Add Network Equipment** ISP toolbar, as shown here: 
4. Fill in the fields of the **Rack Creation dialog** as shown in Figure 221, below (see item 5 below for making the selection in the **Type** field)

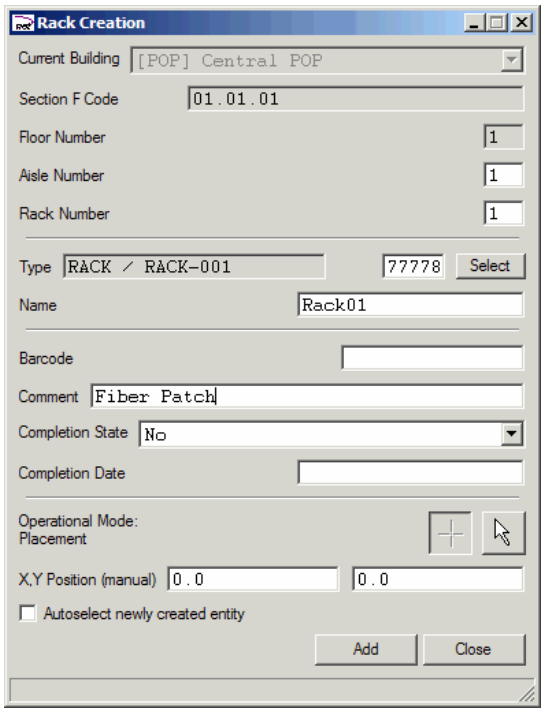


Figure 221 – Rack Creation panel

5. To select the type of equipment to add, click the **Select** button. This will bring up a list of entries in the equipment dictionary that match the class of equipment you are trying to add. You can refine this list further using the query operators and filter values at the top of each column. When you have highlighted the row containing the rack type you would like to add, click the **Select** button on the equipment dictionary list panel. This will close the panel and return you to the **Rack Creation** panel. For this example, select the Virtual Rack entry.

Note: If you know the numerical Catalog ID of the equipment you would like to add, you can type it directly into the Type field shown in Figure 221, above without having to list entries from the equipment dictionary.

6. When all the information has been specified in the **Rack Creation** panel, click the location on the floorplan drawing where you would like to place the rack (click twice—once to set input focus to the drawing window, and again to specify the location of the new rack). The new rack should appear similar to that shown below.

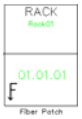
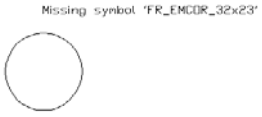


Figure 222 – New rack appearance in floorplan view.

Note: The rack symbol will not be displayed correctly if the directory containing the block file is not on AutoCAD's support path. In this case, a symbol similar to the following will be displayed instead of that shown in Figure 222:



To remedy this problem, ensure that the `blklib\floorplan` subdirectory of your SPATIALnet installation is included in AutoCAD's support file search path.

7. Repeat items 3 to 6 above to create another rack in the **`Central POP`** building (change the **Rack No**, **Name** and **Comment** fields to different values), and then repeat items 1 to 6 to create a new pair of racks in the **`Office Tower`** building.

Note: The following must be unique for each rack in a building:

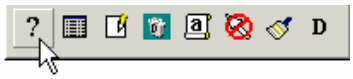
- **Combination of Floor No, Aisle No, Rack No.**
- **Name**

These values may be duplicated in different buildings, however.

Step 91: Generating a Rack Elevation View

Let us now look at our new racks front-on in an elevation view. To do this:

1. Bring up the floorplan view of the **`Central POP`** building, which should now contain the two racks added in Step 90 (the first two items in Step 90 will remind you how to do this).
2. Change the system selection type to Multiple by clicking the **Current Selection** button on the **General** toolbar as shown here:



   Then choose **`Multiple`** in the **Selection Mode** field of the dialog box that is displayed. Also, if it is shown, click the **Clear** button to remove any existing selection (this button will be hidden if there is no current selection).
3. Select the pair of racks in the floorplan view by drawing a box around them, using the normal AutoCAD/Windows graphical selection technique.
4. Click on the Rack Elevation toolbar button, as shown here:  Two rack elevation symbols, one representing each of the selected racks, will be displayed in a new drawing window.

Step 92: Adding Chasses

With the elevation view you created in Step 91, above, it is easy to insert equipment into the racks. In this step, we will insert a fiber patch panel and then a chassis into the racks. (We will insert some transmitter and receiver cards into the chassis in Step 93.)

1. If a rack elevation showing the two racks in the **Central POP** building is not visible, re-do Step 91, above, to display it.
2. Select the rack labeled **Fiber Patch** by clicking on it in the elevation view. A small triangle will be displayed at the base of the rack indicating that it is now selected.
3. Run the Add Chassis function by one of the following methods:
   a. Run the **FM ISP > Add > Chassis...** menu command.
   b. Click the **Add Chassis** button on the **Add Network Equipment** ISP toolbar, as shown here: 
4. Fill in the fields of the **Chassis Creation** dialog as shown in Figure 223, below (see item 5 below for making the selection in the **Type** field). Note that we will place the chassis at mount point 44 on the rack, which is selected from the **Position** pull-down list.

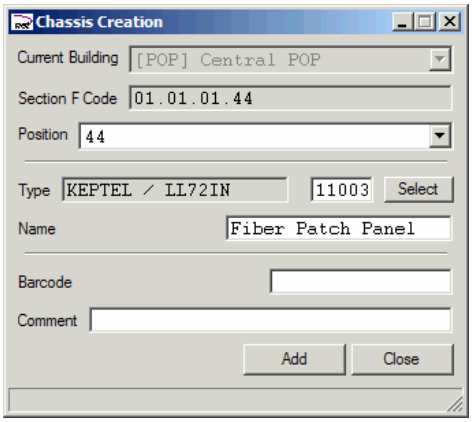


Figure 223 – Chassis Creation panel

5. To select the type of equipment to add, click the **Select** button. This will bring up a list of entries in the equipment dictionary that match the class of equipment you are trying to add. You can refine this list further using the query operators and filter values at the top of each column. In this case, we will add a **Antec/LL72IN Fiber Patch Shelf**.

   When you have highlighted the row containing the chassis type you would like to add, click the **Select** button on the equipment dictionary list panel. This will close the panel and return you to the **Chassis Creation** panel.

Note: If you know the numerical Catalog ID of the equipment you would like to add, you can type it directly into the Type field shown in Figure 223, above without having to list entries from the equipment dictionary.

6. When all the information has been specified in the **Chassis Creation** panel, click the **Add** button shown in Figure 223.
7. Now add a transmitter/receiver chassis in the second rack by selecting the rack, and repeating items 2 to 6 above. In item 5, choose the **Antec LL2 Light Link 2** chassis from the equipment dictionary.
8. The new chasses should appear similar to those shown in Figure 224, below.

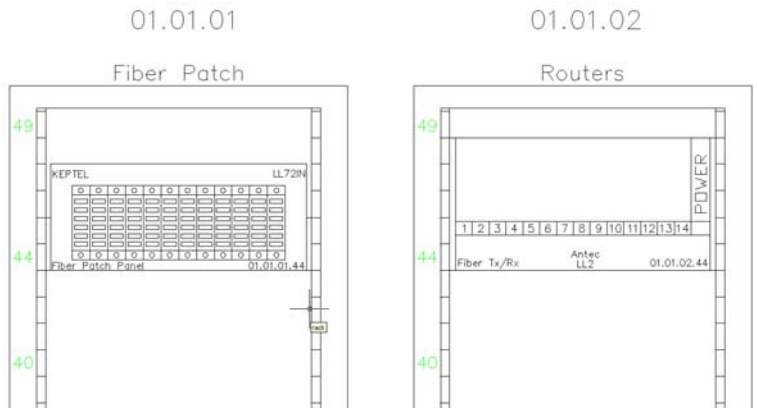
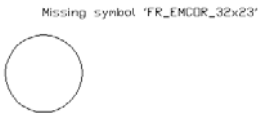


Figure 224 – New chasses appearance in elevation view.

Note: The symbols will not be displayed correctly if the directory containing the elevation block files is not on AutoCAD's support path. In this case, symbols similar to the following will be displayed instead of those shown in Figure 224:



To remedy this problem, ensure that the blklib\elevation subdirectory of your SPATIALnet installation is included in AutoCAD's support file search path.

9. Repeat items 3 to 7 above to create a similar pair of chasses in the racks in the **Office Tower** building.

Note: The following must be unique for each chassis in a building:

- **Combination of Floor No, Aisle No, Rack No, Mount Point No.**
- **Name**

These values may be duplicated in different buildings, however.

Step 93: Adding Cards

Now that we have some racks and chasses in place, the last pieces of equipment we'll install at each site are some cards that fit into the LL2 chassis. As you'll see, the process for inserting cards is almost a re-run of Step 91 and Step 92, giving the ISP environment a consistent set of procedures for managing any kind of equipment.

To insert an optical transmitter into the LL2 chassis,

1. Make sure the rack elevation view you were working with during Step 92 is open and the LL2 chassis (shown on the right in Figure 224) is visible. If it is not, run Step 91 again.
2. Select the LL2 chassis labeled **Fiber Tx/Rx** (shown on the right in Figure 224) by clicking on it in the elevation view. A square SPATIAL*net* manipulator will appear at the bottom left corner of the chassis when it is selected.
3. Run the Add Card function by one of the following methods:
    a. Run the **FM ISP > Add > Card...** menu command.
    b. Click the **Add Card** button on the **Add Network Equipment** ISP toolbar, as shown here:

4. Fill in the fields of the **Card Creation** dialog as shown in Figure 225, below (see item 5 below for making the selection in the **Type** field). Note that we will place the first card in slot 1 of the chasses, and the second card in slot 2. Slot numbers are selected from the **Position** pull-down list shown below.

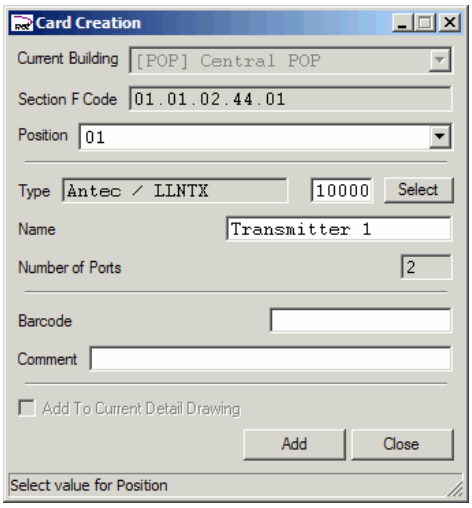


Figure 225 – Card Creation panel

5. To select the type of equipment to add, click the **Select** button. This will bring up a list of entries in the equipment dictionary that match the class of equipment you are trying to add. You can refine this list further using the

query operators and filter values at the top of each column. In this case, we will add an **Antec/LLNTX Optical Transmitter**.

When you have highlighted the row containing the card type you would like to add, click the **Select** button on the equipment dictionary list panel. This will close the panel and return you to the **Card Creation** panel.

Note: If you know the numerical Catalog ID of the equipment you would like to add, you can type it directly into the Type field shown in Figure 225, above without having to list entries from the equipment dictionary.

6. When all the information has been specified in the **Card Creation** panel, click the **Add** button shown in Figure 225.
7. Now add an optical receiver to the same chassis by repeating items 2 to 6 above. In item 5, choose the **Antec/LLNRX Optical Receiver** from the equipment dictionary and insert this card into slot 2 of the LL2 chassis.
8. The new cards should appear similar to those shown in Figure 226, below.

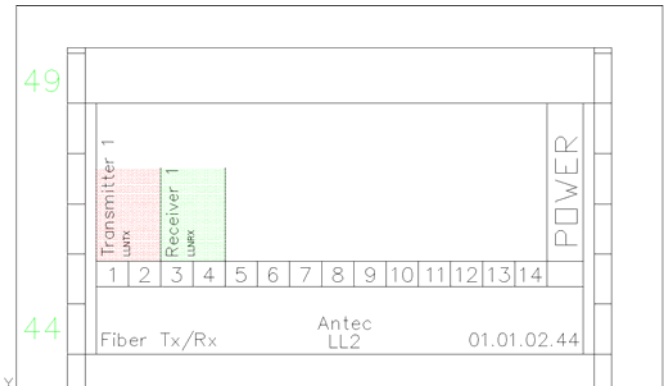
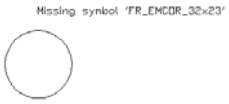


Figure 226 – New cards' appearance in elevation view.

Note: The symbols will not be displayed correctly if the directory containing the block files is not on AutoCAD's support path. In this case, a symbol similar to the following will be displayed instead of that shown in Figure 226:



To remedy this problem, ensure that the blklib\elevation subdirectory of your SPATIALnet installation is included in AutoCAD's support file search path.

9. Repeat items 2 to 7 above to create a similar pair of cards in the **Fiber Tx/Rx** chassis in the **Office Tower** building.

Note: The following must be unique for each card in a building:

- **Combination of Floor No, Aisle No, Rack No, Mount Point No, Slot No.**
- **Name**

These values may be duplicated in different buildings, however.

Step 94: Creating a Connectivity Schematic Detail Drawing

We've covered two of the three ISP view types (floorplans and rack elevations). Let's now look at the third (schematics).

In Step 92 and Step 93, we inserted a fiber patch panel and some optical equipment, but we did not connect them in any way. So, while we have a record of our equipment and its location, we do not yet have any record of its connectivity. This is where schematic views are very helpful.

Schematic views are similar to floorplans, in that they exist as persistent objects in the database. Therefore, before you can use a schematic view, you need to create it in the database and name it. By contrast, rack elevation views are generated "on the fly" and do not exist in the database. When you close a rack elevation window, you have to regenerate the elevation view by selecting the same set of racks again.

One difference between schematic and floorplan drawings, however, is that floorplan drawings are associated with a unique floor in the building. Schematic drawings, on the other hand, are just associated with the building itself, and know nothing about floors. This makes it easier to represent connected equipment on different floors.

To create a new schematic drawing:

1. Make sure the **Current Building** is set to the building you are working on (**Central POP** for this example).

Note: If you have an ISP view open already (a rack elevation, floorplan or schematic), Current Building will be set to the building containing the data in the view. If you do not have an ISP view open already, or you would like to check Current Building, click the Current Selection button on the General toolbar, and inspect or make a selection from the SPATIALnet Current Selection Panel. You can change Current Building by clicking on the field where the current value is displayed and making a selection from the pop-up list. See Figure 227, below for an example.

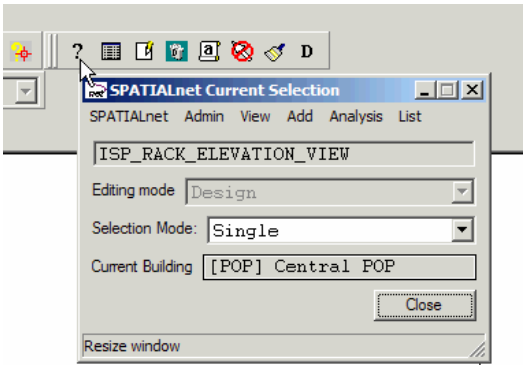


Figure 227 – Current Selection panel allows you to set Current Building.

2. Run the menu command **FM ISP > Detail Drawings > List Detail Drawings**. A panel with an empty list box should be displayed.

3. Click the **Add...** button on the **All Detail Drawings** list panel.
4. Fill in the fields of the Detail Drawing Creation panel as shown in Figure 228, below.

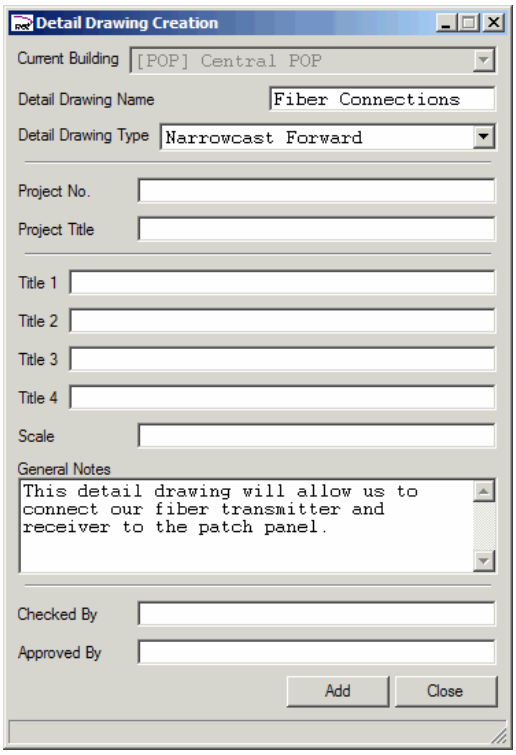


Figure 228 – Detail Drawing Creation panel

5. When the fields are filled in as required, click the **Add** button shown in Figure 228.
6. Click the **Close** button to dismiss the panel.

The new detail drawing you just created should now be visible in the **All Detail Drawings** list for the current building.

7. Select the new row in the **All Detail Drawings** list panel, click the panel's **Display>>** button and select **Detail Drawing** from the pop-up menu. This will retrieve the detail drawing from the database and display it on the screen,

It should be immediately obvious that this drawing is currently empty. This is because SPATIAL*net* FM takes the approach that connectivity schematics should be more than a diagram of all connections that the software can find the network without any context. Rather, schematics should present a form of structured documentation which conveys the underlying design intentions and organization of the network in a way that is meaningful to human operators. To achieve this, you must tell SPATIAL*net*

FM which pieces of equipment are to appear in which schematic drawings, and where they will be placed within those drawings. SPATIAL*net* FM will then take care of the rest (connectivity, data updates, etc.)

So, since we have not yet placed any equipment into our new schematic, it is currently empty. Let us remedy that state of affairs now.

Step 95: Placing Equipment into a Schematic View

To place equipment into a schematic view, it is easiest to first select the equipment in another view, and then place it into the schematic. Rack elevations are ideal for selecting the equipment we need, so make sure the rack elevation view we generated in Step 91 is currently visible on the screen (see Step 91 above if you need to regenerate it).

In addition, the window containing the currently empty schematic drawing should also be visible. An easy way to achieve this is to use AutoCAD's **Window > Tile Vertically** menu option. Then arrange the windows so they look something like Figure 229, below.

We will place a schematic representation of the patch panel, the transmitter and the receiver into the schematic view. To do this:

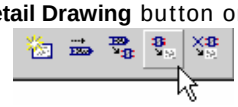
1. Click on the piece of equipment in the rack elevation view that you wish to place into the schematic view.
2. Click the **Place Selected Card in Detail Drawing** button on the Detail Drawings toolbar, as shown here: 
3. The **Detail Drawing Card Creation** panel will appear, as shown in Figure 229, below.
4. Click on the Schematic window to activate it for input. Then click at the location at which you wish to place the schematic representation of the selected equipment. The system will display the appropriate symbol at the location you specified.
5. Close the **Detail Drawing Card Creation** panel.
6. Repeat items 1 to 5 above until all of the equipment you wish to display in the schematic has been placed (in the current example, this includes the fiber patch panel, the optical transmitter and the optical receiver. A suggested layout is shown in Figure 229, below).

Figure 229, below shows the sequence of steps as they typically appear on the screen.

Note: Many of the Tooltips and other user interface features refer to this function as placing a "Card" into a Detail or Schematic drawing. The function

actually applies to any connectable device, including racks, standalone equipment and chasses as well as cards.

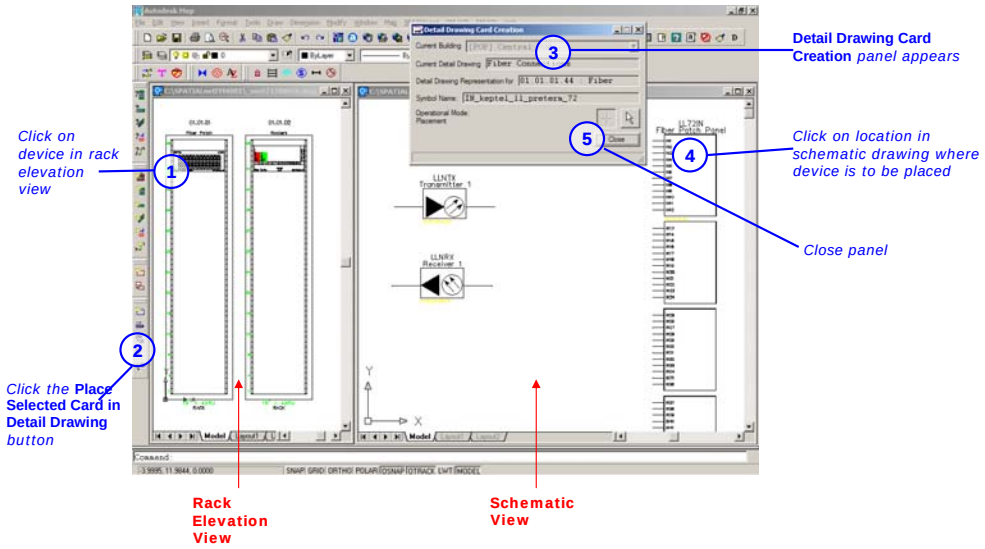
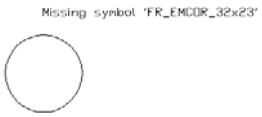


Figure 229 – Sequence of steps for placing equipment from a rack elevation view into a schematic view

Note: The symbols will not be displayed correctly if the directories containing the block files are not on AutoCAD's support path. In this case, a symbol similar to the following will be displayed instead of those shown in Figure 229:



To remedy this problem, ensure that the `blklib\elevation` and `blklib\detail` subdirectories of your SPATIALnet installation are both included in AutoCAD's support file search path.

Step 96: Connecting Equipment

Once equipment is visible in a schematic drawing, connecting it is very easy. Notice that if you select an object in a schematic view, yellow circles are drawn at each connectable port. Thes circles are SPATIAL*net* FM's "connection manipulators" and are used to form new connections when you drag and drop them from one port to another.

To try this out, let's connect the transmitter and receiver to the patch panel with some jumper cables.

1. Ensure the schematic drawing shown in the right window pane in Figure 229, above, is visible and has the input focus.
2. Click on the transmitter symbol. A connection manipulator (shown as a yellow circle) will be drawn at each of the ports.
3. Click on the connection manipulator and drag it over each piece of equipment. When a connection manipulator senses another piece of equipment nearby, it will activate the manipulators on that equipment's ports.
4. Continue to drag the connection manipulator to the port labeled IN 1 on the patch rack. Drop the manipulator onto this port.
5. A pop-up will prompt you to create a new connection between the ports. Click on this button, as shown in Figure 230, below.

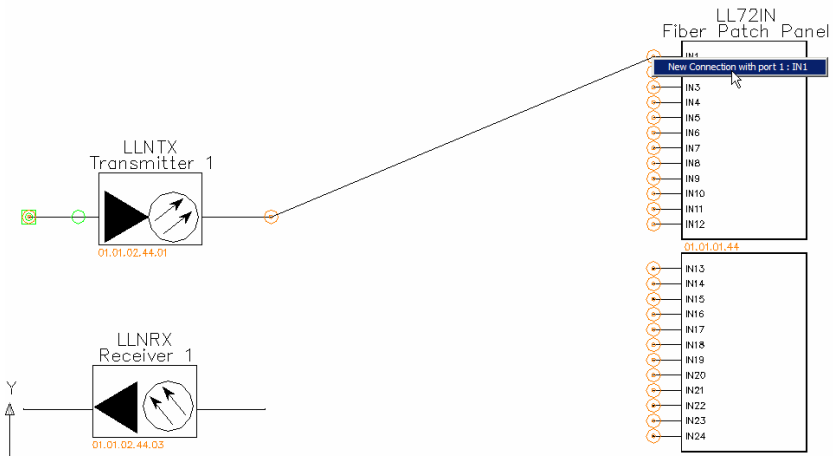


Figure 230 – Creating a connection between equipment ports.

6. When you click the pop-up button to create the connection, the **Cable Creation** panel will be displayed, prompting you for details about the jumper cable to be used for the connections.
7. Click the **Select** button on the panel and select a Fiber Jumper cable type from the equipment dictionary. Fill in the other fields with values you choose.
8. Click the Add button on the Cable Creation panel. This will create a connection between the ports you selected, and will show this graphically in the schematic.

9. When you select the connection, green manipulators will be displayed at each vertex of the connection line, and red manipulators will be displayed at each segment's mid-point. The green manipulators move the vertices, while the red manipulators move the entire segment orthogonally. These tools let you tidy up the appearance of the schematic.

Notes:

1. **Any graphical changes you make to the schematic (move equipment, move linework, etc.) will be stored in the database and will be present each time the drawing is viewed in future sessions.**
2. **Deleting a connection line in a schematic deletes the connection between the ports, and removes the cable making the connection from the database.**
3. **Deleting a piece of equipment from a schematic can either:**
    a. **Delete the equipment, and all its representations from the datatbase, or**
    b. **Delete the equipment from the schematic only. In this case, any connections to the equipment are replaced by a "feather" hyperlink symbol:**

4. **CAD-Scrape may be used in schematic drawings to add miscellaneous annotation, mark-up, etc.**

Step 97: Moving Equipment

Step 96 completed the process for adding and connecting inside plant equipment. Let us now see how we move it around.

To change the location of floor-mounted equipment (racks and standalones), you simply select the symbol representing the equipment and drag it to its new location using the manipulator.

Moving equipment in rack elevation views is more interesting, as rack-mounted equipment can only be moved to valid, unoccupied mount-point locations, and cards or modules can only be moved to valid, unoccupied slots. SPATIAL*net* FM's elevation views provide a visual "drag and drop" approach to this, similar to creation of connections in schematic views that we saw in Step 96.

Suppose we wish to move the LL2 chassis we created in Step 92, and all the cards it now contains, to a new location either in the same rack or a different rack. To do this:

1. If it is not already visible, generate the rack elevation view containing the two racks in the **Central POP** building, as described in Step 91, above.
2. Select the LL2 chassis by clicking on it. Notice that a green square manipulator is displayed in the bottom left corner. Click on this manipulator to begin dragging the chassis.
3. As you begin to drag the chassis, all the rack mount-points in the rack elevation view are highlighted with yellow square markers. Drop the chassis onto one of these markers in either rack.

4. The mount-point numbers near your drop point will be displayed. Click on the mount point number at which you wish to re-mount the chassis and the chassis and all its contents will be moved.

Step 98: Creating a Cable Run List Report

In addition to the tools we have seen so far for managing data about your network, SPATIAL*net* FM also provides many reporting tools. We shall look at one of these in this step, the Cable Run List.

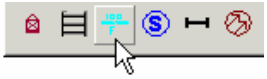
The Cable Run List report lists all of the inside plant connections, disconnections and re-connections that are made in the current job, and stores these in a .CSV file which can be read with Microsoft Excel. To produce a cable run-list for the current job:

1. Run the menu command **FM ISP > Reports > Generate a Cable Run List**
2. Type a filename into the **Output Path** field, or click the button labeled **...** to browse to a location where the file will be stored. Note that the filename should end in the extension **.csv**
3. Click the **Save List** button.
4. Open the file you saved in Microsoft Excel.

Step 99: Connecting ISP Buildings with Outside Plant Cables

One of the most powerful features of SPATIAL*net* FM is the ability to manage a comprehensive and detailed model of both inside and outside plant networks in a single application. To utilize the full power of this feature, you must be able to connect inside plant to outside plant. Since this part of the tutorial focuses on inside plant (see Part A for outside plant), we shall create only minimal outside plant in this step—just enough to connect together the two buildings that have been with us since Step 86.

To do this:

1. Create a map view showing both buildings (Refer to Step 87, above, for a reminder of how to do this. Use the **Display>> Map View** button).
2. Make sure both buildings have at least one rack in them. If they do not, you will have to add one before you can continue with this step.
3. Select the **Central POP** building by clicking on it.
4. Run the Add Fiber Cable command to create a fiber between the **Central POP** and **Office Tower** buildings via one of the following methods:
    a. Run the menu command **FM OSP > Add > Fiber Cable**
    b. Click the Add Cable button on the Fiber Add toolbar, as shown here: 

5. Fill in the fields of the Fiber Cable Creation panel as shown in Figure 231, below.

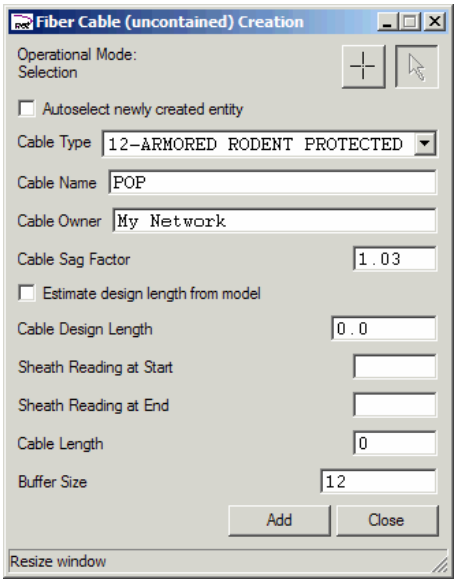


Figure 231 – Fiber Cable Creation Dialog

6. Ensure that the "Selection" arrow is selected in the **Fiber Cable Creation** Dialog, as in Figure 231, above, and also shown here:

7. Click the **Add** button. You will be prompted to specify which rack the fiber cable you are about to create should be connected to, and which splice enclosure within the rack. Since none of our racks have splice enclosures in them yet, you must create one with the **New Splice** button.
8. After creating the new splice, expand the **OSP Equipment** branch of the rack view and **Select** the splice, as shown in Figure 232, below. This will begin the cable creation process.

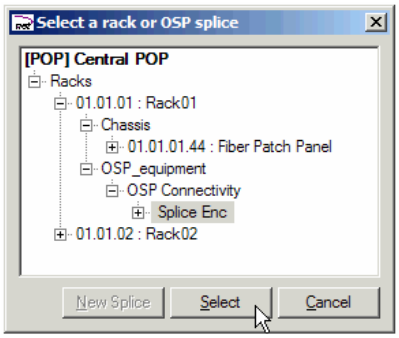


Figure 232 – New OSP fiber splice point in rack

9. We won't bother with any real outside plant routing for this cable, so just right click on the **Office Tower** building, and select this from the displayed pop-up to connect the cable (Hint: if the Cable Creation dialog box is overlapping one of the buildings, use the button on the **Line Capture Control** panel to hide it).
10. You will need to specify the terminating splice enclosure for the cable in the **Office Tower** building, just as you did for the **Central POP** in items 7 and 8, above.
11. Click the Done button on the Line Capture Control panel to finalize the cable and add it to the database.

The last task in the process is to add a fiber termination panel so that we can connect the fiber to the ISP patch shelf we created in Step 92. To do this:

12. Select the rack containing the end of the OSP fiber we just created (**Rack01** if you followed the example in Figure 232). You may do this by bringing up a floorplan view or rack elevation (see Step 91) containing it, running the menu command **FM ISP > List > Racks**, or by any other selection method.
13. When the rack is selected, bring up its Details panel by clicking on the **Details** button (if you are looking at a list panel), or clicking the **Details** button on the **General** toolbar, as shown here.
14. Find the **Splice Enclosure** entry under the **OSP Equipment – OSP Connectivity** branch and select it by clicking the **Select** button on the Details panel.
15. Add a Term Panel to the splice enclosure (see Step 29 of Part A on page 88). Choose the **72-port IO** panel and call it **Central POP TERM 1.**
16. With the **Splice Enclosure** selected as in item 14, splice the Term Panel to the OSP Fiber (see Step 43 of Part A on page 133).

This completes adding an (albeit minimal) outside plant cable and connecting it physically to a rack.

Step 100: Creating OSP/ISP Interfaces

Before we can trace a signal from inside plant to outside plant, we need to tell the system how the two networks are interfaced together. That is, we need to model the front-to-back connection of the patch panel chassis we created in Step 92. In SPATIAL*net* FM, this kind of connection exists as a named mapping called an *OSP Interface*. We will now create one.

To do this:

1. Make sure Current Building is set to **Central POP** (see the Note in Step 94 for more information about **Current Building**)

2. Run the menu command **FM ISP > List > OSP Interface**. This will list the OSP interfaces which have been defined for the **Current Building**. At present, there are none so this list will be empty.
3. Click the **New** button on the list panel and fill in the fields of the **Interface Creation** panel with the values shown in Figure 233, below.

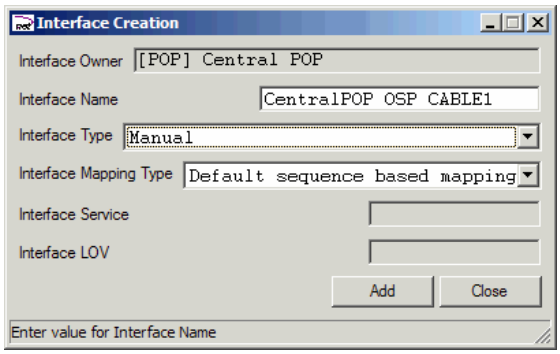


Figure 233 – Interface Creation panel.

4. Click the Add button to create the new interface, which will be named **CentralPOP OSP CABLE1**.

We now need to apply this interface to map the front of the fiber patch panel to the back (i.e., to the fiber termination created in item 15 of Step 99, above).

To do this:

5. Select the fiber patch panel by any means (rack elevation drawing, list, entry in Rack01 Details panel, etc.).
6. Click the **Edit** button. The **Chassis Modification** panel will be displayed. Note that the **Interface** field is currently set to **nothing selected**.
7. Click on the **Interface** field and choose **Select current building** from the pop-up menu. This will display a list of all OSP Interfaces defined for the current building.
8. Choose the **CentralPOP OSP CABLE1** interface (listed under **ISP/OSP Interfaces**) and click the **Select** button.
9. Click **Apply** and then **Close** on the Chassis Modification panel.

Figure 234 below shows the sequence of panels used to select the fiber patch panel and map it to the OSP Interface.

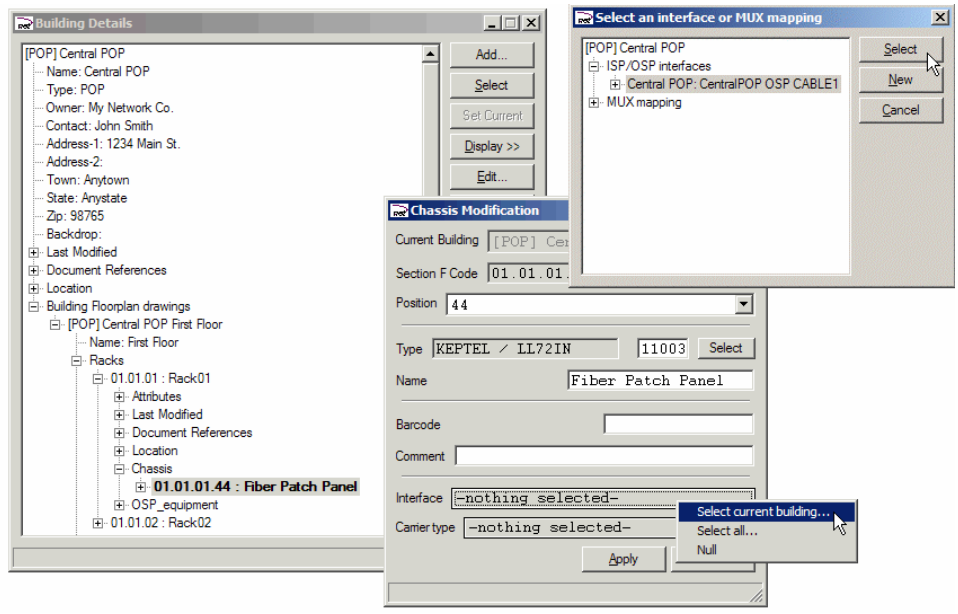


Figure 234 – Mapping an OSP Interface to the fiber patch panel

At this point, the OSP Interface is pointing to the front panel. We now need to add the back panel part. To do this, we follow an identical process to items 5-9 above, but we select the **Term Patch Port Bank** under the rack's **OSP connectivity** branch.

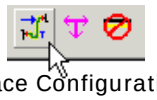
This completes the connection of the inside plant network to the outside plant. It is now possible to trace connections through both networks, as we shall see in the next step.

Step 101: Tracing

Now that we have connected our inside and outside plant networks, let's trace a physically connected path from one building to the other. To do this:

1. Open the map view showing the OSP cable connecting the **Central POP** and **Office Tower** buildings.
2. Open the Detail Schematic drawing for **Central POP** you created in Step 94.
3. Use the AutoCAD menu option **Window > Tile Vertically** to show both windows approximately equally on the screen.
4. Select the jumper connection from Transmitter 1 to the fiber patch panel by clicking on the line connecting the two.

5. Begin a trace by clicking on the **Trace Fiber** toolbar button on the **Trace** toolbar, as shown here:



6. Fill in the fields of the Trace Configuration panel as shown in Figure 235, below.

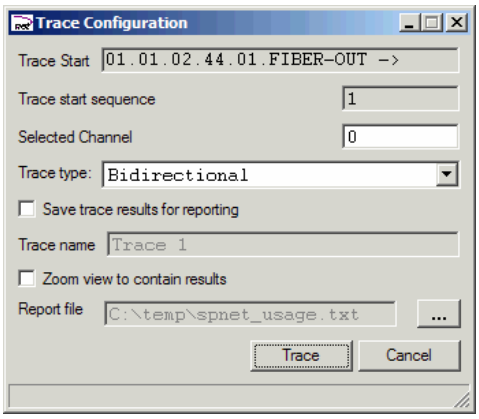


Figure 235 – Trace Configuration Panel

7. Click the **Trace** button. The system will begin at the selected inside plant jumper cable and work its way through the inside and outside plant networks. The physical path traversed by the trace is visually highlighted and a report is produced. Note in particular that the trace begins inside the **Central POP** building, travels along the OSP cable, and ends at the **Office Tower** building. The results are shown in Figure 236, below.

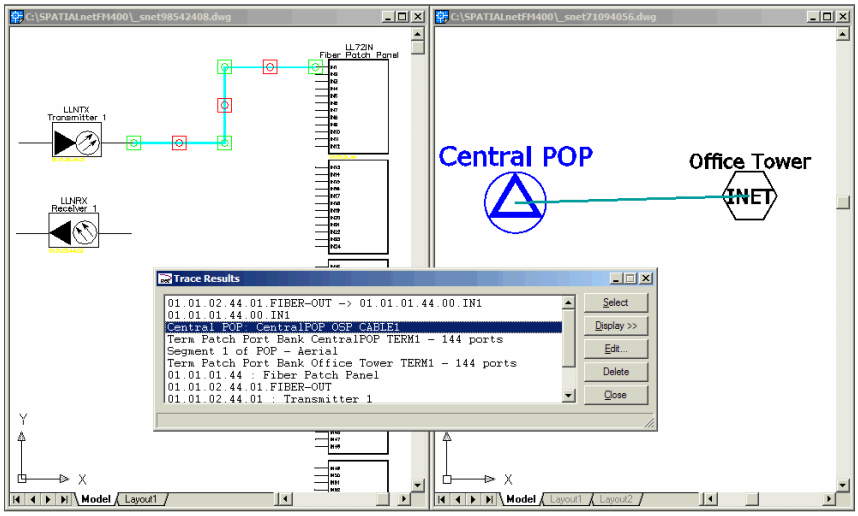


Figure 236 – Trace results showing tracing from ISP to OSP and into next building.

Step 102: Configuring a Rack Type

Note: Step 102 and Step 103 are only relevant to users who will be responsible for configuring new types of inside plant equipment in SPATIAL*net* FM's equipment dictionary and/or creating ISP symbols. If you will not be required to do this, you can skip these steps.

If you have worked through Part A of this tutorial, which deals with outside plant, you will already be familiar with how to update the various dictionaries SPATIAL*net* FM uses to specify the various classes of OSP equipment; a similar approach is used for inside plant. The main difference, however, is that in ISP all equipment configuration is managed from a single equipment dictionary, instead of a separate dictionary existing for each class of equipment. (The reason behind this is that a common set of properties can be used to specify almost all types of ISP equipment, whereas each class of OSP equipment requires different specification properties.)

Entering the specification of a new type of ISP equipment into the system generally involves two tasks:

- Creating an equipment dictionary entry.
- Creating various kinds of intelligent symbology and registering these with the system.

We shall cover both of these below. Before we do, however, we must review some background about SPATIAL*net* FM's ISP view types.

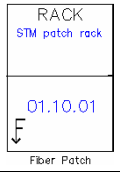
Inside Plant View Types

Another key difference between inside and outside plant equipment is that there is only a single outside plant view type (the map view), while there are three inside plant view types:

- Floorplan views
- Rack elevation views
- Connectivity schematic detail views.

Full details of these view types are provided in the *SPATIALnet FM User Manual*. We will just review the basics here, as they apply to the configuration of equipment types.

The most important fact about ISP views is that a single piece of equipment will have very different appearances and behaviors in each of the three view types, as summarized in the table below. Note, however, that a given piece of equipment may not be represented in all view types (rack mounted equipment has no representation in floorplan views, for example).

ISP View Type	Type of display	Types of actions which can be performed	Example equipment block for view type
Floorplan	"Top down" plan view showing location of racks and free-standing "Standalone" equipment (chasses, cards and cables are not shown).	Can move objects around to change their location, but can't graphically change connectivity or vertical rack layouts.	

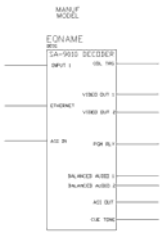
Rack Elevation	Front elevation view showing actual dimensions of racks, chasses, cards and standalones. Cables are not shown.	Location of objects in racks can be maintained by "drag and drop" functions. Cannot graphically change floor location or connectivity.	
Schematic Detail	Ortho-schematic circuit diagram showing ports on equipment and any connections between them (i.e. jumper cables, patch cords, etc.). Any kind of equipment can be displayed in this view, so long as it is connectable to the network.	Can graphically manage connectivity by dragging and dropping connections between ports. Cannot graphically change floor location or vertical rack layouts.	

Table 4. Summary of ISP View types.

The block library is organized so that symbols for each of the three types of views are stored in a separate sub-directory, as shown below. Normally, these directories are added to AutoCAD's Support File Path when the system is installed.

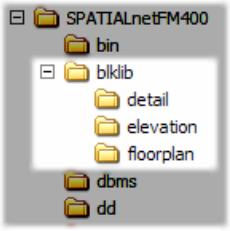


Figure 237 – SPATIALnet ISP block library organization

When you configure a new piece of equipment, you need to decide which of the three types of views it will be visible in, and create symbology accordingly. This means that for floor-standing equipment (racks and standalone equipment), you may need to configure three separate symbols for a single type of equipment. For all other types (chasses and cards), you will have to configure two at most. Cables do not require symbols in any of the view types as they are drawn automatically by the system when required.

Intelligent Symbology

As we saw in the earlier steps, SPATIAL*net* FM's ISP functions provide a highly interactive graphical environment for viewing and managing inside plant. For this to work, the symbols displayed in inside plant views need to have certain kinds of data encoded within them. Specifically:

- The location of equipment mount points on rack symbols in Rack Elevation views
- The location of card/module slot insertion points on chassis symbols in Rack Elevation views

- The location and types of ports on any kind of connectable equipment in Schematic Detail views.

If you don't do this, users will not be able to interact with the equipment using the tools covered earlier. Fortunately, supplying the required information is a fairly simple task.

In this step, we shall limit ourselves to looking only at how to encode equipment mount points (since these are generally the only ones that apply to racks). We'll look at the others when we configure a chassis in Step 103.

Let us suppose that a new type of rack with 50 mount points is to be added to the ISP equipment dictionary. The rack is nothing but a passive frame (i.e. it has no network electronics) so it will only appear in floorplan and elevation views, and not in schematics. We will therefore only need two symbols. Luckily, we almost never have to create new symbols from scratch, as we can copy existing symbols already in the library.

Floorplan Symbol

Floorplan symbols do not contain any intelligence. We just need to decide how we want the rack to appear in a floorplan. Usually we just copy or slightly modify an existing floorplan symbol for this. In this case, let's just copy the appearance of an existing type of rack and not change it.

The equipment library contains a block for a 49-RU EMCOR rack. We will use this without any modifications as the basis for the floorplan symbol that will represent our new rack type.

To do this:

1. Locate the file **fr_emcor_32x23.dwg** in the **blklib\floorplan** directory.
2. Make a copy of this file in the same directory and call it **fr_generic_50ru.dwg**.

This completes preparation of the floorplan symbol for the rack type.

Elevation Symbol

For the rack's elevation symbol, we will again use the 49-RU EMCOR rack as our template, but this time we need to make some changes. Specifically, we need to add an extra RU and register a new mount point.

Before we change the file, let us examine what we have in it.

1. Locate the file **er_emcor-19x49ru_rack.dwg** in the **blklib\elevation** directory.
2. Make a copy of this file in the same directory and call it **er_generic_50ru.dwg**.
3. Open **er_generic_50ru.dwg** as a native drawing in AutoCAD using AuotCAD's **File > Open** command.

The file depicts the front vertical view of the rack, with each rack unit measured off, as shown in Figure 238, below. However, in addition to the visible content of this symbol, there are tiny circles at each mount point location, shown (slightly exaggerated in size) in the detail on the right of Figure 238. These circles are actually blocks with attributes that tell SPATIAL*net* FM where equipment can be mounted on the rack, and supply the RU number of each mount point. This information is used to

configure the mount point numbers available on the rack, and in elevation views to provide the "drag and drop" functionatliy.

Note: You cannot mount equipment at a rack location where there is no marker block present.

Each of the marker blocks is actually an instance of the block **_snet_rack_mount_point**. This block has a single attribute, **MOUNT_POINT_ID**, whose value is just the RU number of the mount point. The DWG file defining the **_snet_rack_mount_point** block is supplied with SPATIAL*net* FM, and can be found in the **symbol\smt** directory.

To modify the existing elevation symbol so that it contains an additional RU, you must therefore:

4. Modify the graphical representation of the rack to include an extra RU. Normal AutoCAD drawing functions can be used for this, as the file is just a plain DWG.
5. INSERT an additional **_snet_rack_mount_point** with a **MOUNT_POINT_ID** value of 50 at the location at which equipment mounted at RU 50 will be displayed.

Note: Do not copy an existing _snet_rack_mount_point, as duplicate mount point numbers may result. This will prevent the block from being registered. Use the AutoCAD INSERT command (or any equivalent way of inserting a block).

6. Save the file when done (make sure it is written into the **blklib\elevation** directory).

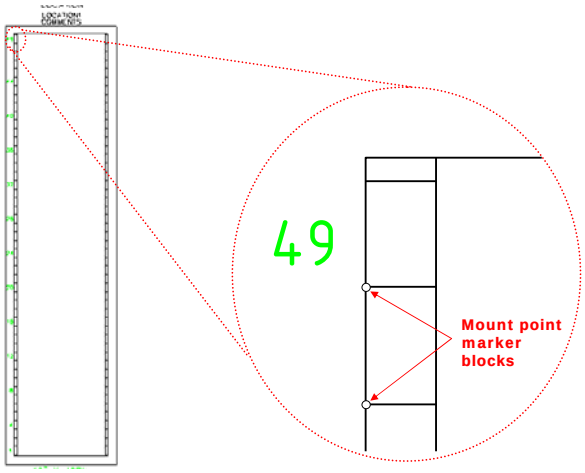


Figure 238 – Rack Elevation symbol for rack, showing mount point marker blocks (slightly exaggerated in size)

Now that the mount point marker blocks have been entered into the DWG file correctly, the new symbol must be registered with SPATIAL*net* FM so the mount point information can be loaded into the database. This will complete configuration of the symbology for the new rack type. To do this:

7. Ensure the file **er_generic_50ru.dwg** is open and in the currently active AutoCAD window.
8. Run the menu command
   FM ISP > System Configuration > Rack Elevation Drawings > Register RE symbol.
9. A list of the mount points that SPATIAL*net* FM has found in the drawing will be displayed.
10. Click the **Register** button to load the rack configuration information into the database.

Note: As of SPATIALnet FM 4.0, the list of marker blocks displayed on the ISP RE Symbol Creation panel was limited to about 40 entries. Therefore, some marker blocks which are in the drawing may not appear in the list. However, they will be registered correctly. To verify this, run the menu command: FM ISP > System Configuration > Rack Elevation Drawings > Query RE symbol after the symbol has been registered, and display the Details of the selected rack elevation configuration. All mount points should be present.

Equipment Dictionary Entry

Now that the symbology is complete, we can add the new rack type to the equipment dictionary. Since all of the configuration information for the rack has already been entered graphically and registered with the database, creating the equipment dictionary entry becomes very simple.

To do this:

11. Run the menu command **FM ISP > System Configuration > Equipment Dictionary > Add Entry…**
12. Fill in the fields with the values shown below (note the reference to the symbols we just created. Also note there is no representation of the rack in a schematic view).

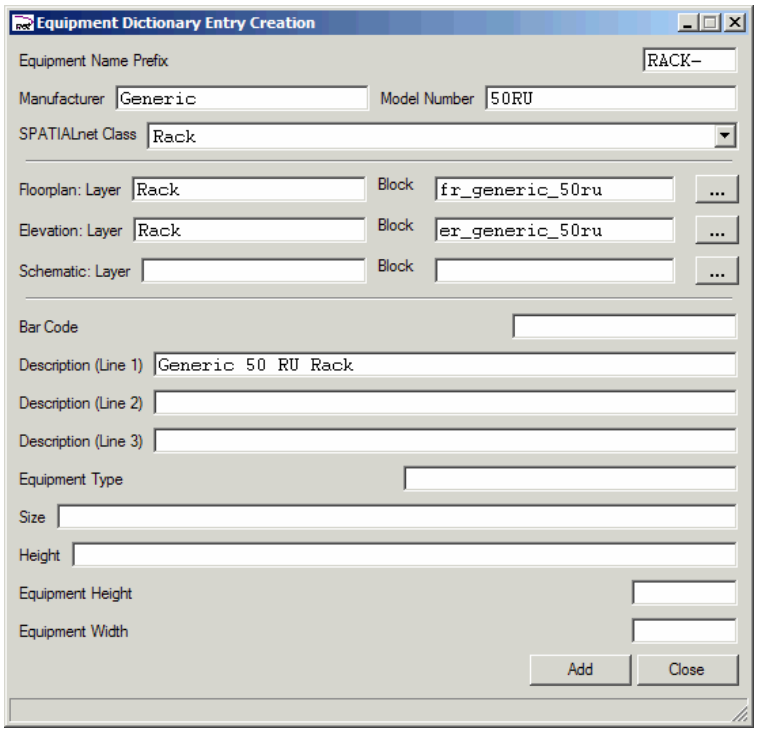


Figure 239 – Equipment Dictionary Entry Creation form for new rack.

13. Click the Add button shown in Figure 239 to create the new equipment dictionary entry. The system will display a message with the new Catalog ID assigned to the rack type.

This completes specification of the new rack type. Instances of this equipment can now be added to the database.

Step 103: Configuring a Chassis Type

The process for configuring new chassis types is almost identical to that for racks, so the background provided in Step 102 applies equally well here (you may want to review this now). The only additional features we will contend with that were not covered in Step 102 are the addition of chassis mount points and the definition of a schematic representation of the equipment which includes the specification of its ports. Conceptually, however, these are also almost identical to the configuration process covered in Step 102.

As with our approach in Step 102, we will begin the process by modifying an existing chassis, in this case a Cisco 2611 router. For the purposes of this exercise, let us suppose that a new model called the 26XX has been released with an additional slot for a PC card on the front panel. Configuration of chasses follows the same

procedures as the configuration of racks in Step 102, that is, configure the symbology, register it, and then add an equipment dictionary entry.

Intelligent Symbology

We want to begin with the existing 2611 chassis and add that extra slot. As with mount points on racks (see Step 102, above), chasses use a marker block to indicate the location and number of each of the card insertion slots. The marker block is called **_snet_chassis_mount_point3**, and can be found in the **symbol\smt** sub-directory of the SPATIALnet installation folder. This block has one attribute, **MOUNT_POINT_ID** which stores the number of the slot represented by the marker.

To create the symbology for the new chassis:

1. Open the drawing **ec_cisco_2611_chassis.dwg** in the **blklib\elevation** subdirectory of the SPATIALnet installation folder.
2. Save this in the same directory as **ec_cisco_26XX_chassis.dwg**.
3. Draw an extra rectangle representing the new PC Card bay on the front panel.
4. INSERT a **_snet_chassis_mount_point3** block and set the **MOUNT_POINT_ID** to **03**.
5. Save the drawing.
6. Run the menu command
   FM ISP > System Configuration > Rack Elevation Drawings > Register RE symbol.
7. A list of the mount points that SPATIAL*net* FM has found in the drawing will be displayed.
8. Click the **Register** button to load the chassis configuration information into the database.

This completes the process for creating the symbology for the new chassis type.