QualNet®

QualNet 7.1
Product Tour

August 2013

SCALABLE Network Technologies, Inc.

600 Corporate Pointe, Suite 1200
Culver City, CA 90230

+1.310.338.3318 TEL
+1.310.338.7213 FAX

SCALABLE NETWORK TECHNOLOGIES

SCALABLE-NETWORKS.COM

SCALABLE Network Technologies, Inc.

600 Corporate Pointe, Suit 1200

Culver City, CA 90230

+1.310.338.3318 TEL
+1.310.338.7213 FAX

SCALABLE-NETWORKS.COM

Table of Contents

1 Introduction

The purpose of this product tour is to introduce some of the main features and capabilities of QualNet.

This product tour is organized as follows:

- Chapter 2 demonstrates the modeling and analytical capabilities of QualNet by walking through a pre-defined scenario. We will do the following:
  - Start QualNet GUI and open a WiFi scenario that has already been created.
  - Explore the components and properties of the scenario.
  - Run and visualize the scenario and observe its runtime behavior.
  - Analyze the results obtained by running the simulation.
- Chapter 3 describes how to create network scenarios in QualNet. We walk through the steps of creating a scenario that is similar to the WiFi scenario used in Chapter 2.

This tour assumes that QualNet 7.1 has been installed at the default location.

- For Windows systems, the default installation directory is C:/scalable/qualnet/7.1.
- For Linux systems, the default installation directory is ~/scalable/qualnet/7.1.

Notes:
1. Refer to *QualNet Installation Guide* for help with installing QualNet.
2. Refer to *QualNet User's Guide* for detailed instructions for using QualNet.
3. For technical help on QualNet, please contact QualNet Support at support@scalable-networks.com or visit our Support website at support.scalable-networks.com.

2 Simulation Demo

2.1 Opening a Scenario in QualNet GUI

This section shows how to start the QualNet GUI, describes the GUI components, and then shows how to open a scenario.

Starting GUI on Windows

To start the QualNet GUI on a Windows system, do one of the following:

- Double-click the following icon on the desktop (this option is available only if you chose to install desktop shortcuts during installation):

OR

- Open a command window and type the following commands:

```
cd %QUALNET_HOME%\bin
QualNetGUI.exe
```

Starting GUI on Linux

To start the QualNet GUI on a Linux system, do one of the following:

- Double-click the following icon on the desktop (this option is available only if you installed QualNet using the installer's GUI and chose to install desktop shortcuts):

OR

- Open a command window and type the following commands:

```
cd $QUALNET_HOME/bin
./QualNetGUI
```

QualNet GUI Components

The QualNet GUI has four components: Architect, Analyzer, Packet Tracer, and File Editor (see the Components Toolbar below).

- **Architect** is used for creating scenarios (in *Design* mode) and running simulations (in *Visualize* mode).
- **Analyzer** is used for analyzing simulation results.
- **Packet Tracer** is used for analyzing packet traces obtained by running simulations.
- **File Editor** is used for editing text files.

When you start the QualNet GUI, by default it opens in the Design mode of Architect. Figure 2-1 shows the that window is displayed when the QualNet GUI starts.

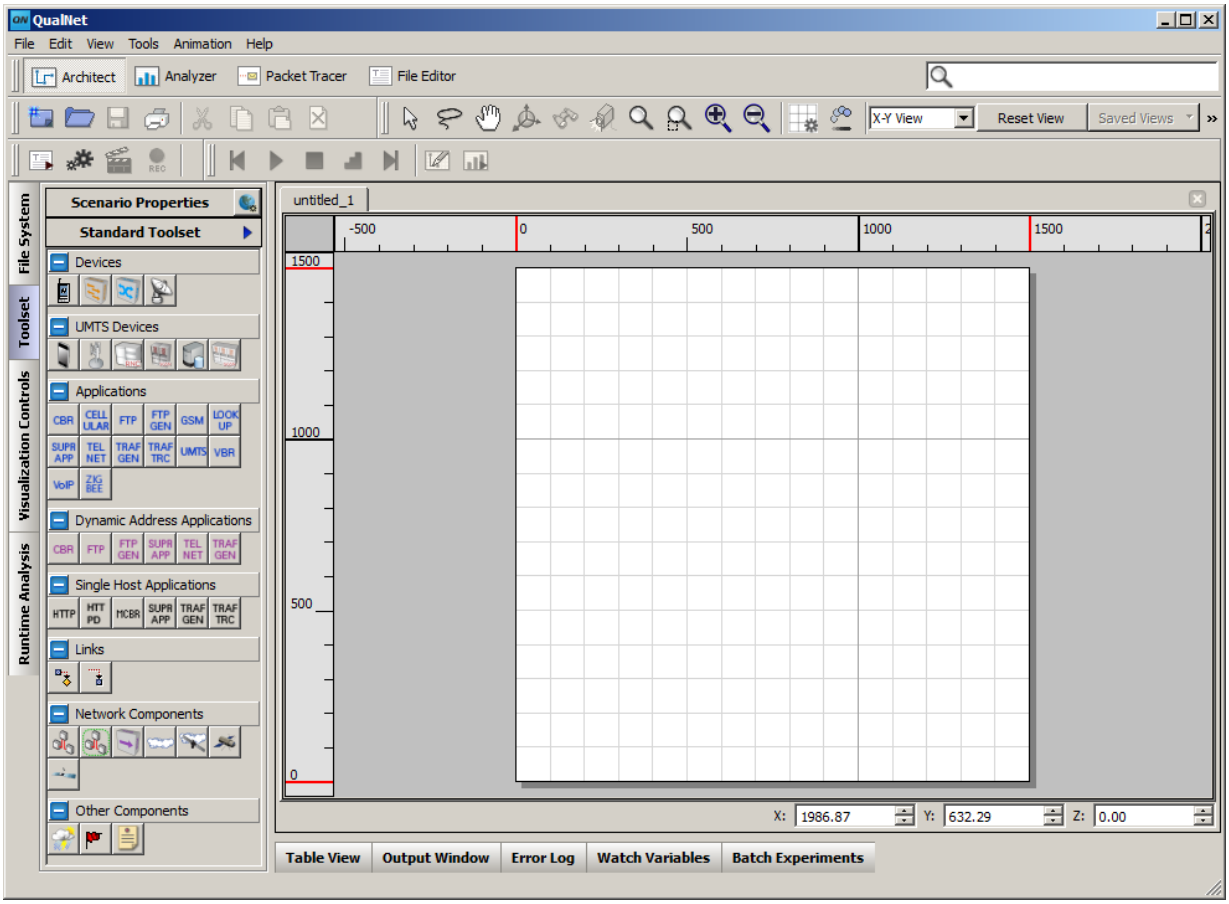


FIGURE 2-1. QualNet GUI: Initial Display

Loading the Demo Scenario

For this example, we will load the WiFi Demo scenario.

1. To load the demo scenario, go to the **File** menu and select **Open File**.

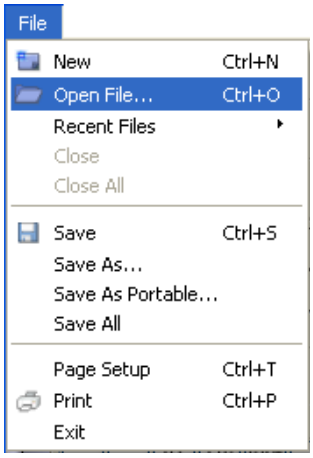


FIGURE 2-2. Open File from File Menu

A file selector window opens.

- On Windows, navigate to the folder C:/scalable/qualnet/7.1/scenarios/demo/WiFiDemo.
- On Linux, navigate to the folder ~/scalable/qualnet/7.1/scenarios/demo/WiFiDemo.

2. Select **WiFiDemo.config** and click **Open**.

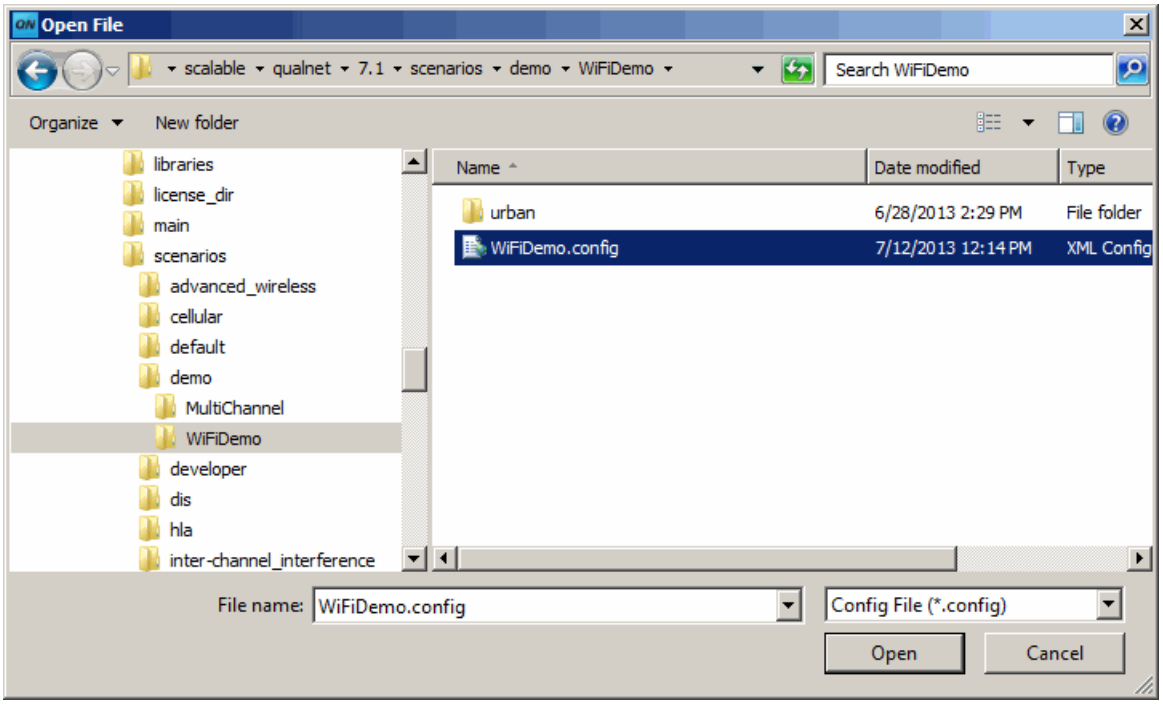


FIGURE 2-3. Select Demo Scenario File

The following WiFi demo is displayed on the canvas (main work area) of the QualNet GUI:

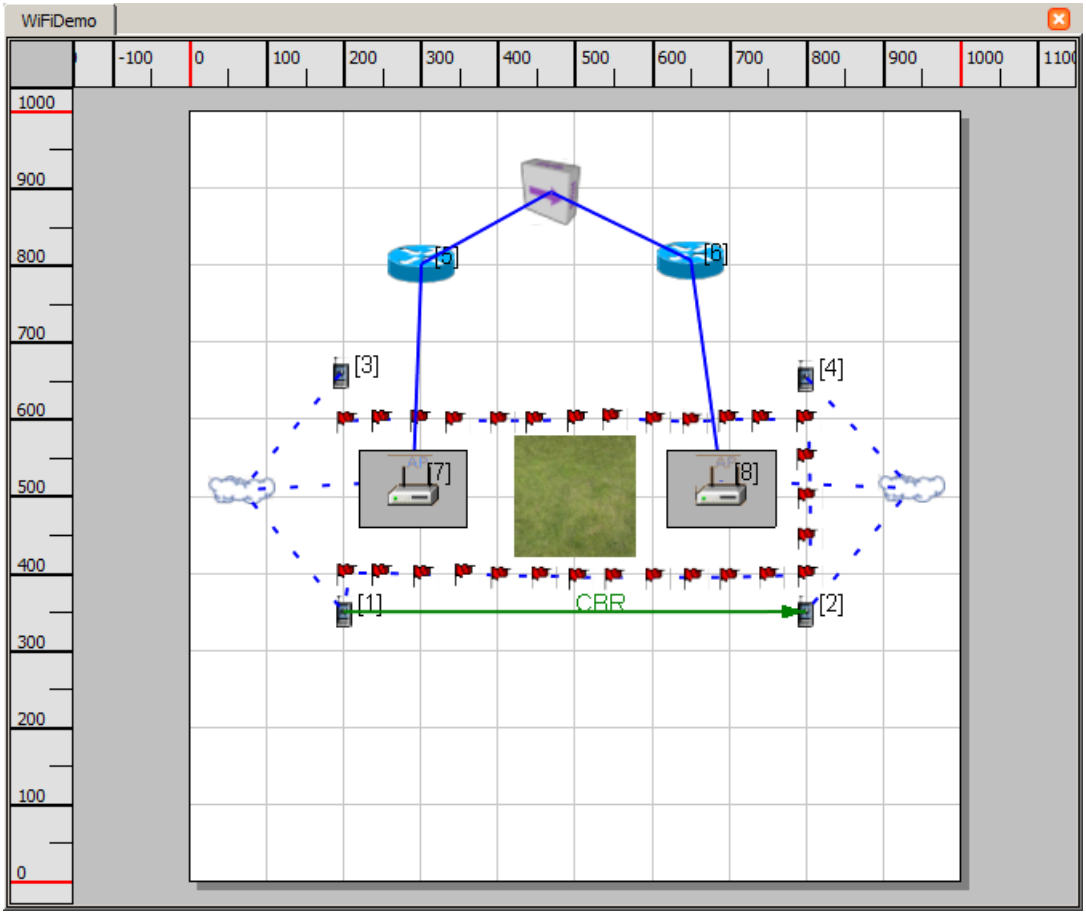


FIGURE 2-4. WiFi Demo

2.2 Scenario Description

The demo scenario is a simple WiFi scenario:

- There are two WiFi access points and four WiFi mobile stations.
- The two access points are connected via a simple wired network. Each access point is connected to a router via a wired point-to-point link. The two routers are in turn connected via an Ethernet hub.
- A simple urban terrain with two buildings and one park is used.
- Mobile station node 1 is the source of a UDP flow sending packets to another mobile station, node 3.

During the simulation, node 1 moves around in the field from the coverage area of one access point to that of the other. As node 1 moves, it switches association between the two access points.

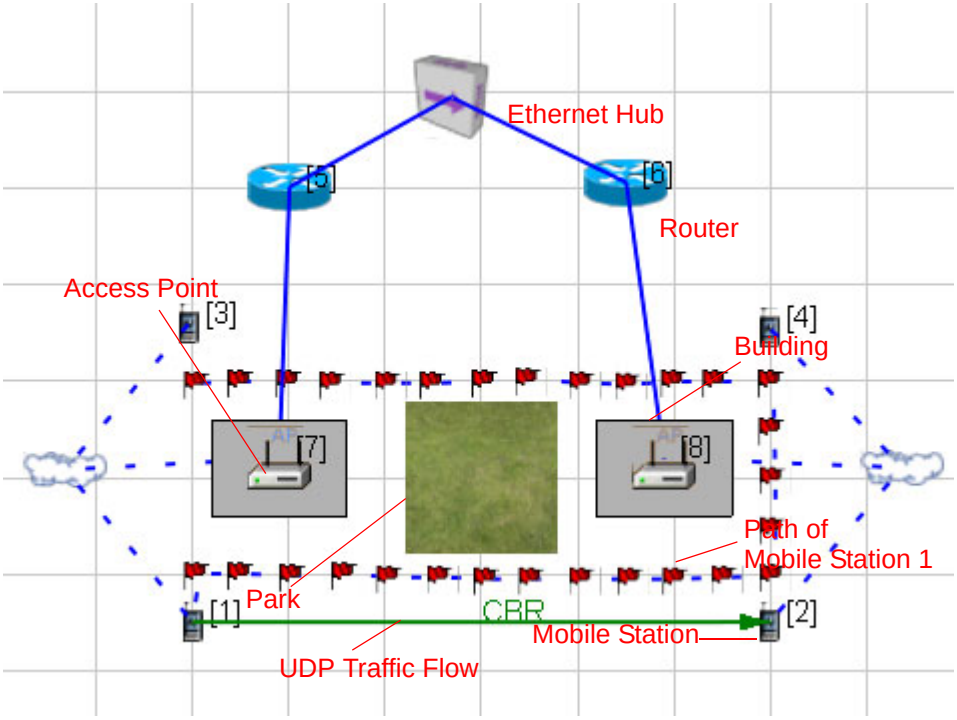
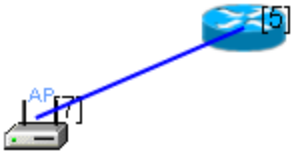
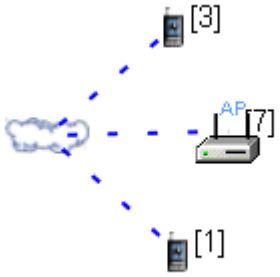


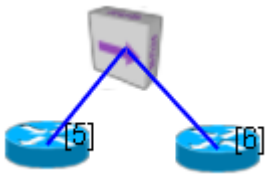
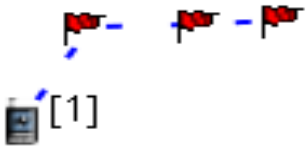
FIGURE 2-5. Scenario Details (X-Y View)

The scenario components shown in Figure 2-5 are described next.

Icons and Links

In the scenario, the icons and lines represent the following:

Icons and Links	Description
	Nodes (communication devices). Note that the number inside the square brackets is the node ID (i.e., [3], [5], [7]).  represents a mobile station.  represents an access point.  represents a router.
	A solid blue line represents a wired link, such as a cable.
	The cloud icon represents a wireless subnet and the dashed lines connecting nodes to the cloud indicate that the nodes are part of the wireless subnet. All nodes connected to a cloud icon belong to the same IP subnet and can communicate by means of radios.

Icons and Links	Description
	The hub icon represents a wired Ethernet subnet. Nodes connected to a wired subnet can communicate with each other directly.
	Red flags indicate the path that a mobile station takes during the simulation (in this case, for mobile station 1).
	The green arrow represents a unicast traffic session between two nodes (in this case, between mobile stations 1 and 2), and the CBR label indicates the type of traffic session.

Terrain

The grey rectangles in Figure 2-5 represent buildings. (By default, the scenario opens in X-Y view and buildings appear as rectangles when viewed from the top.) Similarly, the green rectangle represents a park. See Figure 2-7 to view the terrain in 3D view.

2.3 Navigating within the Scenario

The scenario opens in 2D (X-Y plane) view by default.

To change the to 3D view, do the following:

- Select **3D View** from the drop down list in the **View** toolbar.

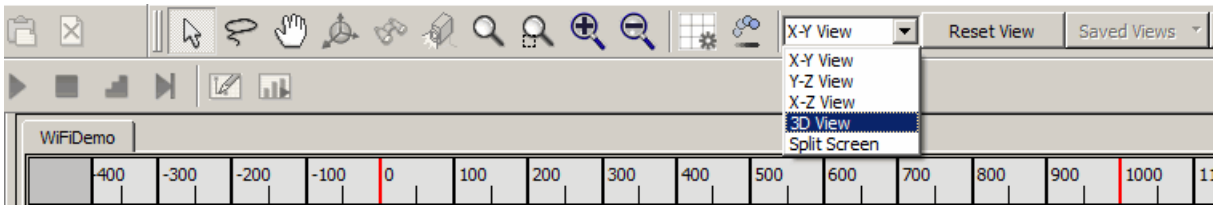


FIGURE 2-6. Switch to 3D View

The scenario view changes to the following:

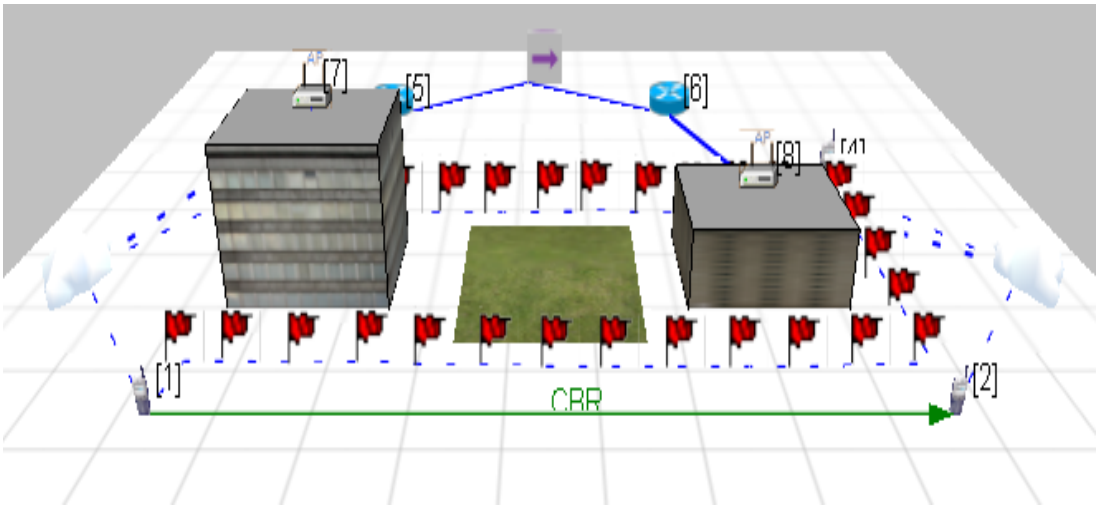


FIGURE 2-7. Scenario in 3D View

Use the navigation buttons described below to zoom in and out, rotate, and pan. (These buttons are located in the **View** toolbar.)

To zoom out: Click the button. The cursor changes to . Click on the canvas and drag the mouse down.

To zoom in: Click the button. The cursor changes to . Click on the canvas and drag the mouse up.

To pan: Click the button. The cursor changes to . Click on the canvas and use the cursor with the left mouse button pressed to pan through the scenario.

To rotate: Click the button. The cursor changes to . Click on the canvas and use the cursor with the left mouse button pressed to change the angle of view.

Use these buttons to explore the scenario topology. Locate all four mobile stations in the scenario. (Nodes 2 and 4 are hidden by the buildings in the default view.)

Changing Scenario View

The following is the display after zooming in on the left building and changing the angle of view:

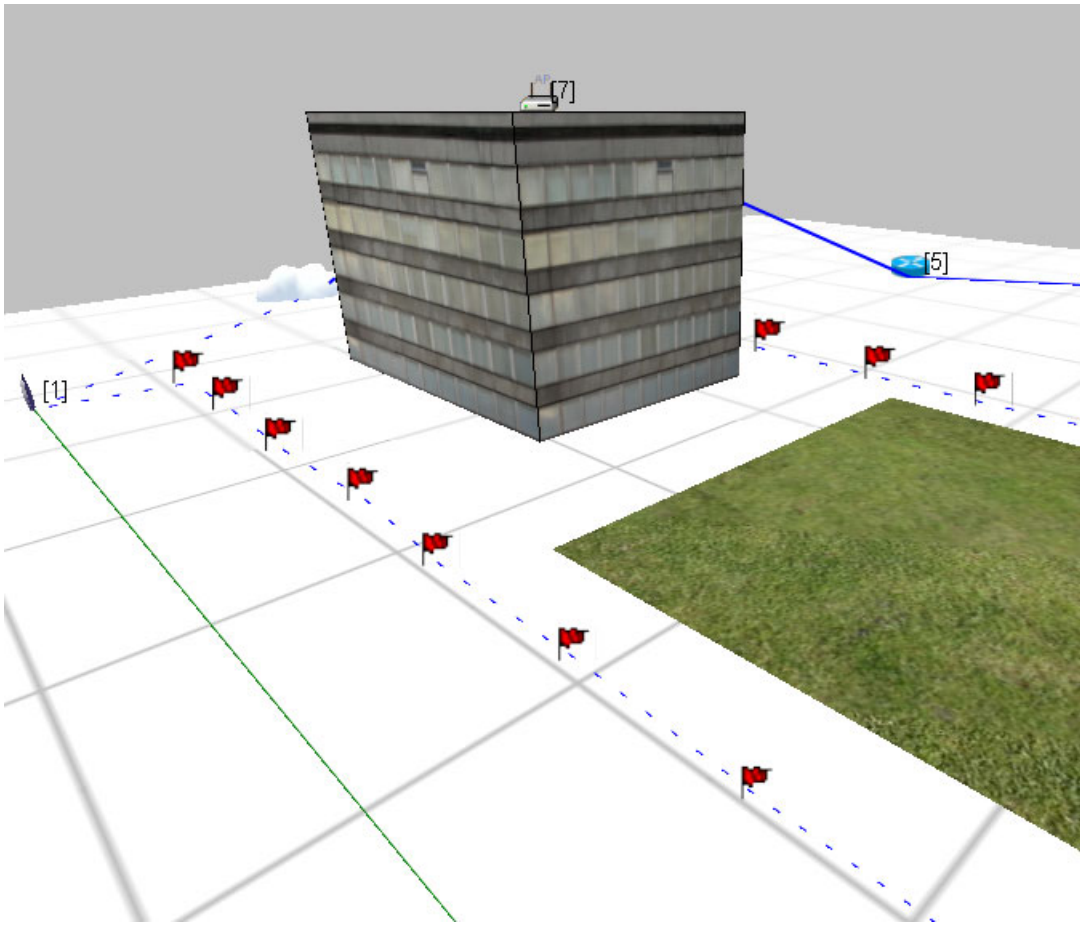


FIGURE 2-8. Zoom View

You can restore the original 3D view by clicking the Reset View button in the **View** Toolbar at any time.

To continue with the tour, change the view back to the X-Y view.

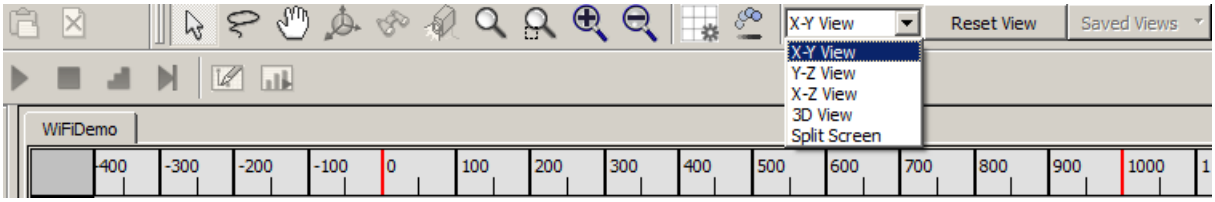


FIGURE 2-9. Switch to X-Y View

2.4 Scenario Configuration

General parameters for the simulation and for the different components are configured in various Properties Editors.

In this part of the tour, we will show where some of these properties are set. (We will not modify any properties here.)

Note: If you prefer, you can skip this section and go directly to Section 2.5 to run the scenario. You can return to this section later if you want to explore how simulation parameters are set.

General Simulation Parameters

General simulation parameters, such as simulation time, and scenario-wide properties, such as channel frequencies and terrain properties, are set in the Scenario Properties Editor.

To open the Scenario Properties Editor, do one of the following:

- Click the **Scenario Properties** button in the Toolset panel.

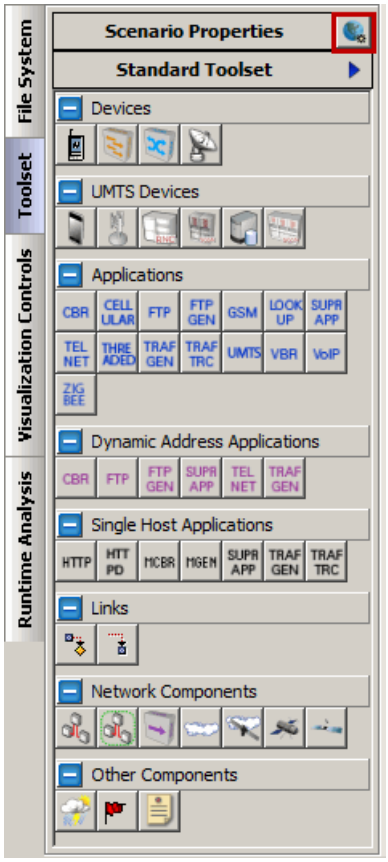


FIGURE 2-10. Scenario Properties Button and Standard Toolset

OR

- Select **Scenario Properties** from the **Tools** menu.

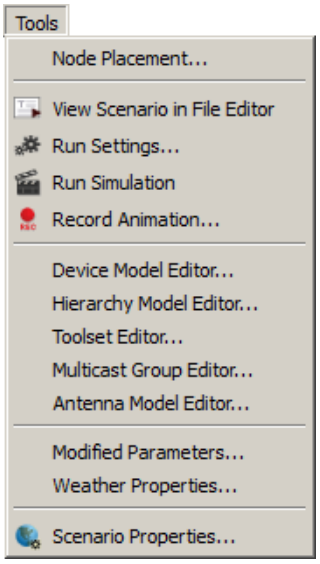


FIGURE 2-11. Open Scenario Properties Editor from Tools Menu

This opens the Scenario Properties Editor shown below.

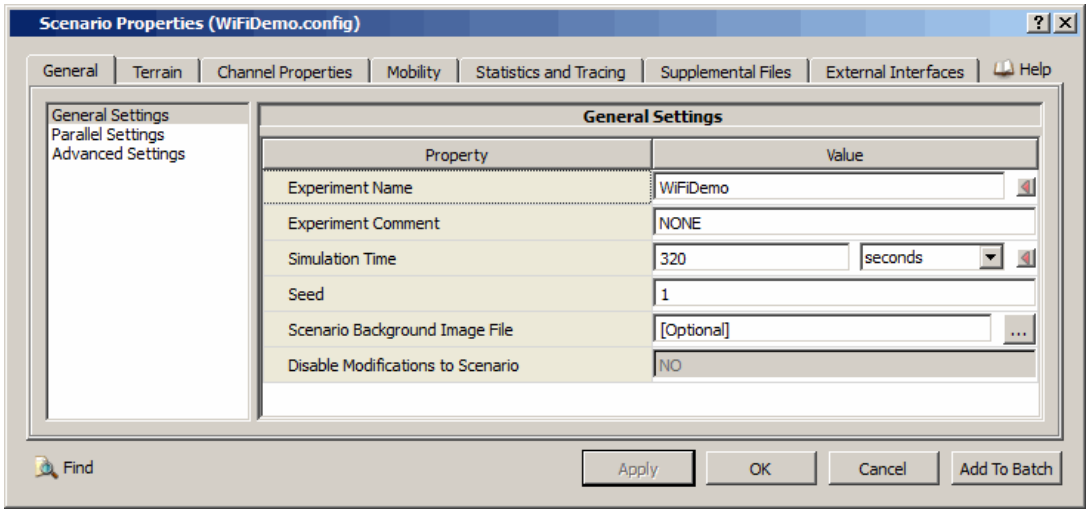


FIGURE 2-12. Scenario Properties Editor

General Properties

Properties are organized under different tabs. The left panel of the **General** tab lists several property groups. Selecting a group in the left panel displays the properties in that group in the right panel along with their values. In the above figure, the **General Settings** group in the **General** tab has been selected. One of the properties in this group is **Simulation Time**, which has been set to *320 seconds*, indicating that the simulation will run for 320 seconds.

Open the other tabs of the Scenario Properties Editor to get an idea of the types of properties set here.

Node Properties

Properties specific to a node (i.e., a communicating device, such as an access point, mobile station or router) are configured in the Default Device Properties Editor. A node's characteristics (including the icon to represent the node on the canvas) are determined by the values assigned to the properties in the Default Device Properties Editor for that node.

You can open the Default Device Properties Editor for a node from the **Table View** panel or from the canvas.

- To open the Default Device Properties Editor for node 7 (which is configured to be an access point), from the **Table View** panel, do the following:

  1. Click on the **Table View** button at the bottom of the display.

FIGURE 2-13. Table View Button

This opens the **Table View** panel. All components in the scenario are listed in this panel under different tabs. All nodes in the scenario are listed in the **Nodes** tab. Note that a name is assigned to each node.

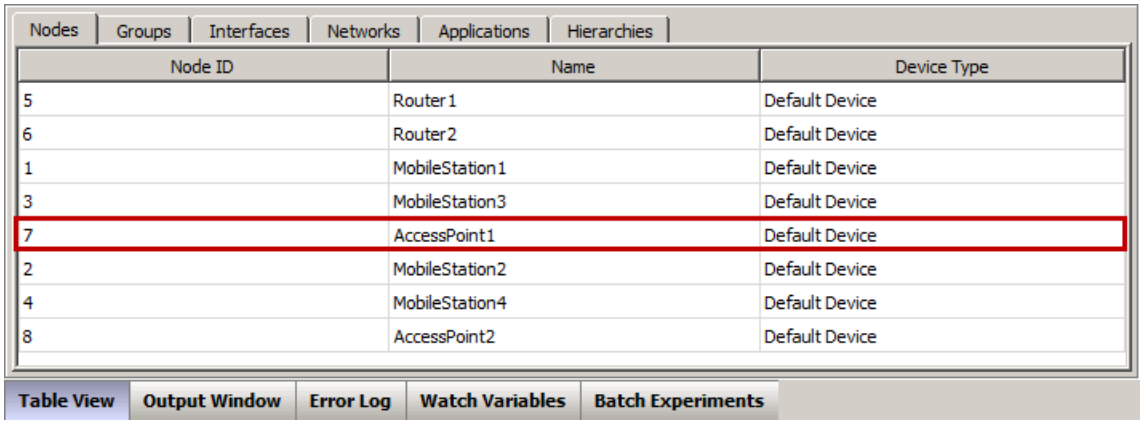


FIGURE 2-14. Table View Showing Nodes

2. Double-click on the row for Node ID 7. This opens the Properties Editor for AccessPoint1 (see Figure 2-15).

Note: When you select a node in the table, its icon gets highlighted on the canvas.

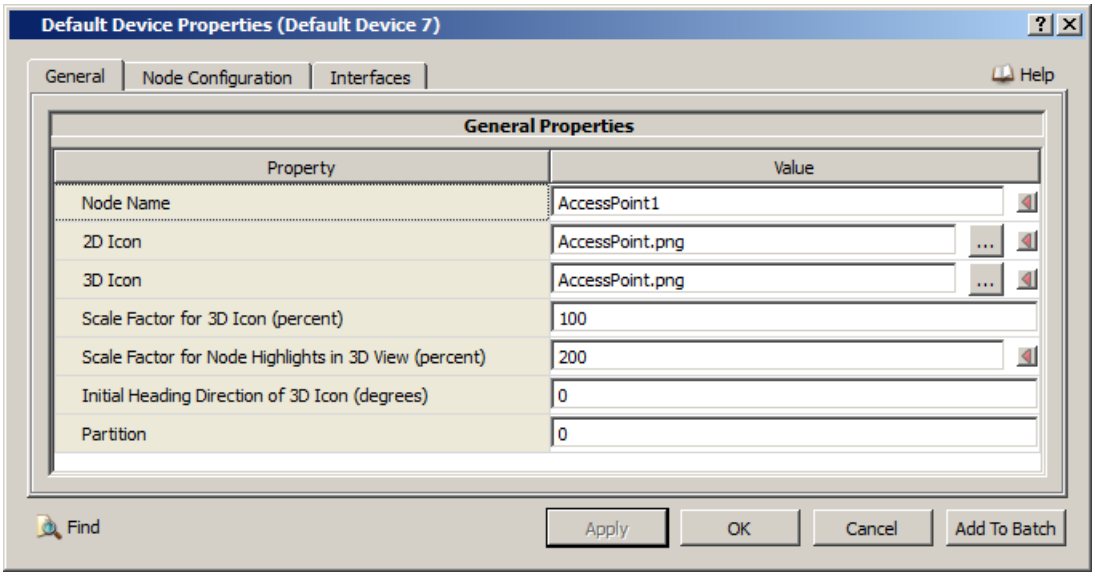


FIGURE 2-15. Properties Editor for AccessPoint1

- To open the Default Device Properties Editor for a node from the **canvas**, do the following:
  - Click the **Select** button on the **View** toolbar.
  - Right-click a node icon on the canvas and select **Properties** from the menu.

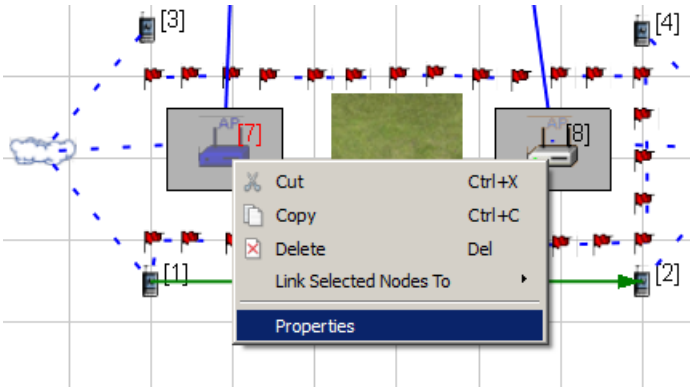


FIGURE 2-16. Open Default Device Properties Editor from Canvas

How Properties are Organized

Properties are organized under different tabs and groups within tabs. Select the **Node Configuration** tab and click on **Routing Protocol** in the left panel. The routing-related parameters are displayed in the right panel (see Figure 2-17). For example, the **Routing Protocol IPV4** is set to *AODV*, indicating that the Ad-hoc On-demand Distance Vector routing protocol is used for IPv4 networks.

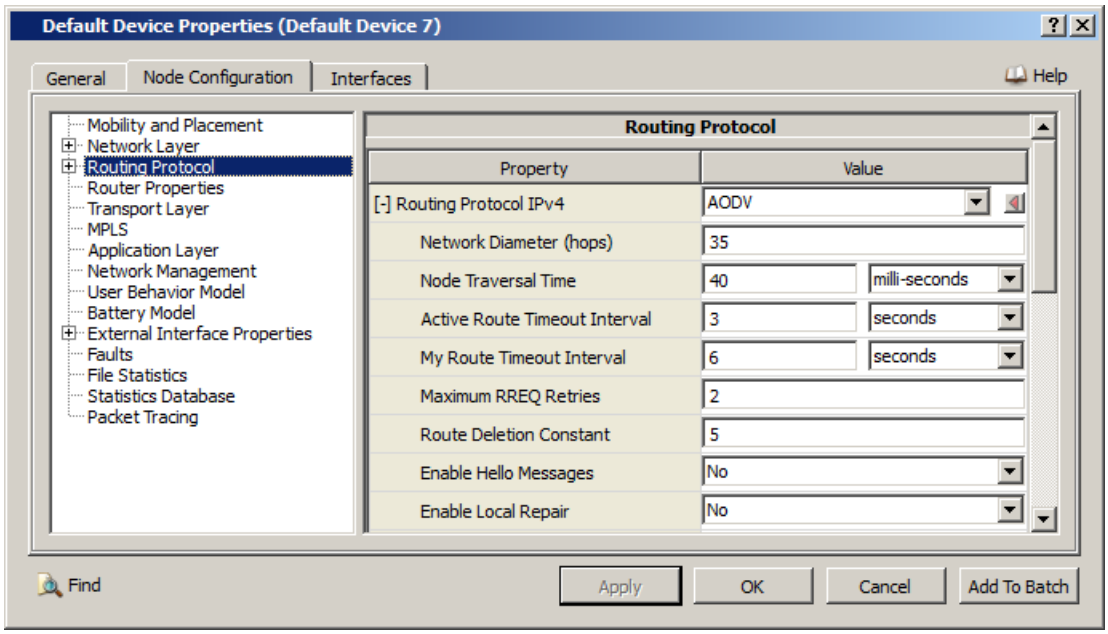


FIGURE 2-17. Routing Protocol Properties for AccessPoint1

When you click on different property groups in the left panel, properties belonging to each group and their values will be displayed in the right panel.

Interface Properties

Properties specific to an interface of a node are set in the Interface Properties Editor. A node can have one or more interfaces. Different interfaces of the same node can have different properties. For example, each of the nodes 7 and 8 has two interfaces: one to a wireless subnet and the other connecting it to a router. Each of the mobile stations (nodes 1 to 4) has a single interface to a wireless subnet.

We will now examine the properties of the interfaces of node 7.

1. Open the Default Device Properties Editor for node 7, as previously described.
2. Click on the **Interfaces** tab. The left panel shows the two interfaces of this node. Interface 0 is the interface to the wireless subnet. Interface 1 is the wired interface connecting the access point to the router.

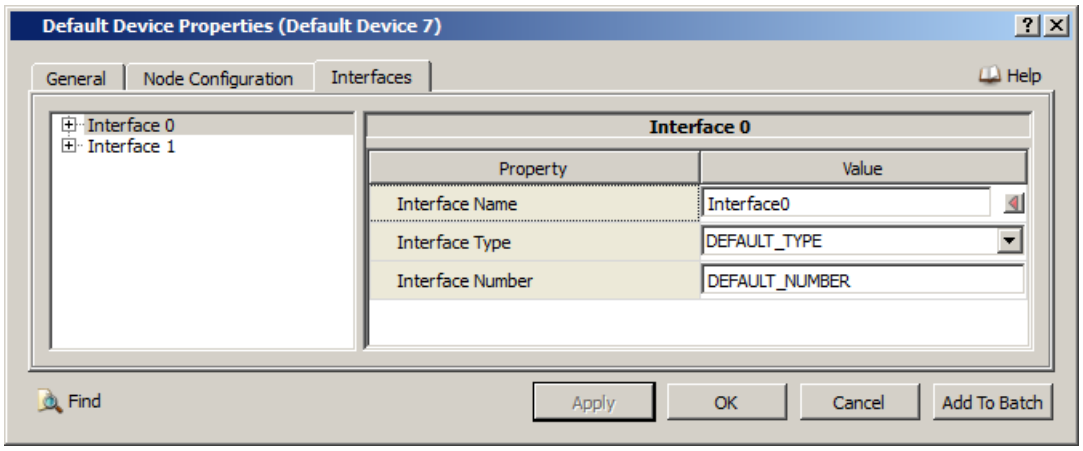


FIGURE 2-18. Interface Properties of AccessPoint1

3. Properties for each interface are organized in different groups. Click on the "**+**" sign before **Interface 0** to display the property groups for the wireless interface. Click on **MAC Layer**. The MAC layer properties for this interface are displayed in the right panel.

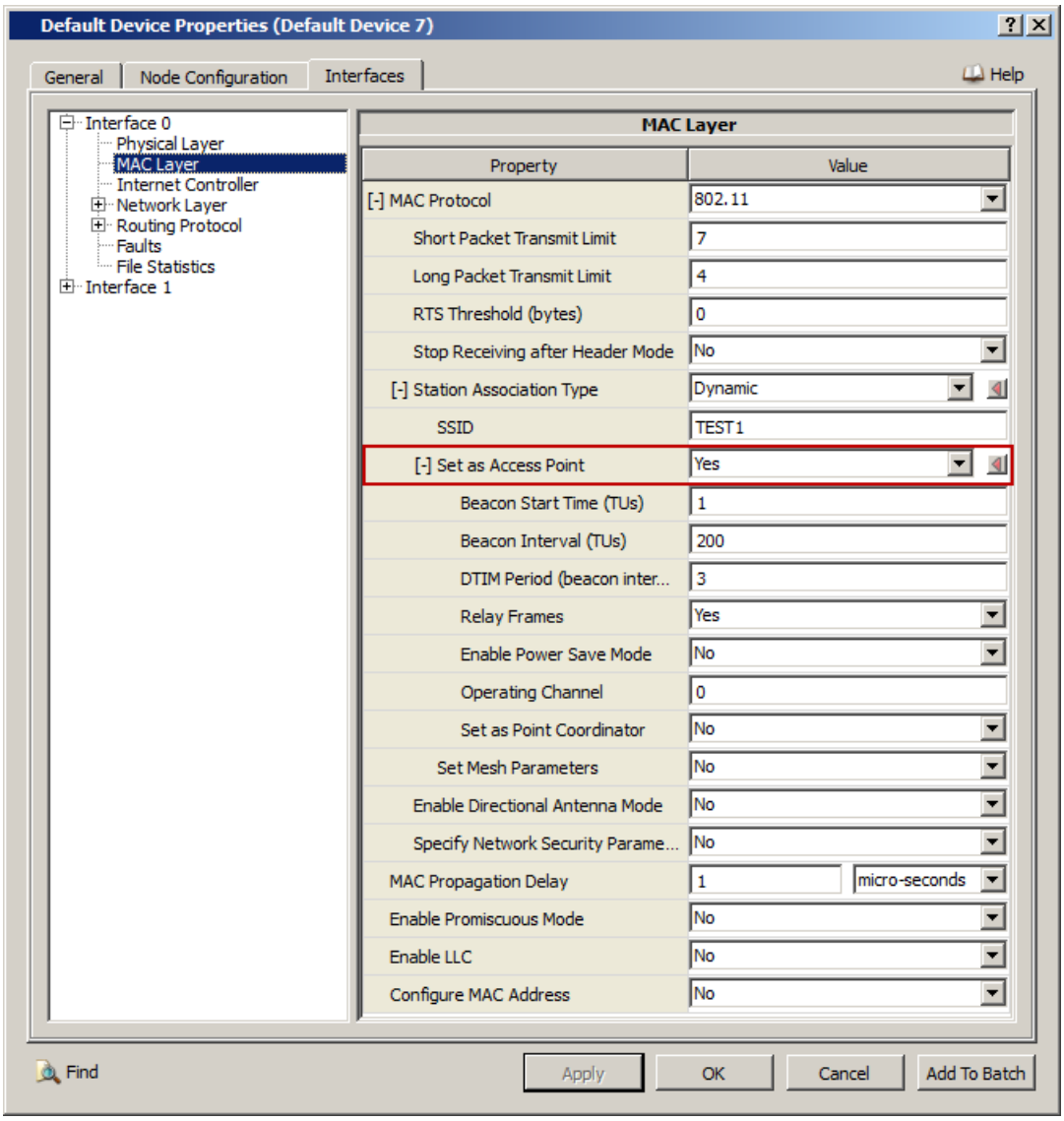


FIGURE 2-19. MAC Layer Properties

Note that **MAC Protocol** has been set to *802.11*, indicating that IEEE 802.11 MAC is used as the MAC protocol at this interface. Parameters for configuring IEEE 802.11 are also displayed.

Note that **Set as Access Point** is set to *Yes*, indicating that this is an access point.

4. Click on the other property groups in the left panel and examine the properties belonging to each group in the right panel.
5. Now click on the "**+**" sign before **Interface 1** to display the property groups for the wired interface. Explore the properties set for this interface. In particular, note that **MAC Protocol** has been set to a value (*Abstract Link MAC*) which is different from the value of **MAC Protocol** set for Interface 0.

We will now examine the properties of the interfaces of node 1 (which is a mobile station) and compare them with the properties of node 7 (which is an access point).

1. Open the Default Device Properties Editor for node 1, as described above.
2. Click on the **Interfaces** tab. This node has only one interface.
3. Click on the "**+**" sign before **Interface 0** to display the property groups for the wireless interface. Click on **MAC Layer**.
4. In the right panel, note that parameter **Set as Access Point** is set to *No*, indicating that this is a mobile station. (For node 7, this parameter is set to *Yes*.)

Wireless Subnet Properties

Properties specific to a wireless subnet are set in the Wireless Subnet Properties Editor. We will examine the Physical layer properties for a wireless subnet here.

To see the Physical layer properties of a subnet, do the following:

1. Open the **Table View** panel and go to the **Networks** tab.
2. All wired and wireless subnets and point-to-point links in the scenario are listed in the **Networks** tab. The nodes belonging to a subnet or connected by a link are also listed.

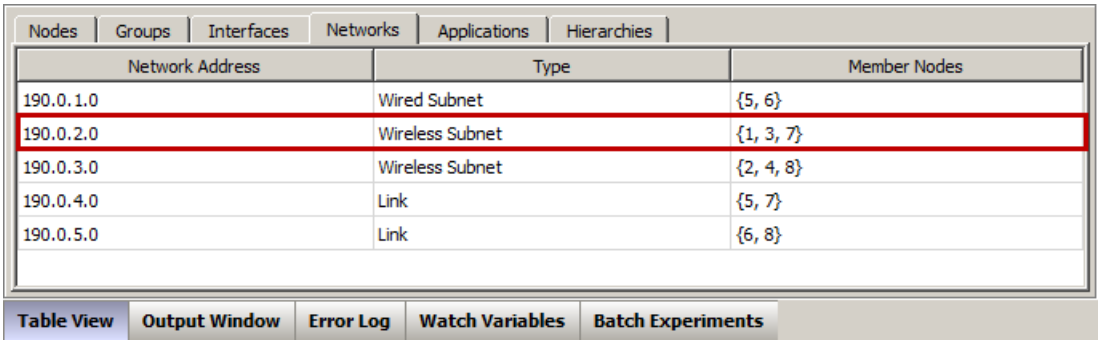


FIGURE 2-20. Table View Showing Networks

3. Double-click on the row for the wireless subnet with network address 190.0.2.0. This opens the Properties Editor for the left wireless subnet. (The subnet icon is highlighted on the canvas.)

4. Select the **Physical Layer** tab. All Physical layer properties for the subnet are set in this tab. Select the **General** property group in the left panel.

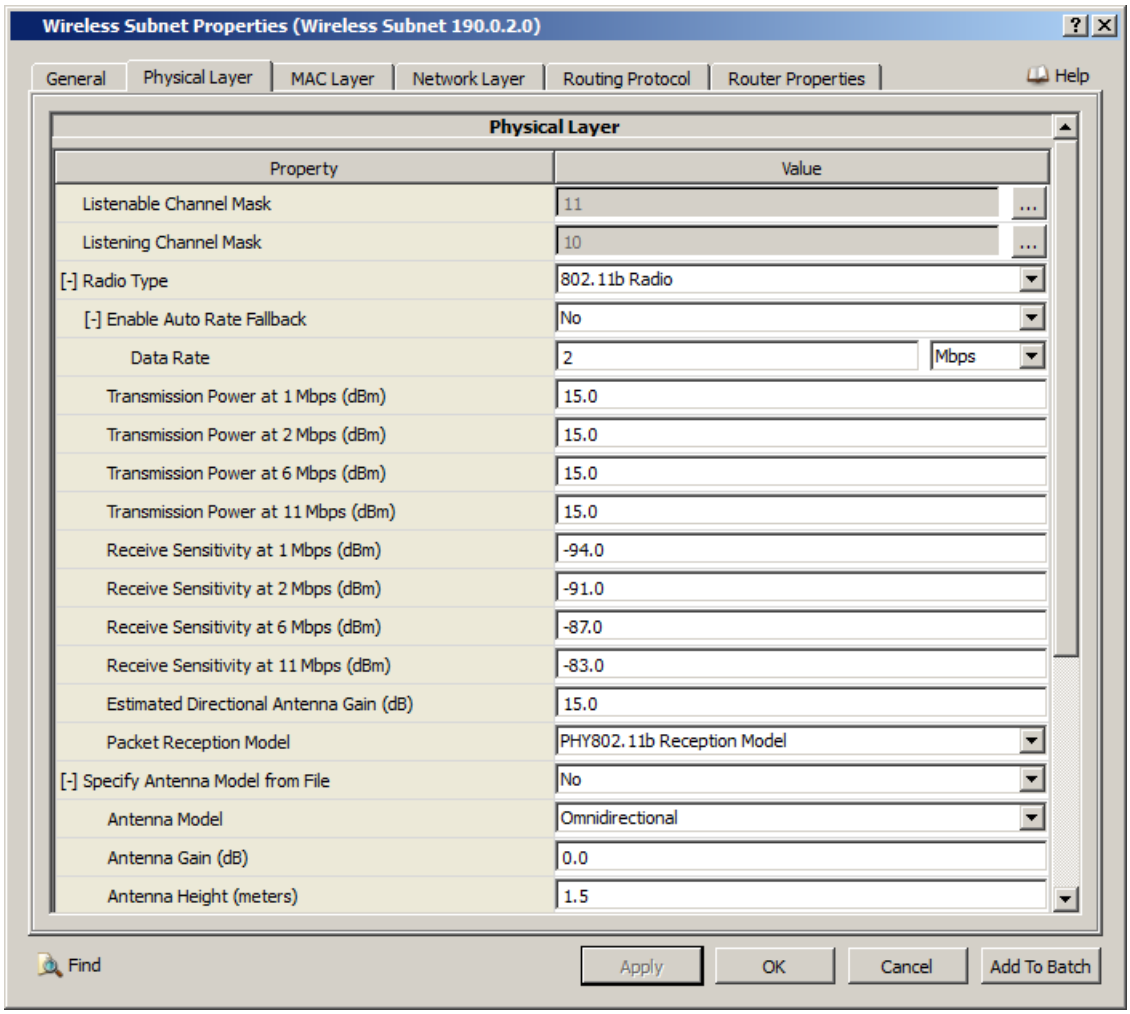


FIGURE 2-21. Physical Layer Properties for a Wireless Subnet

Note that **Radio Type** has been set to *802.11b Radio*, indicating that IEEE 802.11b PHY is used as the radio model for all interfaces belonging to this wireless subnet. Parameters for configuring IEEE 802.11b PHY are also displayed.

Application Properties

A green arrow on the canvas indicates an application session between a pair of nodes (in this scenario, between nodes 1 and 3). Packet traffic is simulated between the two nodes by means of an application called Constant Bit Rate (CBR) traffic-generator. We will examine the parameters of this traffic session here.

To see the properties of the CBR session, do the following:

1. Open the **Table View** panel and go to the **Applications** tab.
2. All application sessions in the scenario are listed in the **Applications** tab. In this example, there is only one traffic session.

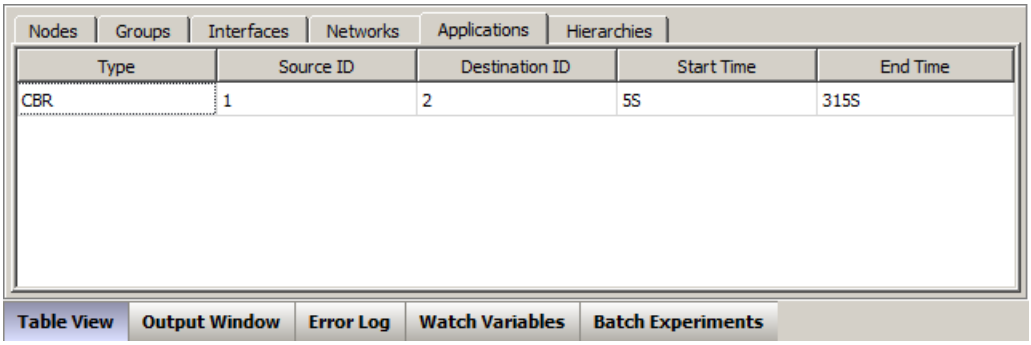


FIGURE 2-22. Table View Showing Applications

3. Double-click on the row for CBR. This opens the Properties Editor for the CBR session between nodes 1 and 2. (The arrow representing the application is highlighted on the canvas.)

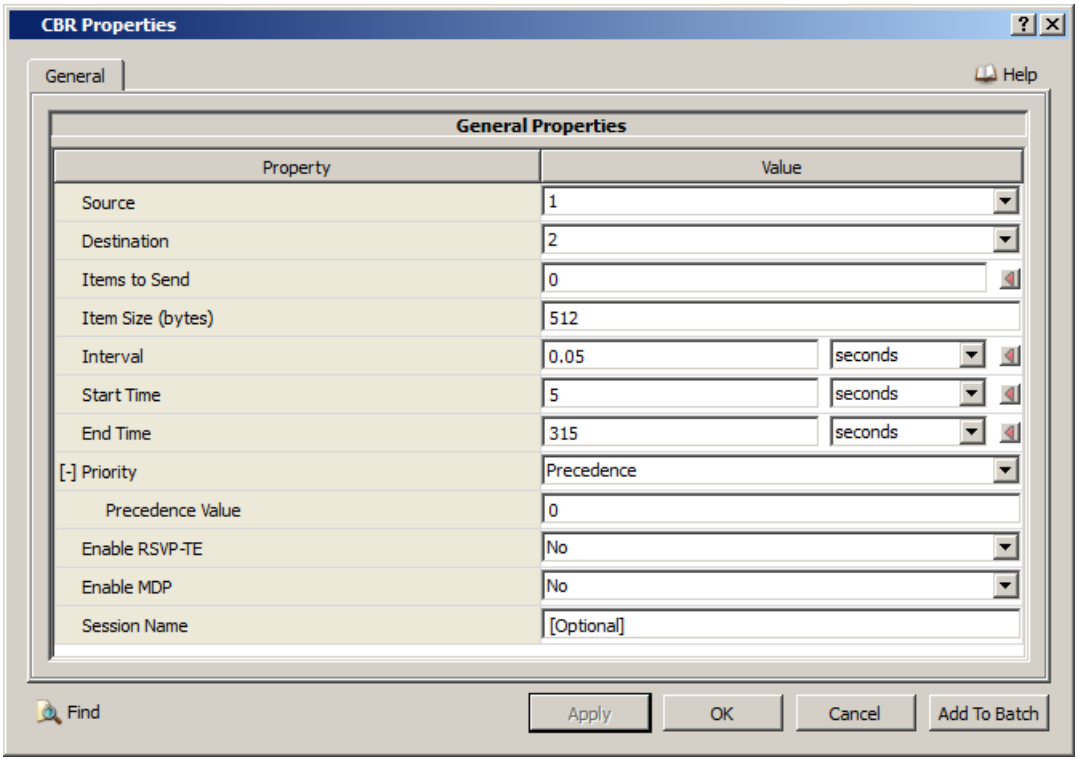


FIGURE 2-23. CBR Properties

All properties for the CBR session are displayed in this Properties Editor. For example, **Interval** has been set to *0.05 seconds*, indicating that a packet is sent from node 1 to node 2 every 50 milliseconds.

2.5 Running the Scenario

This section describes how to run the demo scenario and observe its runtime behavior.

Initialize the Simulation

To initialize the simulation click the **Initialize Simulation** button.

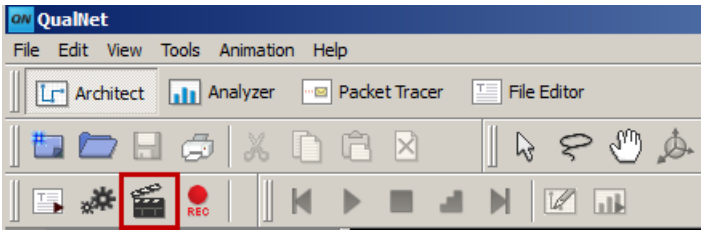


FIGURE 2-24. Initialize Simulation

This changes to the mode of Architect from *Design* mode to *Visualize* mode, as shown in Figure 2-25. The left panel changes to **Visualization Controls**.

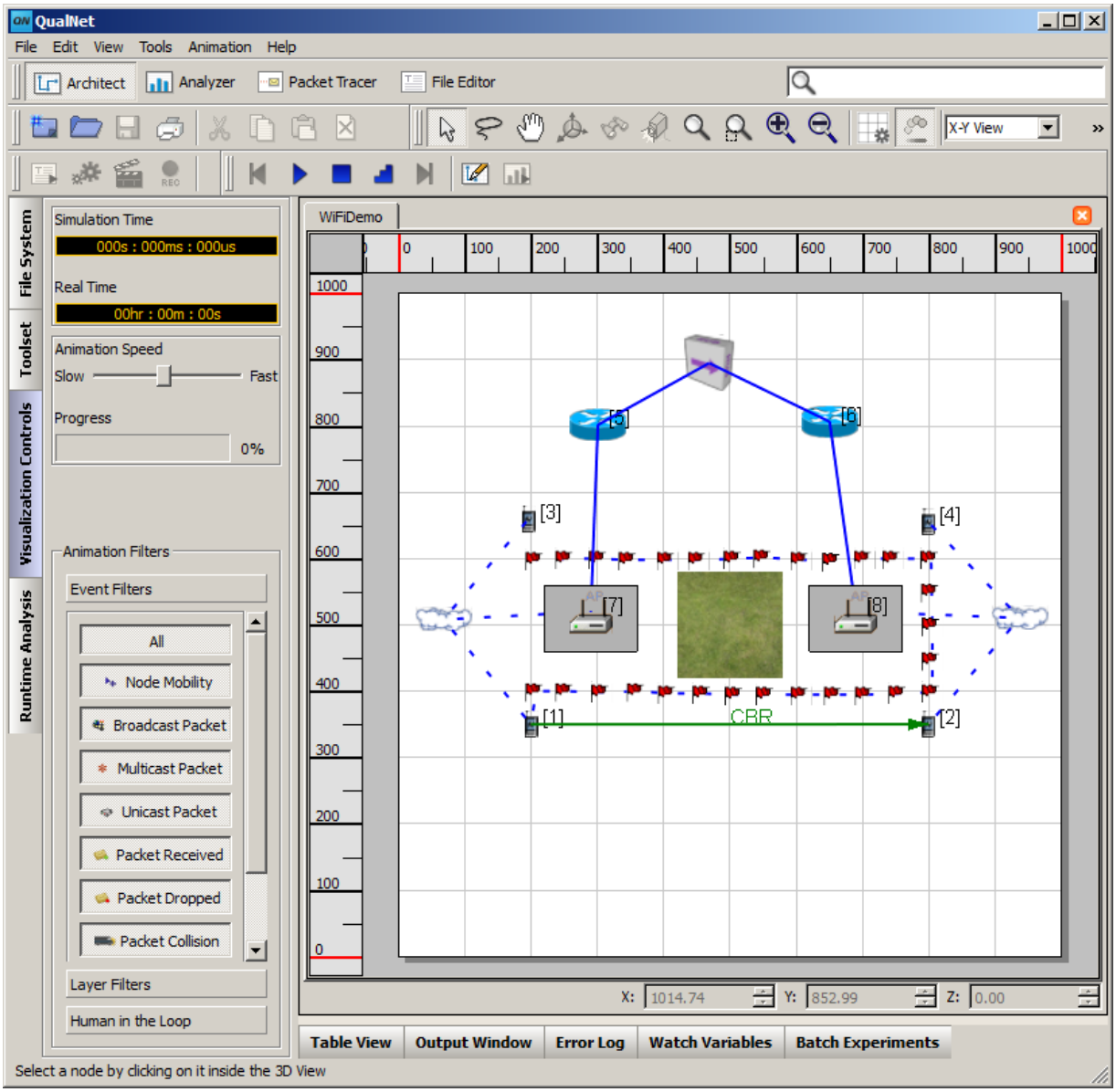


FIGURE 2-25. Visualize Mode of Architect

Some warning messages may be displayed in the **Error Log** panel below the canvas. You can close the **Error Log** panel by clicking on the **Error Log** button at the bottom of the display.

Setting Animation Speed and Filters

In the **Visualization Controls** panel, do the following:

1. Move the **Animation Speed** slider to the left to reduce the animation speed. (You can also change the animation speed while the scenario is running.)

FIGURE 2-26. Animation Speed Control

2. By default, all animation filters are on. For this example, we will turn most of these off.
   a. Under **Event Filters**, filters which enable/disable animation of events is displayed. Turn off all event filters except **Node Mobility**, **Broadcast Packet**, and **Packet Received**. To turn off a filter, click on the button.

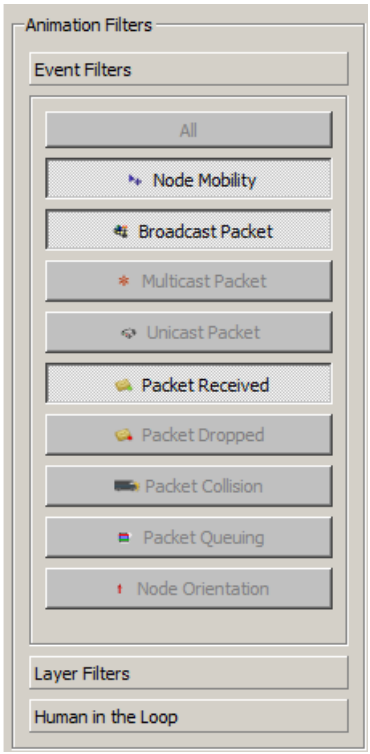


FIGURE 2-27. Event Filters

b. Click on **Layer Filters**. This displays the list of filters which enable/disable animation at different layers. Turn off all layer filters except **Radio** and **Network**.

FIGURE 2-28. Layer Filters

Running the Simulation

To start the simulation, click the **Play** button.

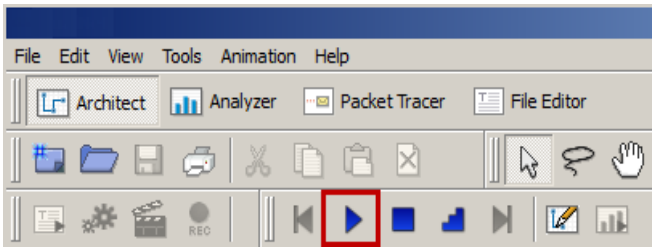


FIGURE 2-29. Play Button

The simulation starts running. While the simulation is running, you can adjust the animation speed in the **Visualization Controls** panel and pause and resume the simulation by means of the **Pause-Resume** button.

In the **Visualization Controls** panel, the simulation time and real time are displayed in the top two fields and the **Progress** bar displays how much of the simulation has completed. You can adjust the animation speed by using the slider.

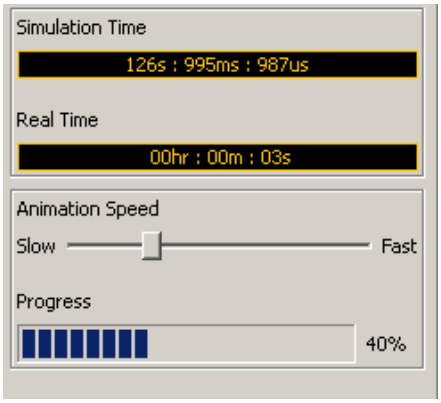


FIGURE 2-30. Visualization Control Panel

While the simulation is running, you will observe the following:

- Circles representing radio transmissions: When a node transmits a packet, an expanding circle starts from the node. When the circle reaches its final size, it covers the approximate area within which the node's transmission can be received.
- Green arrows representing successful packet reception: When a packet transmitted from one node is received by another node, a green arrow is drawn from the sender node to the receiver node.
- Mobile Station 1 (node 1) will move along the path marked by the waypoint markers (red flags).

During the simulation, node 1 sends packets to node 2. The green arrows show the path of packets from node 1 to node 2. In the beginning, node 1 is associated with node 7, and the packets follow the path: node 1 **>** node 7 **>** node 5 **>** node 6 **>** node 8 **>** node 2.

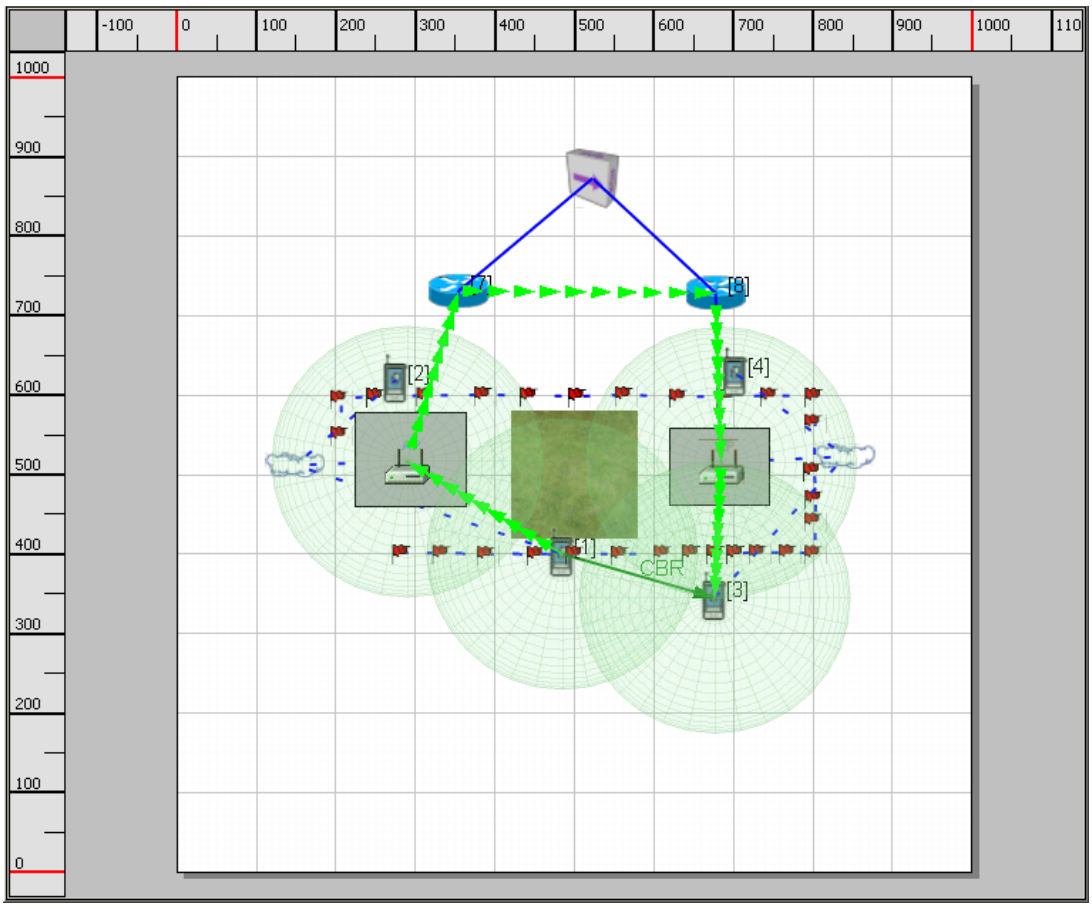


FIGURE 2-31. Scenario Animation

Node 1 gradually moves away from node 7 and closer to node 8. Around 95 seconds, a handover occurs and node 1 associates with node 8 instead of node 7. The packets now follow the path: node 1 **>** node 8 **>** node 2.

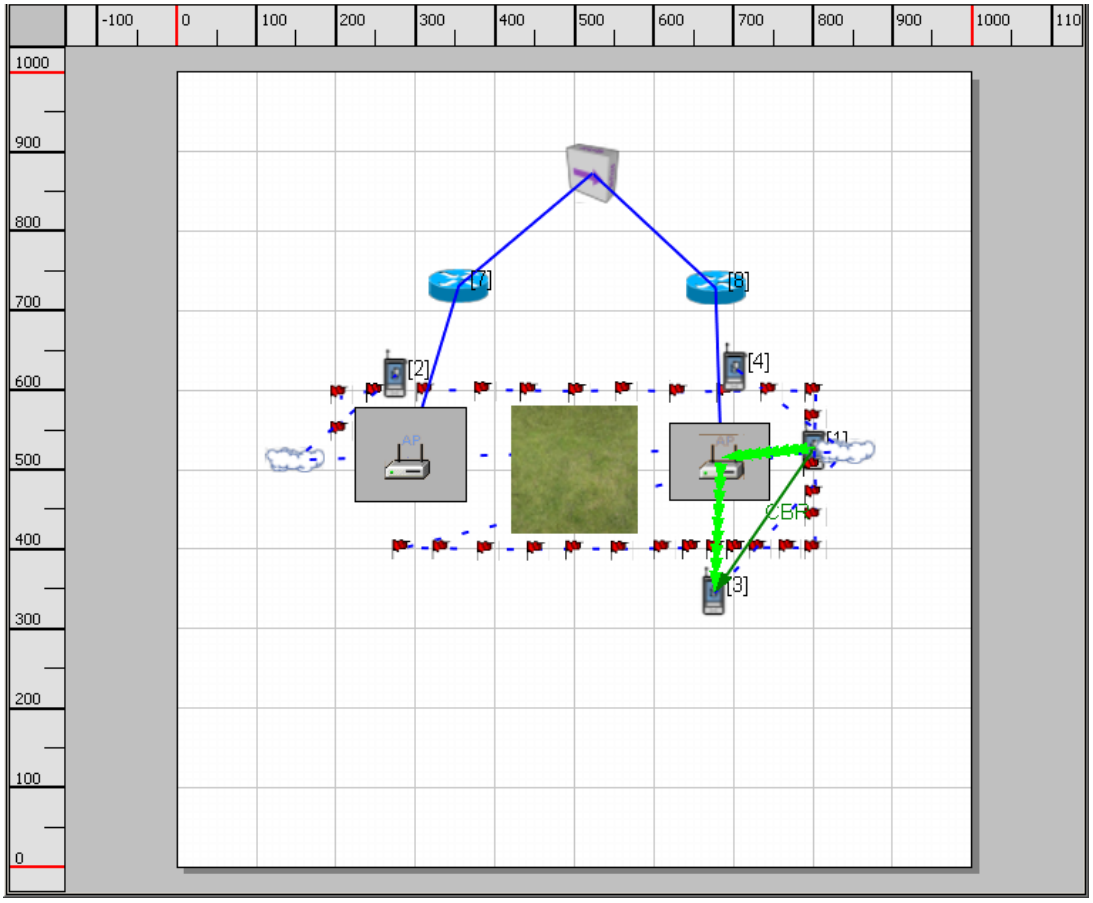


FIGURE 2-32. First Handover: Node 1 Associates with Node 8

As node 1 moves around the right building, it gradually moves away from node 8 and closer to node 7. Around 260 seconds, another handover occurs and node 1 associates with node 8 instead of node 7. The packets again follow the original path: node 1 **>** node 7 **>** node 5 **>** node 6 **>** node 8 **>** node 2.

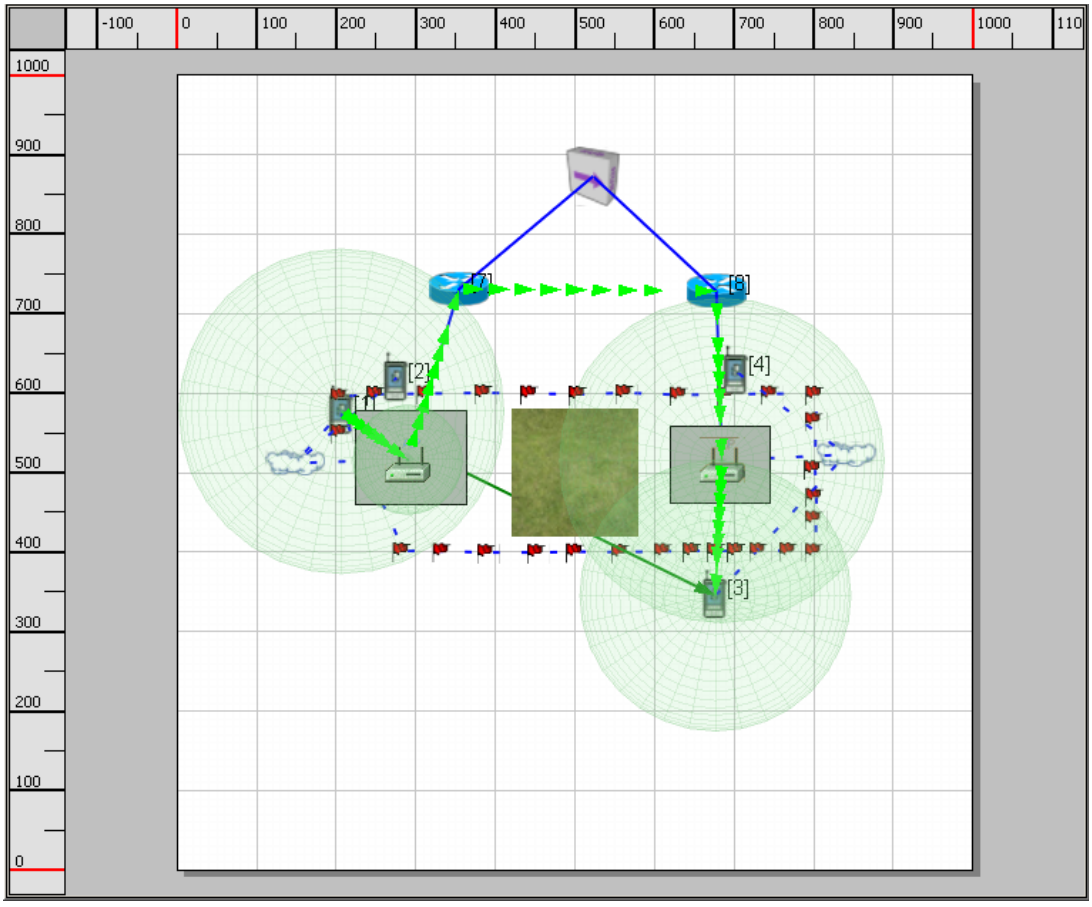


FIGURE 2-33. Second Handover: Node 1 Associates with Node 7

2.6 Analyzing Simulation Results

We will now plot some simulation results in QualNet Analyzer.

After the simulation has completed, switch to Analyzer by clicking the **Analyze Results** button shown below:

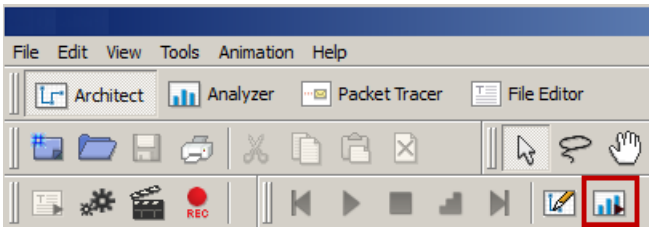


FIGURE 2-34. Analyze Results Button

The following screen is displayed:

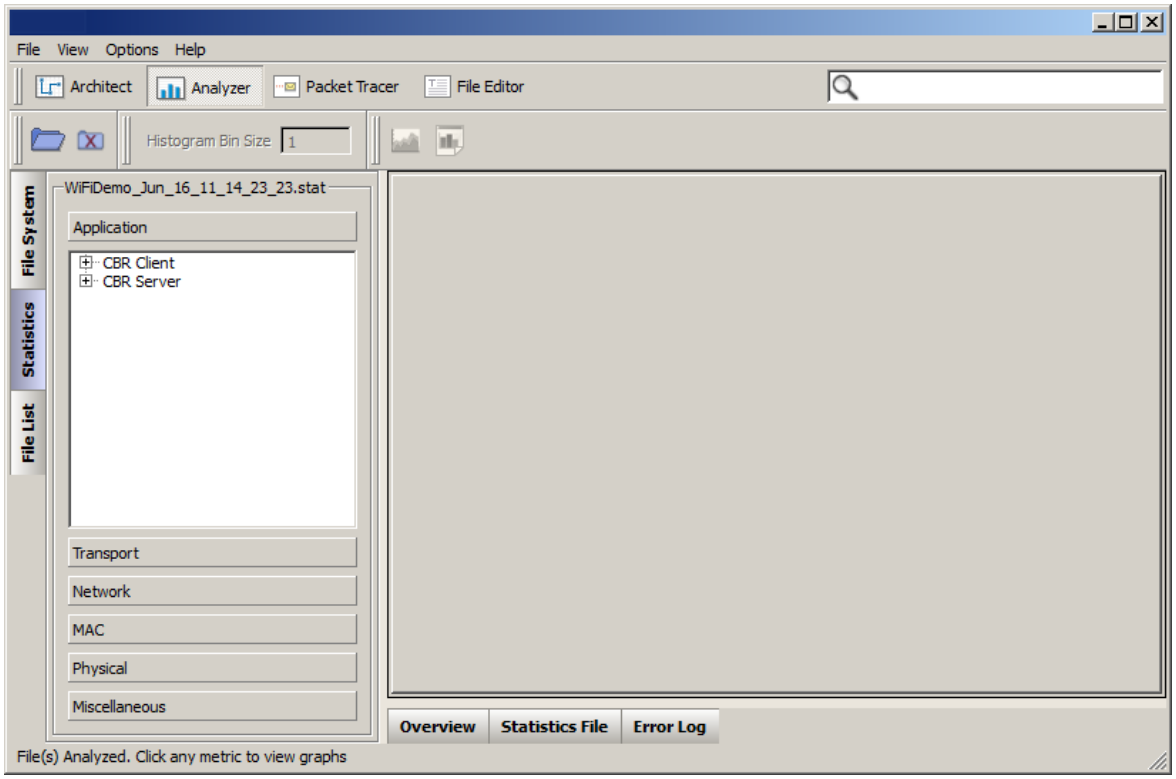


FIGURE 2-35. Analyzer: Initial Display

The left panel lists the protocols at each layer (Application, Transport, Network, MAC, and Physical). Click on the button for a layer to display the protocols at that layer. Click on the '**+**' sign next to a protocol's name to display the statistics collected for that protocol. By default, Application layer protocols for which statistics are available are displayed in the left panel (in this case, CBR Client and CBR Server).

1. Click on the '**+**' sign next to **CBR Client**. The list of statistics collected for the CBR client (packet sender) is displayed.

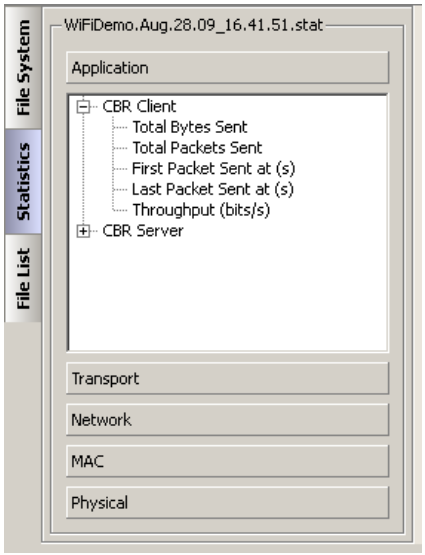


FIGURE 2-36. CBR Client Statistics

2. Click on **Total Packets Sent**. The chart for this statistic is drawn in the right panel.

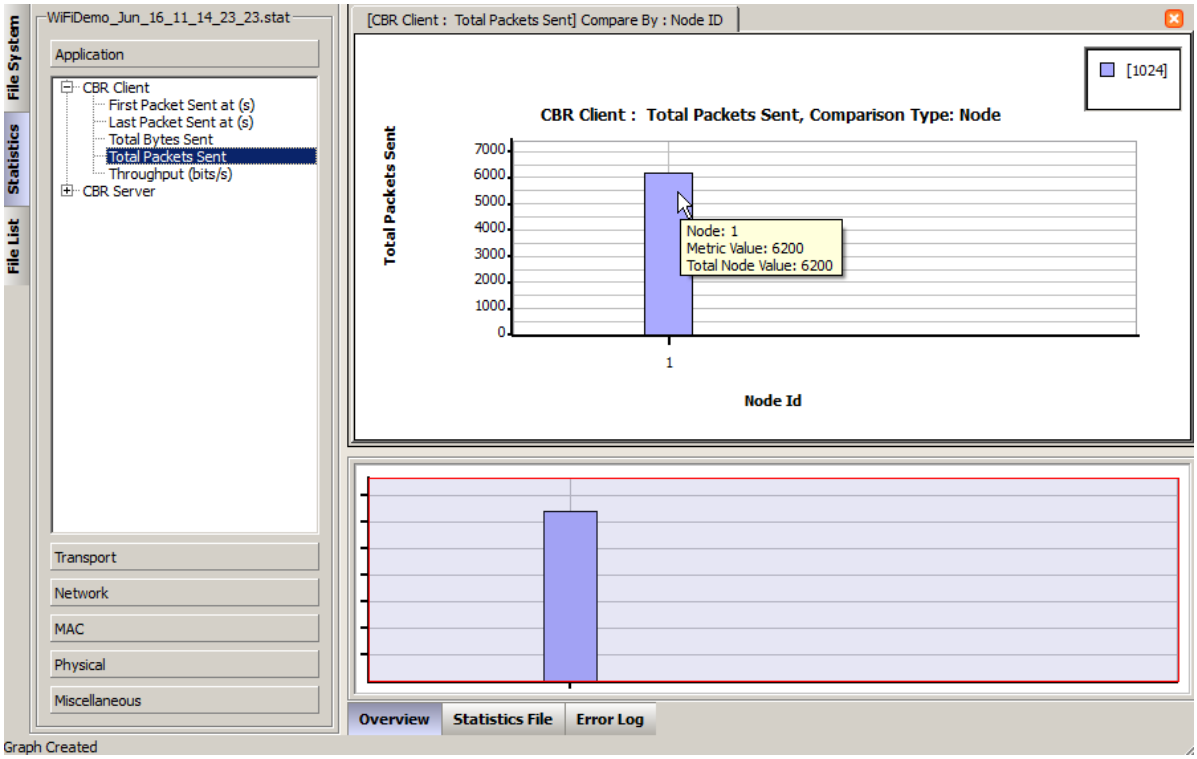


FIGURE 2-37. Total Packets Sent

Since there is only one CBR Client in this scenario, the plot shows a single bar. The top chart is the main display area of the statistic plot. (We will explain the bottom chart a little later.)

Placing the mouse over the bar displays the statistic value. In this case, it shows that 6200 packets were sent by node 1.

3. Click on the '**+**' sign next to **CBR Server**. The list of statistics collected for the CBR server (packet receiver) is displayed.
4. Click on **Total Packets Received**. The chart for this statistic is drawn in the right panel.

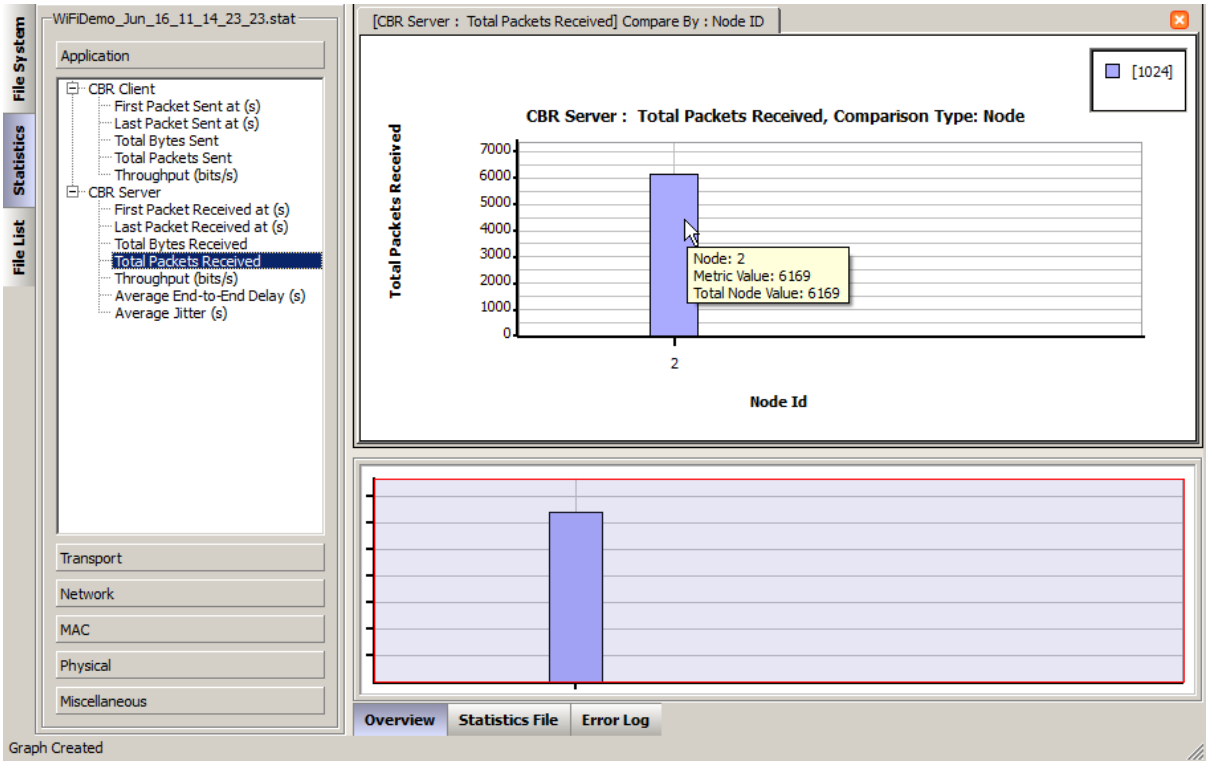


FIGURE 2-38. Total Packets Received

5. Click the button labeled **Network** in the left panel. Network layer models for which statistics are available are listed (in this case, IP, AODV for IPv4, and FIFO queue).

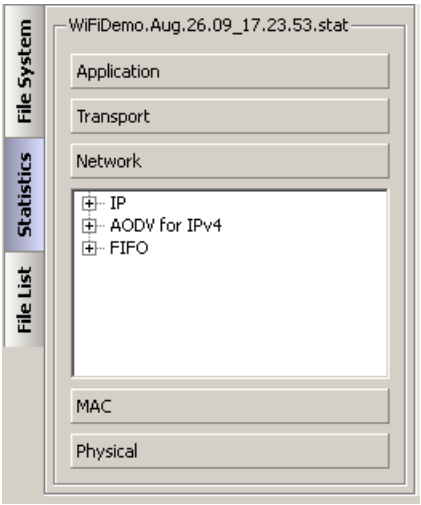


FIGURE 2-39. Network Layer Statistics

6. Click on the '**+**' sign next to **FIFO**. The list of statistics collected for the FIFO queue model is displayed.

7. Click on **Peak Queue Size**. The chart for this statistic is drawn in the right panel.

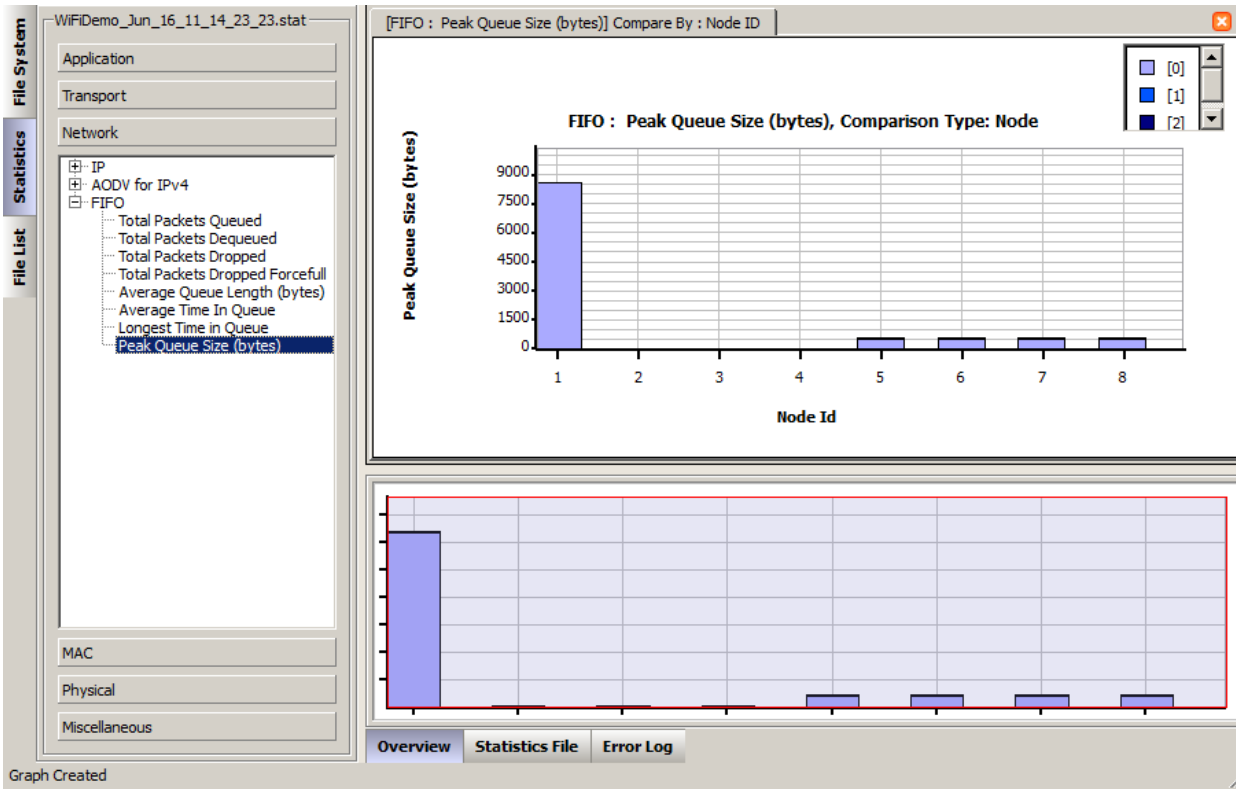


FIGURE 2-40. Peak Queue Size

There are two charts plotted in the right panel. The bottom chart is an overview chart: it shows the statistics for all nodes in the scenario. The top chart is a magnified view of the selected part of the overview chart. The selected part is outlined by a red rectangle. Initially, the entire overview chart is selected. Therefore, both charts in the right panel are identical.

8. Select a region of the overview chart to zoom into. To do this, in the overview chart, resize and move the red rectangle over the region you want to see in more detail. The following figure shows the charts when a region corresponding to nodes 5 and node 6 is selected.

   You can close the overview chart by clicking the **Overview** button at the bottom. This will increase the size of the main (top) chart.

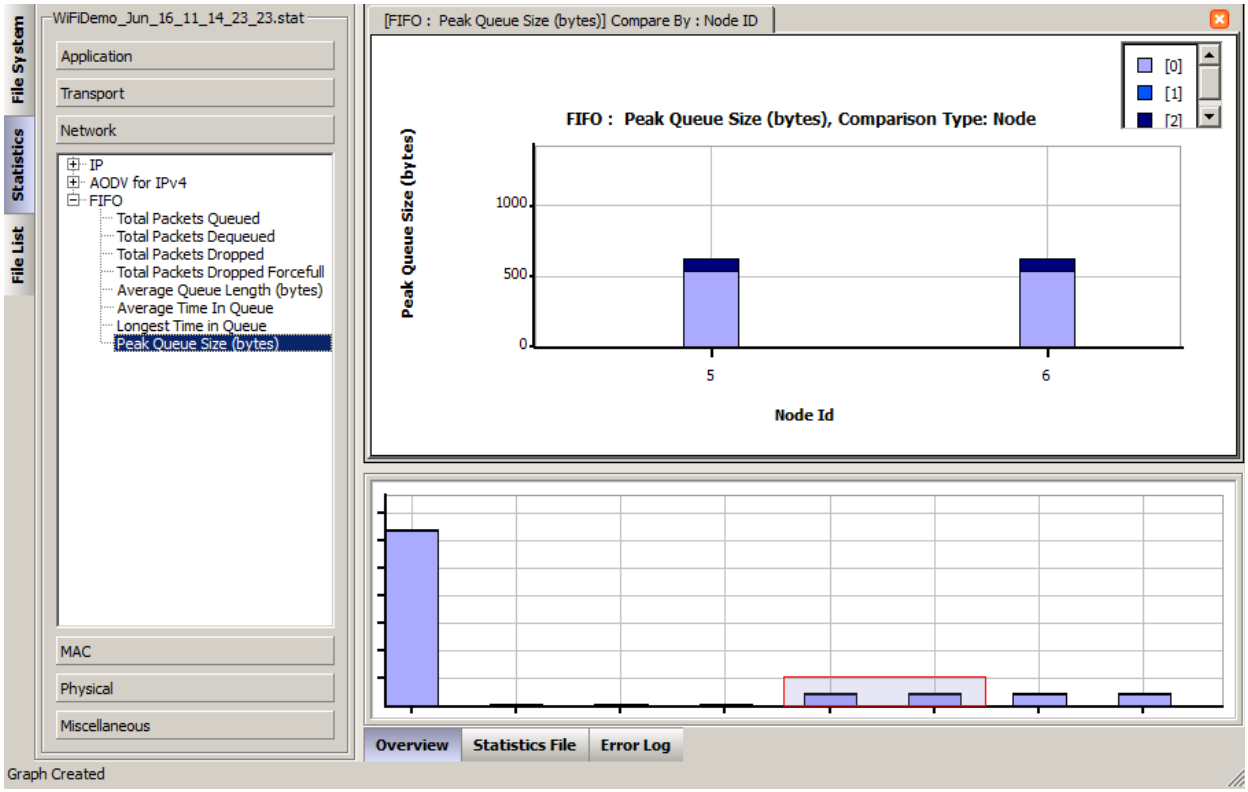


FIGURE 2-41. Peak Queue Size for Nodes 5 and 6

2.7 Running the Simulation Again

From Analyzer, you can switch to Architect to run the simulation again.

1. Click on the **Architect** button in the toolbar to return to Architect.

FIGURE 2-42. Switch to Architect

This takes you back to the Visualize mode of Architect.

2. Click the **Run Simulation** button to run the experiment again.

Note: You can modify the scenario in the Design mode. From the Visualize mode, you can switch to the Design mode of Architect by clicking the **Switch to Design Mode** button shown below.

FIGURE 2-43. Switch to Design Mode

3 Creating Simulation Scenarios

In this chapter, we describe how to create a new scenario. As an example, we will describe the steps to create a scenario that is the same as the WiFi demo scenario of Chapter 2.

3.1 Create a New Scenario

To create a new scenario, start the QualNet GUI and click the **New** button in the toolbar.

FIGURE 3-1. Standard Toolbar

This will open a new, blank scenario in the Canvas area. The name of the scenario is “untitled_1” which is displayed in the scenario tab.

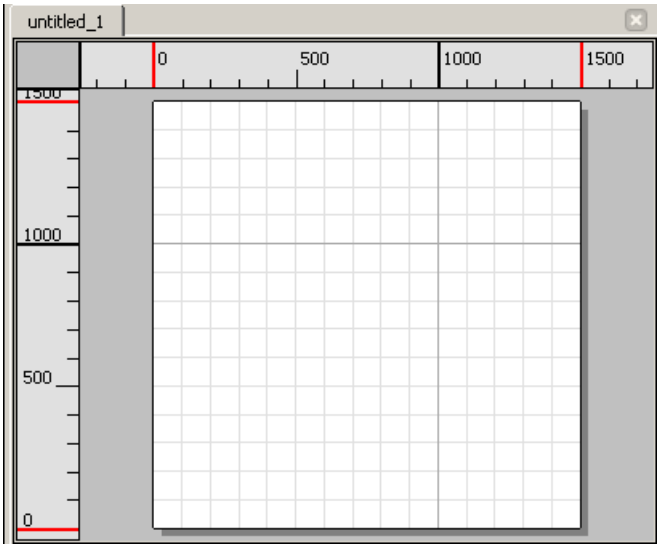


FIGURE 3-2. New Scenario

3.2 General Simulation Parameters

Next, we will set the name and simulation time for the new scenario.

Open the Scenario Properties Editor by clicking the **Scenario Properties** button in the Toolset panel.

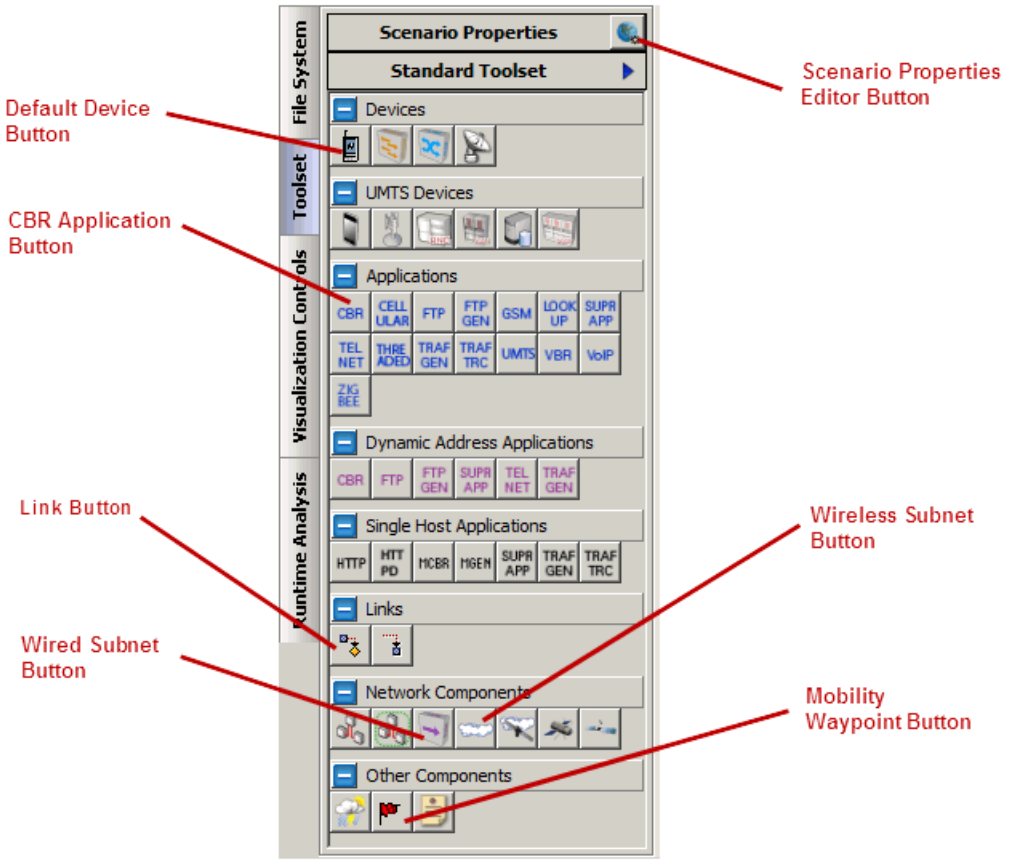


FIGURE 3-3. Toolset Panel

This opens the Scenario Properties Editor shown in Figure 3-4.

In the **General** tab of the Scenario Properties Editor:

- Set **Experiment Name** to *MyWiFi*.
- Set **Simulation Time** to *320 seconds*.
- Click **OK**.

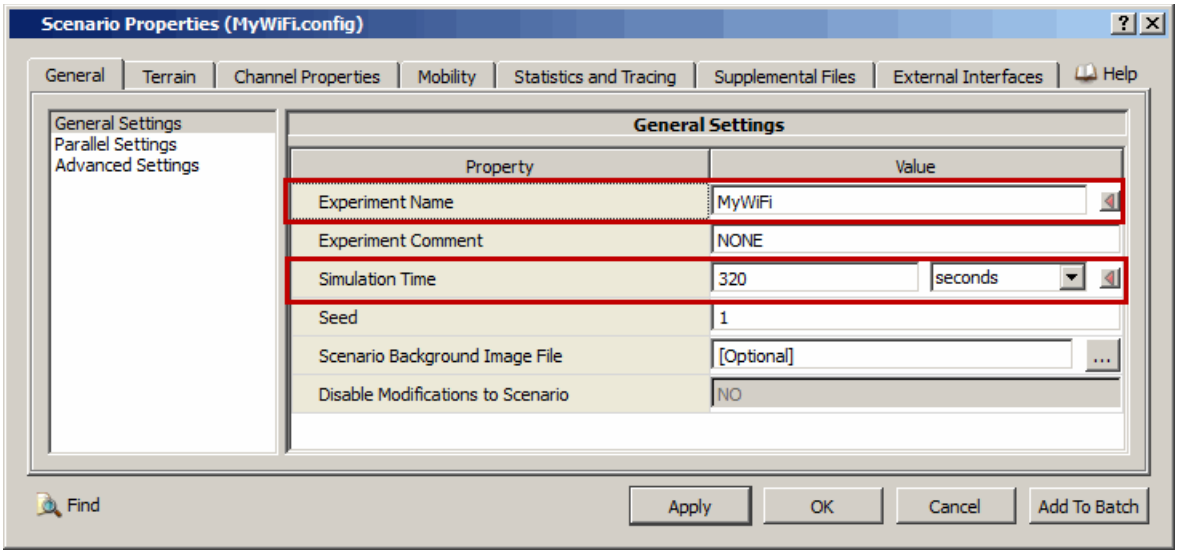


FIGURE 3-4. Setting General Parameters

3.3 Saving Scenario and Copying Terrain Files

Click the **Save** button in the toolbar (see Figure 3-1). In the dialog that opens, navigate to the directory QUALNET_HOME/scenarios/user (where QUALNET_HOME is the directory where QualNet is installed), and enter “MyWiFi” in the **File name** field. This will create a new folder, QUALNET_HOME/scenarios/user/MyWiFi. All files associated with the new scenario will be stored in this folder. The name displayed on the scenario tab also changes to “MyWiFi”.

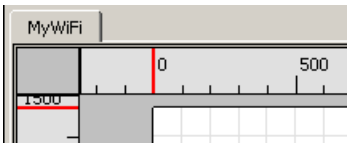


FIGURE 3-5. Saved Scenario

Copy the folder QUALNET_HOME/scenarios/demo/WiFiDemo/urban to QUALNET_HOME/scenarios/user/MyWiFi/urban. This folder contains the file with details of the urban terrain for the scenario (dimensions of buildings, park, etc.). Also copy the icon files for routers and access points, router-color.png and AccessPoint.png.

3.4 Terrain and Channel Parameters

Next, we will set the terrain properties and channel frequencies.

3.4.1 Terrain Parameters

Open the Scenario Properties Editor by clicking the **Scenario Properties** button in the Toolset panel (see Figure 3-3).

Click on the **Terrain** tab and set the following:

- Set each of the X and Y fields of **Scenario Dimensions** to *1000*.
- Set **Urban Terrain Format** to *QualNet Format* by selecting it from the pull-down list. Some additional parameters will be displayed.

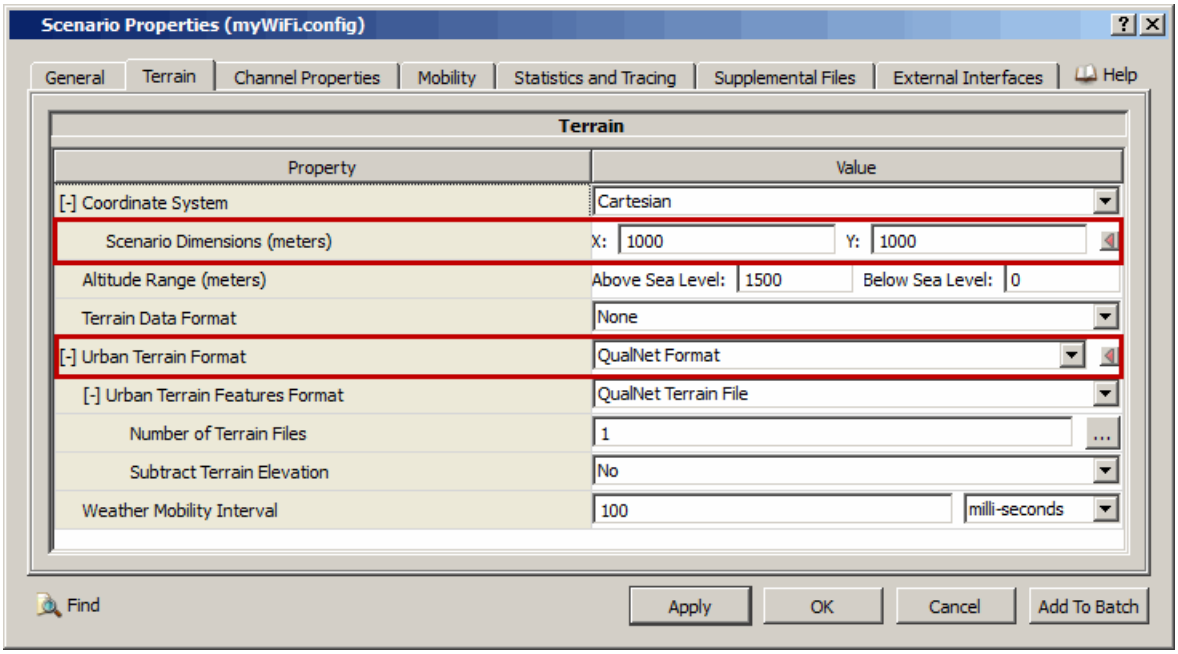


FIGURE 3-6. Terrain Properties

- Leave the parameters unchanged. Click the **Open Array Editor** ... button in the **Value** field of **Number of Terrain Files**. This opens the Array Editor for urban terrain features files.

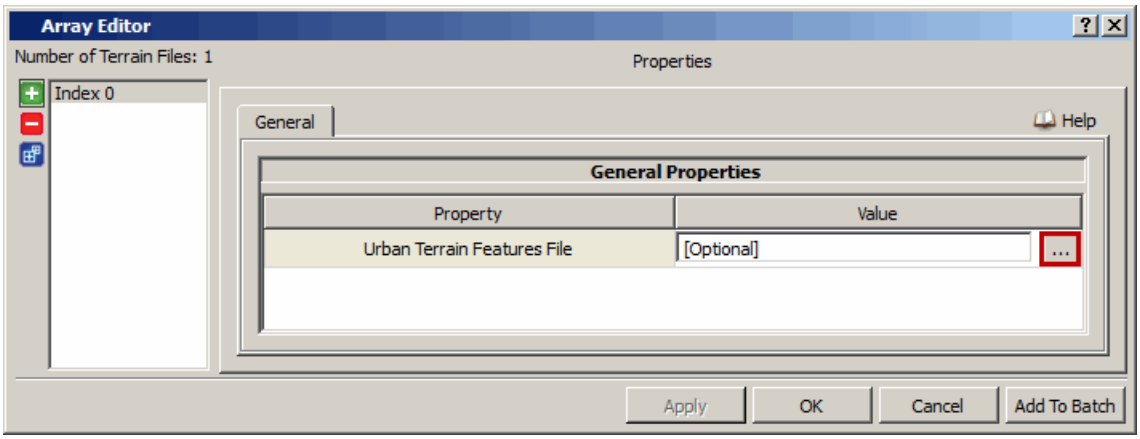


FIGURE 3-7. Array Editor for Urban Terrain Features Files

- Click the **Select File** ... button. This opens a file selector.

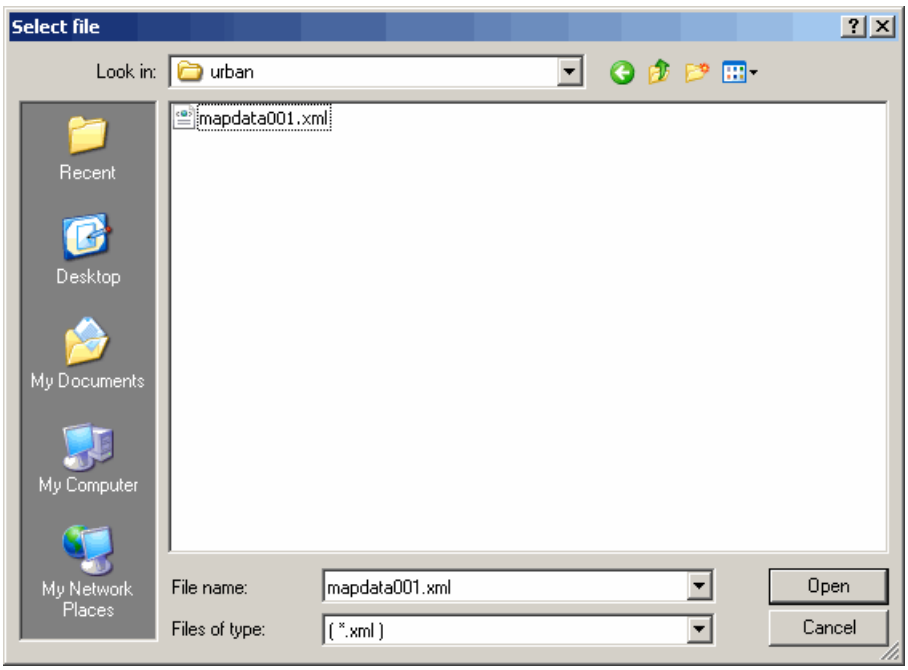


FIGURE 3-8. Select Urban Terrain File

- Navigate to the folder QUALNET_HOME/scenarios/user/MyWiFi/urban, select the file mapdata001.xml, and click **Open**.
- Close the Array Editor by clicking **OK**.

3.4.2 Channel Properties

Click on the **Channel Properties** tab and set the following:

- Set **Number of Channels** to *2*.

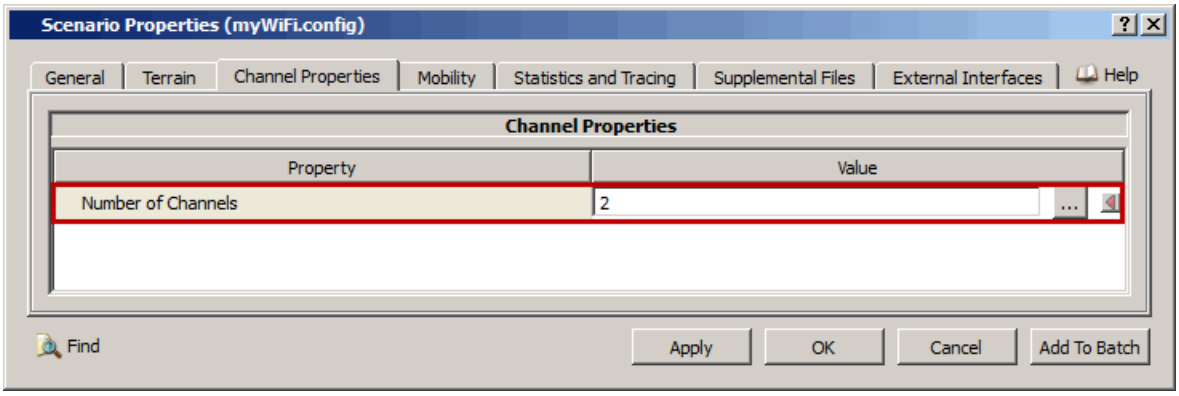


FIGURE 3-9. Number of Channels

- Click the **Open Array Editor** ... button in the **Value** field of **Number of Channels**. This opens the Array Editor for configuring channel properties shown in Figure 3-10.
- Leave the parameters for the first channel (corresponding to Index 0) unchanged. For configuring the second channel, select **Index 1** in the left panel and, in the right panel, set **Channel Frequency** to *2.5 GHz*. Leave all other parameters unchanged.

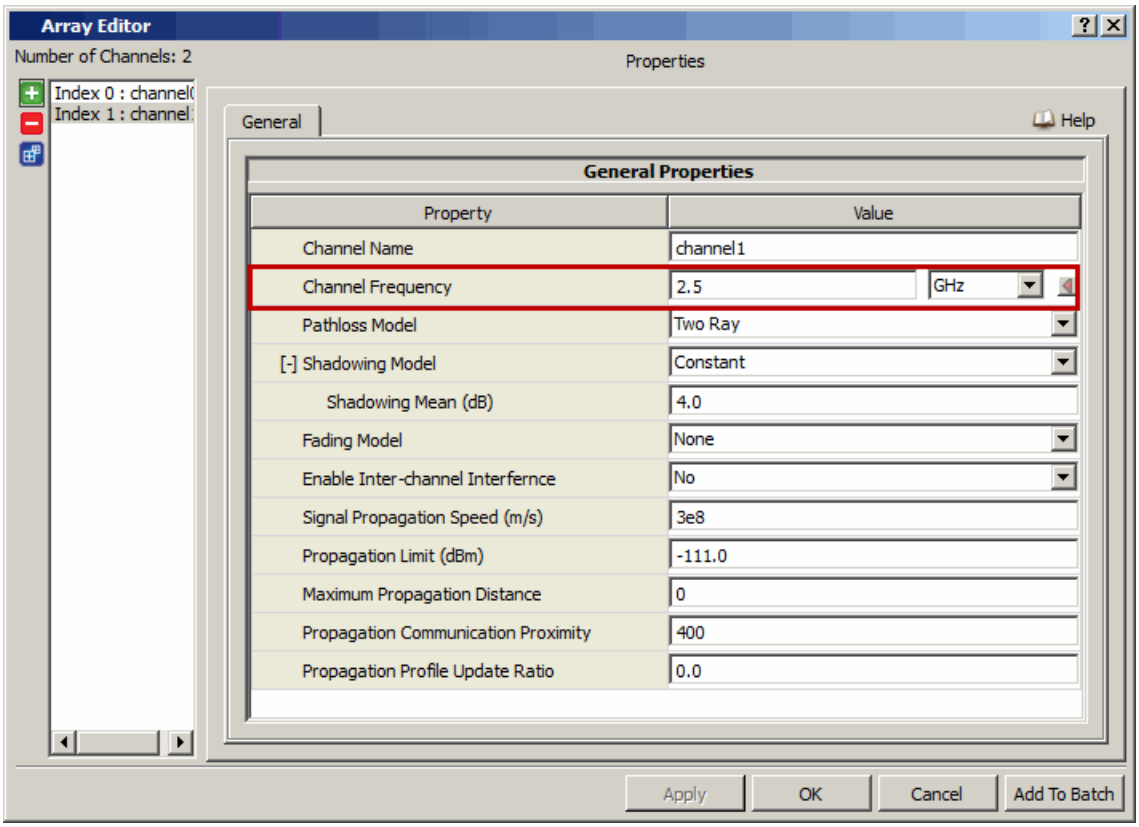


FIGURE 3-10. Channel Frequency

- Close the Array Editor by clicking **OK**.
- Close the Scenario Properties Editor by clicking **OK**.

The canvas will appear as shown in Figure 3-11. This is the X-Y view (view as seen from the top) of the scenario. The grey rectangles represent the buildings and the green rectangle represents the park.

Note: If the buildings and park appear at the bottom of the canvas instead of the center, save the scenario, close it, and reopen it to get the correct display.

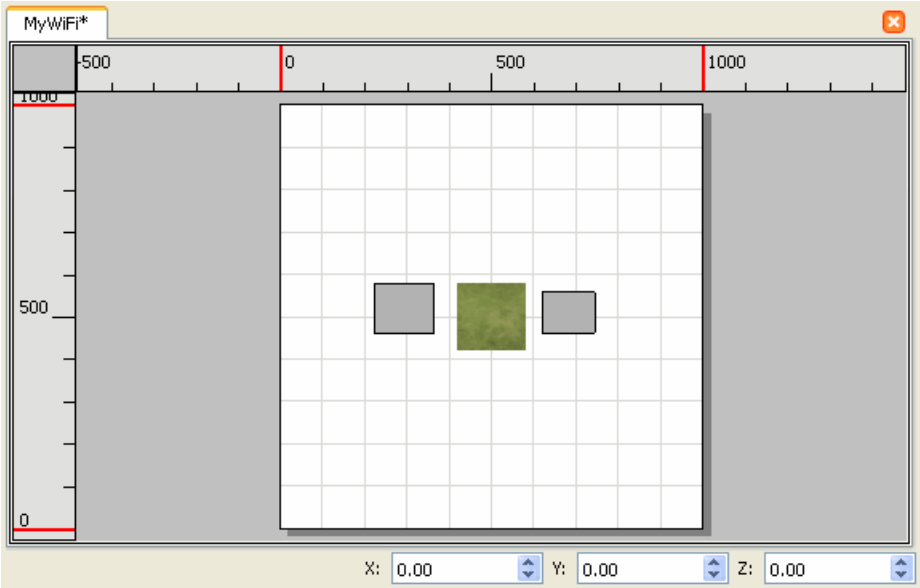


FIGURE 3-11. X-Y View of Scenario

3.5 Creating Network Topology

We will now place the nodes and subnets on the canvas. We will place the mobile stations, routers, wireless subnets and wired subnet in the X-Y view. Since the access points are placed on the top of the buildings, we will place those nodes in the 3D view. All nodes (mobile stations, access points, and routers) are of the type *Default Device*. We will change the icons used to represent the routers and access points later.

Placing Devices

To place Default Device nodes on the canvas, do the following:

1. In the Toolset panel, sliced the Default Device by clicking on the first icon in the **Devices** toolbar (see Figure 3-3).
2. On the canvas, click on the location where the first node is to be placed. A Default Device icon will be displayed at that location.
3. Place additional Default Devices by clicking on the desired locations on the canvas.
4. Exit from insert mode and enter select mode by pressing the **Esc** key or the **s** key, or by clicking the **Select** button on the toolbar.

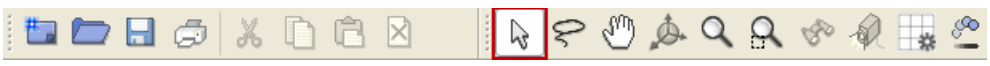


FIGURE 3-12. Select Button in Toolbar

Placing Wired and Wireless Subnets

Subnets are placed on the canvas by selecting the appropriate button in the **Network Components** toolbar (see Figure 3-3) and following the same procedure as for placing Default Devices.

Place six default devices, two wireless subnets, and one wired subnet on the canvas, as shown in Figure 3-13.

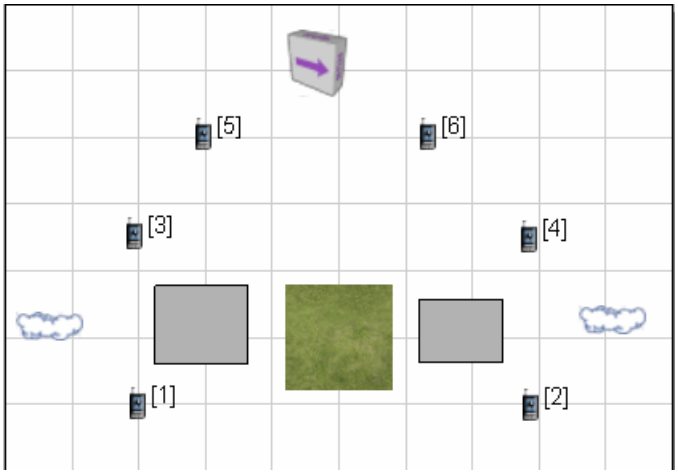


FIGURE 3-13. Placing Nodes and Subnets

Note: If a node is already selected when a wired or wireless subnet is placed on the canvas, a link is automatically created between the subnet and the selected node. To delete any unwanted links, click **Select** button to enter the Select mode, select the link by clicking on it, and click the **Delete** button on the toolbar or press the **Delete** key.

Placing Access Points

To place the two access points, do the following:

1. Change the scenario view from X-Y view to 3D view by selecting 3D from the pull-down list in the toolbar.

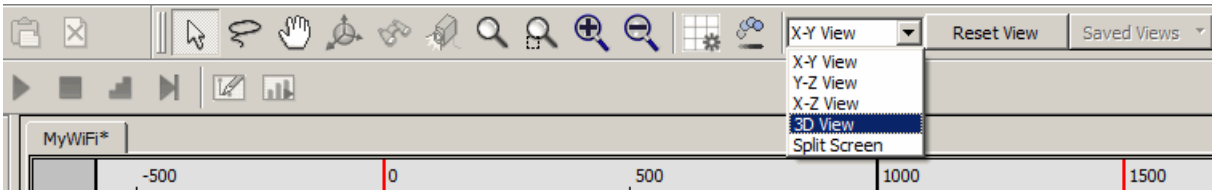


FIGURE 3-14. Changing to 3D View

The scenario view changes to the following:

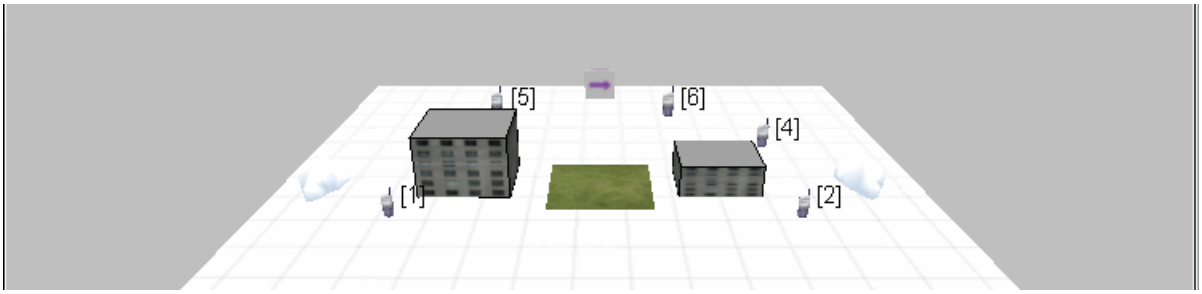


FIGURE 3-15. Scenario in 3D View

2. Place a default device on the top of each of the two buildings, near the center of the roof.

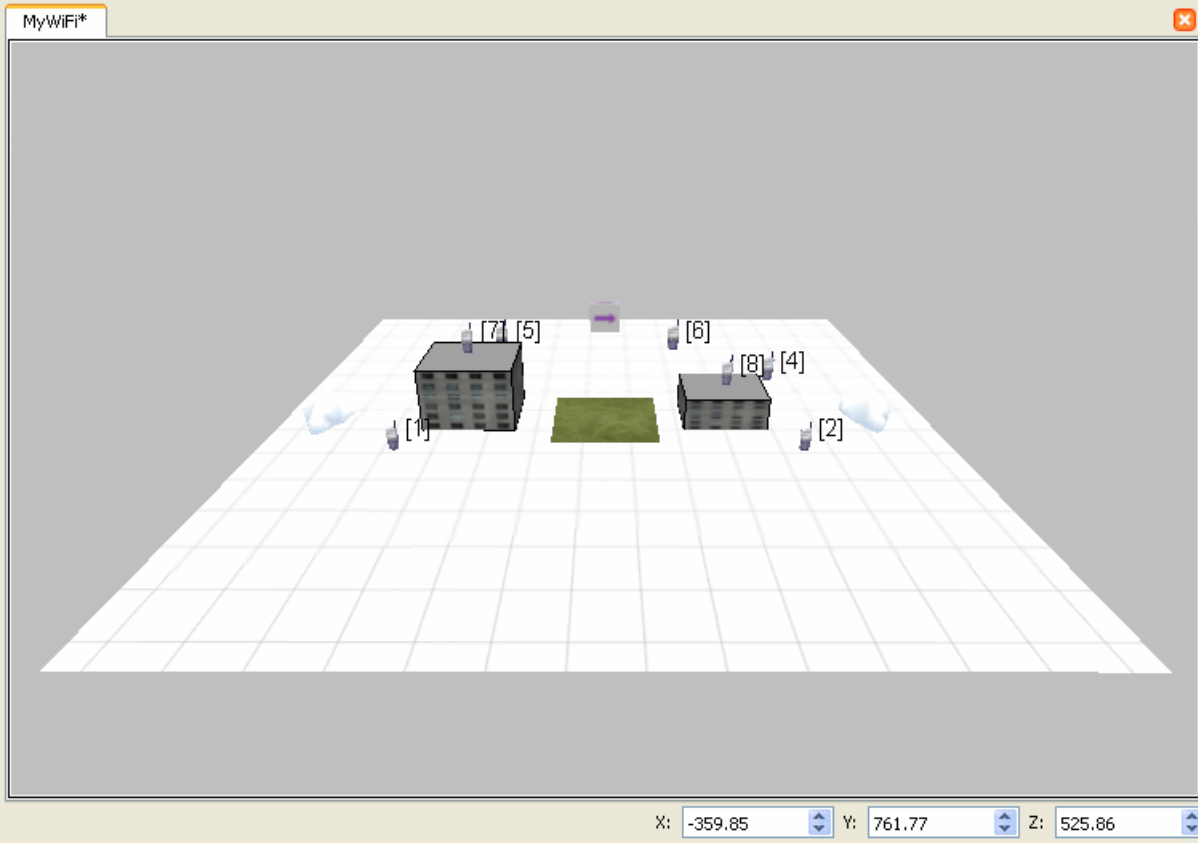


FIGURE 3-16. Placing Access Points on Rooftops

Note: Nodes 7 and 8 will act as access points and their position is critical. They should be placed right at the top of the building at the center of the roof. To ensure that each of these is properly placed, select the node and enter the desired coordinates in the Position Indicators just below the canvas. The coordinates of node 7 (left access point) should be (290, 520, 100) and the coordinates of node 8 (right access point) should be (690, 520, 50).

Creating Links

To create a link between two objects, click on the **Link** button in the **Links** toolbar (see Figure 3-3), click on the first object, drag to the second object, and release.

Switch the scenario view to X-Y view and create the following links:

- Connect nodes 5 and 7.
- Connect nodes 6 and 8.
- Connect nodes 5 and 6 to the wired subnet.

- Connect nodes 1, 3, and 7 to the left wireless subnet.
- Connect nodes 2, 4, and 8 to the right wireless subnet.

The scenario will appear as follows:

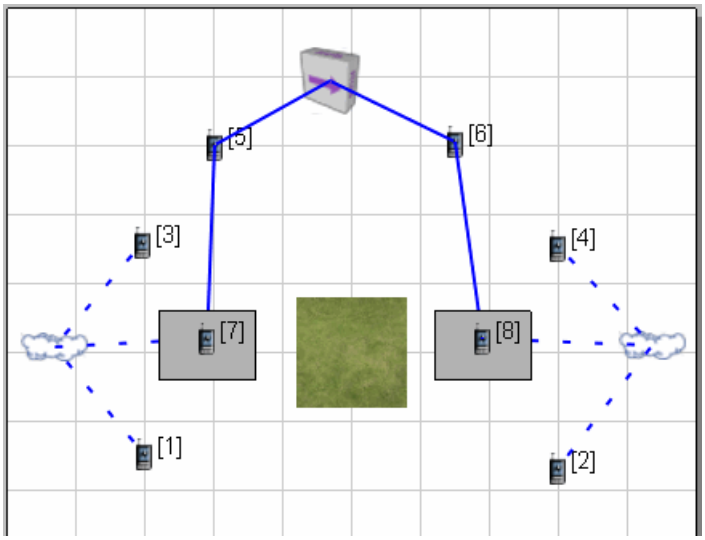


FIGURE 3-17. Connecting Nodes and Subnets

3.6 Specifying Mobility Pattern for Mobile Station 1

The mobility pattern for mobile station 1 (node 1) is specified by setting waypoints. To set a waypoint, a location and a time need to be specified: the mobile station will be at the specified location at the specified time. From one waypoint to the next, the mobile station moves in a straight line at a constant speed that is determined by the two waypoint locations and times.

To set waypoints for mobile station 1, perform the following steps:

1. Select the **Waypoint** button in the **Other Components** toolbar (see Figure 3-3).
2. Select mobile station 1 by left-clicking on it.
3. Next, left-click on the canvas at the desired location for the first waypoint. A waypoint marker is placed at the waypoint location and a line is drawn between mobile station 1 and the waypoint marker.
4. Click on the canvas at the location of the next waypoint. A waypoint marker is placed at that location and it is connected by a line to the previous waypoint. Similarly, place subsequent waypoints on the canvas. Add waypoints roughly at the same distance from each other, as shown in Figure 3-18.
5. After adding the last waypoint, click the right mouse button.

Note: A waypoint can be deleted or moved by selecting the waypoint marker and deleting or moving it, just like any other object on the canvas. Any waypoint can be moved or deleted.

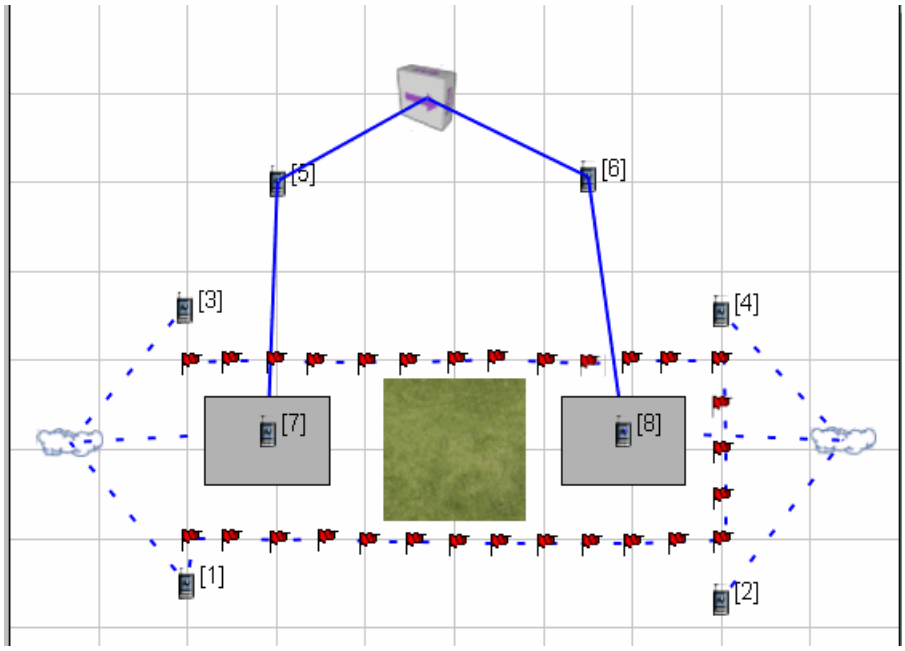


FIGURE 3-18. Adding Waypoints

6. To specify the waypoint times, open the **Mobility Waypoint Editor** by right-clicking on any waypoint marker and selecting **Properties**. Enter the waypoint times in 10 second increments, starting with 0 seconds.

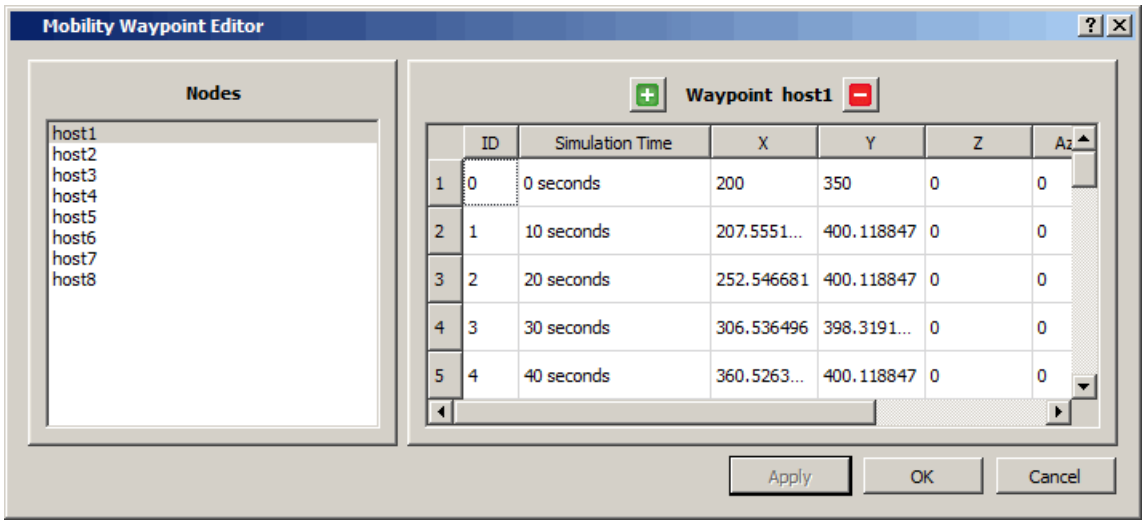


FIGURE 3-19. Mobility Waypoint Editor

Click **OK** to close the Mobility Waypoint Editor.

3.7 Creating an Application Session

The scenario has one Constant Bit Rate (CBR) application session between nodes 1 and 2. To specify a CBR session, do the following:

1. Click on the **CBR** button in the **Applications** toolbar (see Figure 3-3).
2. Click on node 1, drag the mouse to node 2, and release. A green arrow labelled CBR from node 1 to node 2 will be drawn on the canvas.

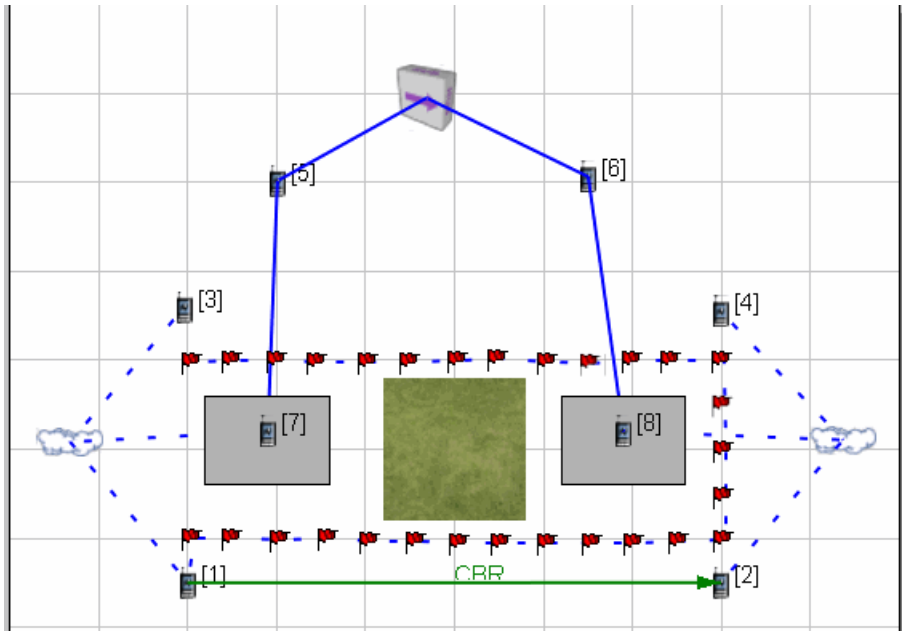


FIGURE 3-20. Adding a CBR Session

3.8 Setting Parameters

We will now set the configuration parameters for the mobile stations, routers, access points, wireless subnets, and the application session.

3.8.1 Parameters for Nodes 1 to 4 (Mobile Stations)

For each of the nodes 1 to 4, set the properties as follows:

1. Open the Default Device Properties Editor by doing one of the following:
   - Go to Select mode by clicking the **s** key and double click on the node on the canvas

   or
   - Open the **Table View** panel at the bottom, go to the **Nodes** tab and double-click the row for the node.

2. Go to the **Node Configuration** tab. In the left panel, click on **Routing Protocol**. In the right panel. set **Routing Protocol IPv4** to *AODV* (by selecting *AODV* from the pull-down list) and click **Apply**.

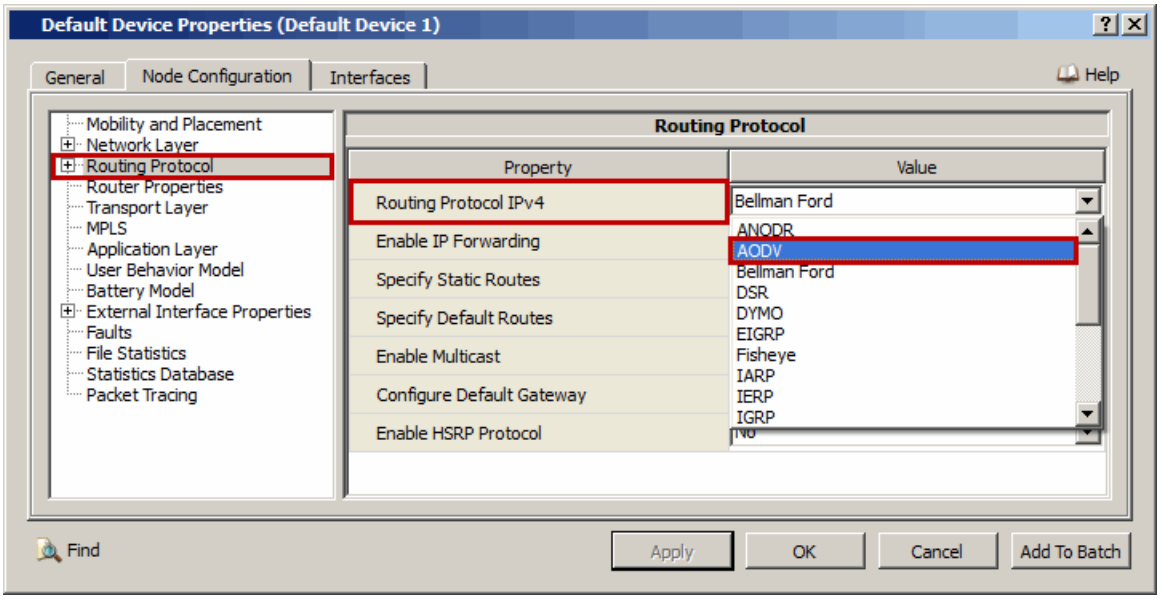


FIGURE 3-21. Setting Routing Protocol to AODV

3. Go to the **Interfaces** tab. In the left panel, expand the list of parameter groups by clicking on the '**+**' next to **Interface 0**. Click on **MAC Layer**.

4. In the right panel, set **Station Association Type** to *Dynamic*. (This will change the list of parameters that appear below it.)

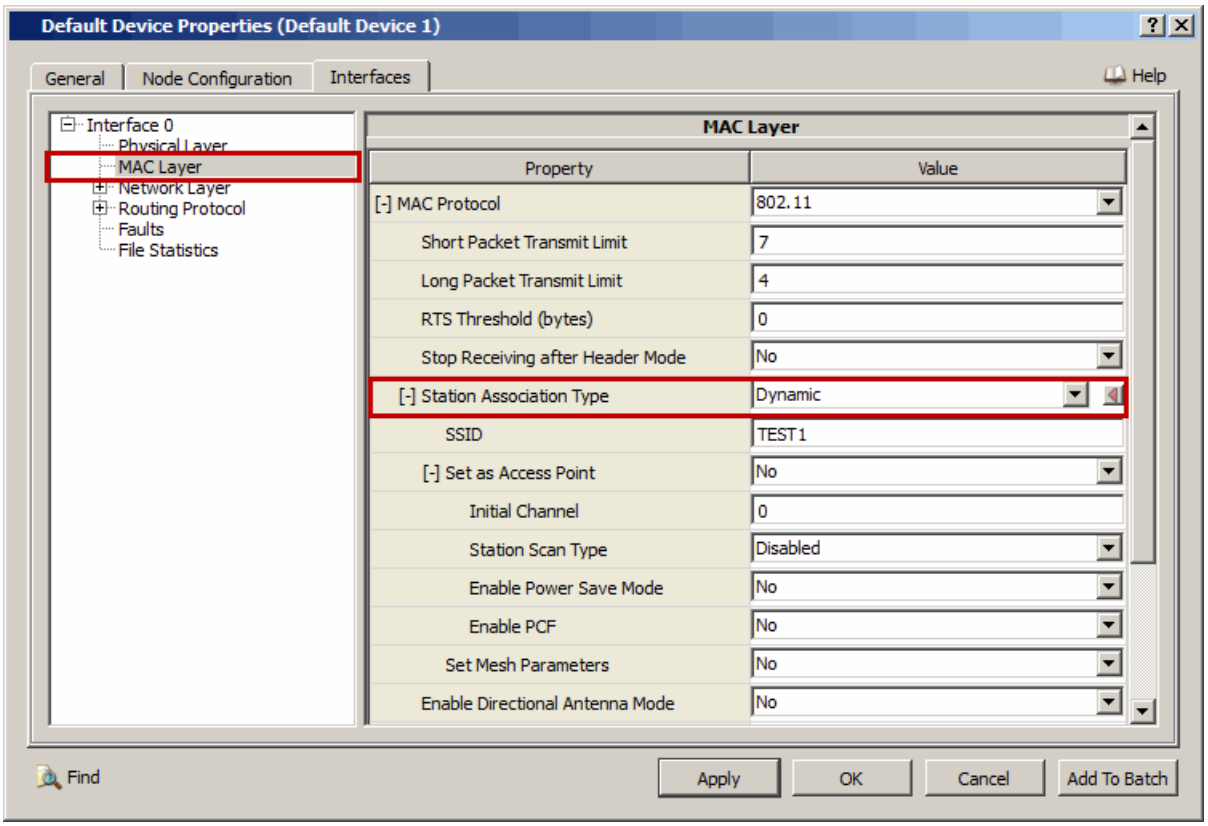


FIGURE 3-22. Setting Station Association Type to Dynamic

5. Set **Station Scan Type** to *Passive* and then set **Configure Handover RSS Trigger** to *Yes*.

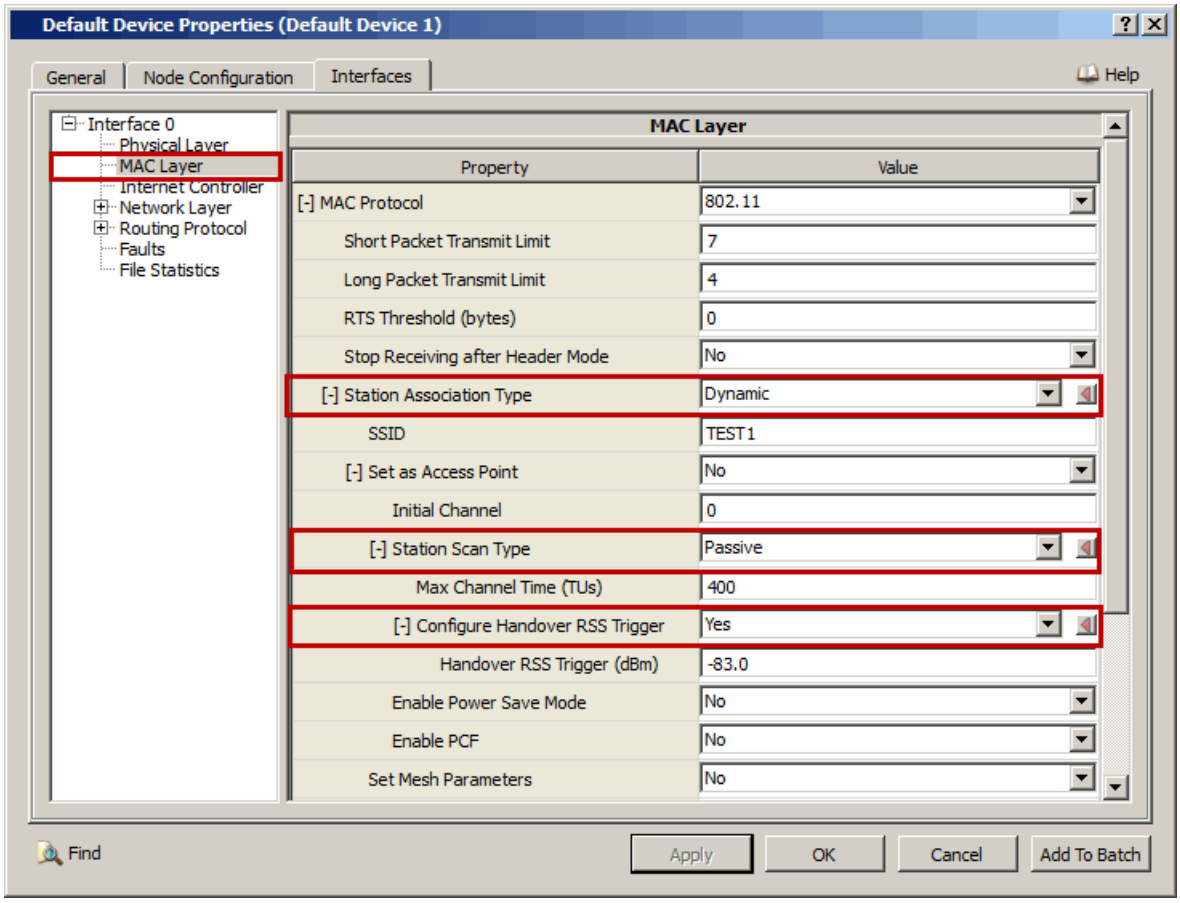


FIGURE 3-23. MAC Parameters for Nodes 1 to 4

6. Click **OK** to apply the changes and close the Properties Editor.

3.8.2 Parameters for Nodes 5 and 6 (Routers)

For each of the nodes 5 and 6, set the properties as follows:

1. Open the Properties Editor for the node (as for node 1).
2. In the General tab, click the **Select File** button in the **Value** column for **2D Icon**.

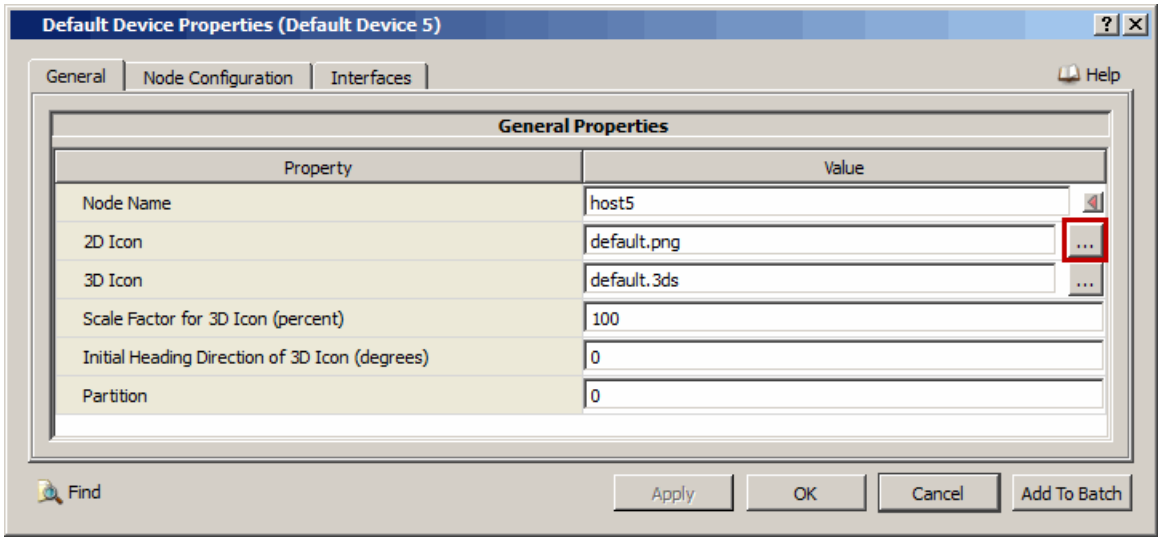


FIGURE 3-24. Setting Icon

3. This will open a file selector. Navigate to the folder where you have saved the scenario and select router-color.png. (This assumes that you have copied this file from the WifiDemo folder as stated in Section 3.1.)
4. Go to the **Node Configuration** tab and set **Routing Protocol IPv4** to *AODV* (as described above for node 1).
5. Click **OK** to apply the changes and close the Properties Editor.

3.8.3 Parameters for Nodes 7 and 8 (Access Points)

For each of nodes 7 and 8, set the parameters as follows:

1. Open the Properties Editor for the node (as for node 1).
2. Go to the **Node Configuration** tab and set **Routing Protocol IPv4** to *AODV* (as described above for node 1).
3. Go to the **Interfaces** tab. Two interfaces are listed in the left panel: one is the wireless interface and the other is the wired interfaces. To identify which is the wireless interface, expand the **Interface** group by clicking on the **+** sign and click on **MAC Layer**. In the right panel, **MAC Protocol** is set to *802.11* for the wireless interface and to *Abstract Link MAC* for the wired interface.

4. For the wireless interface, set **Station Association Type** to *Dynamic* (as for node 1) in the right panel.
5. Change **Set as Access Point** to *Yes*.
6. For node 8 (but *not* for node 7), set **Operating Channel** to *1*.

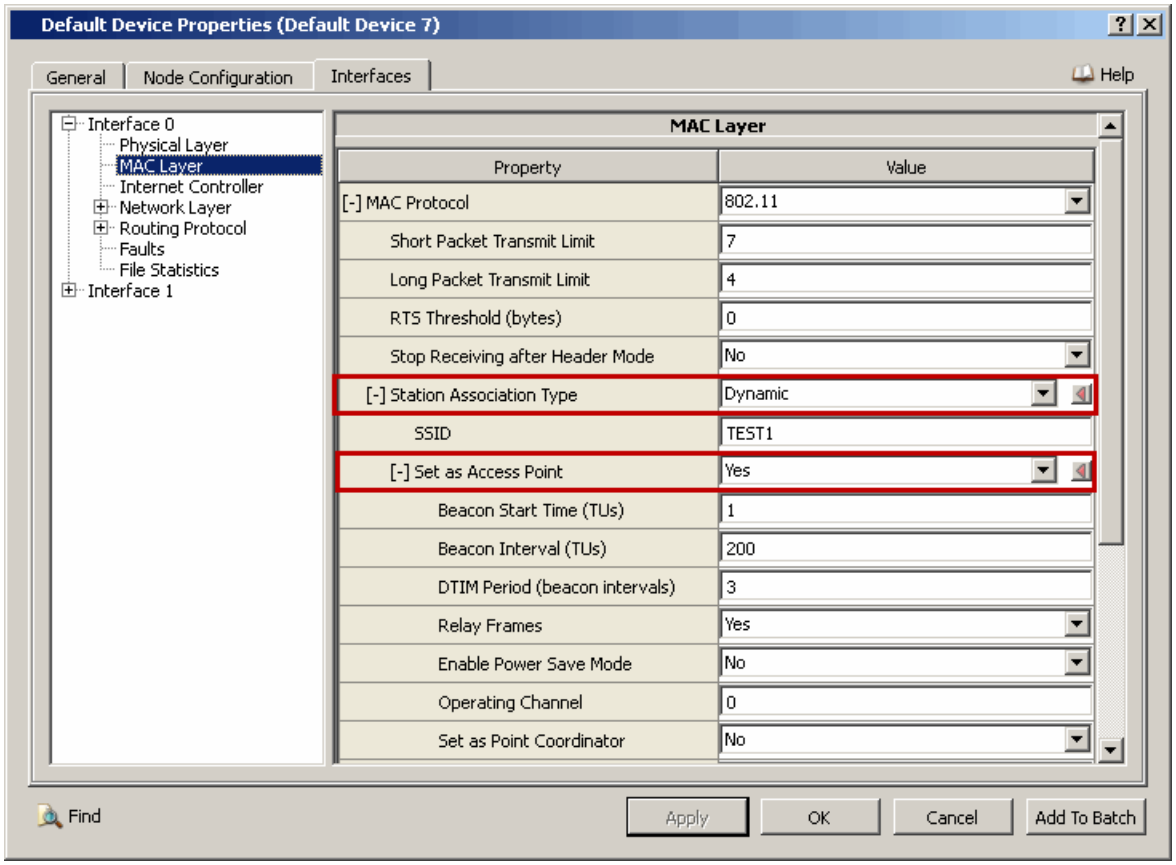


FIGURE 3-25. MAC Parameters for Node 7

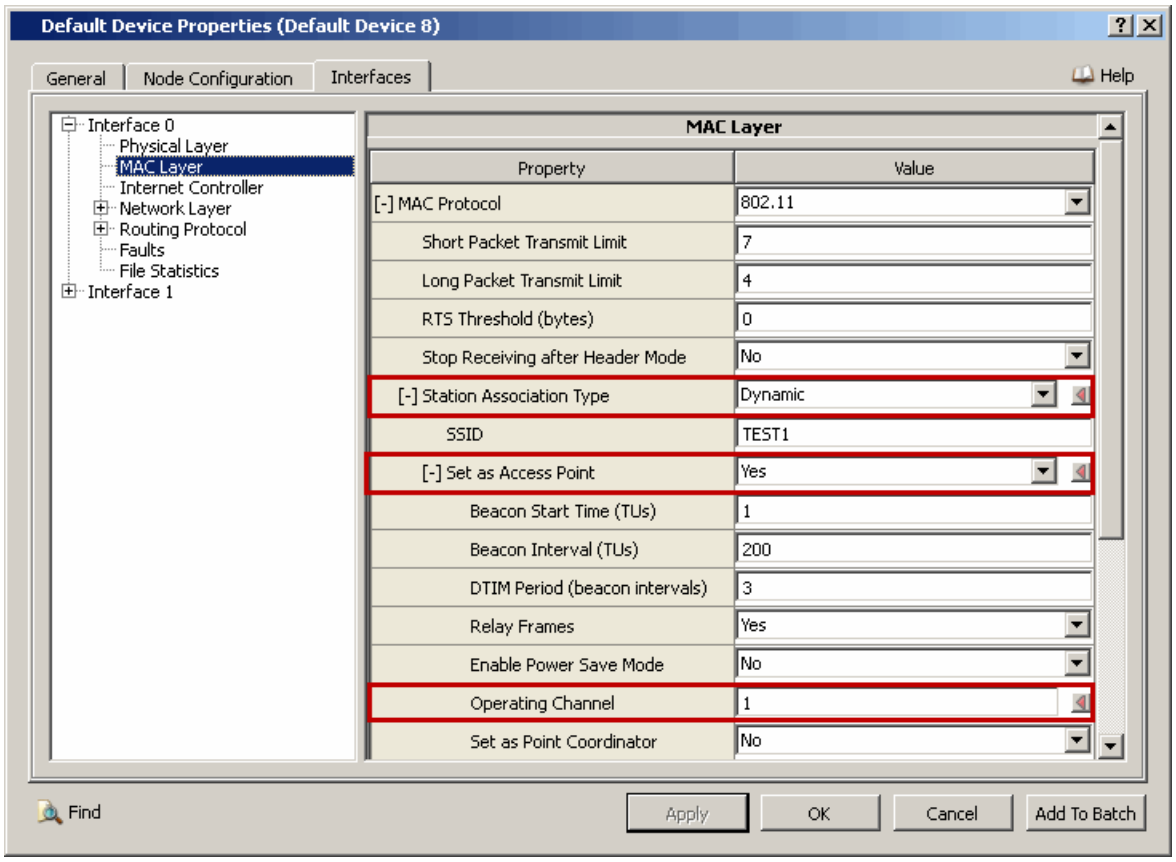


FIGURE 3-26. MAC Parameters for Node 8

7. Go to the **General** tab and set **2D Icon** to *AccessPoint.png*, as described for node 5. (This assumes that you have copied this file from the WifiDemo folder as stated in Section 3.1.)
8. Click **OK** to apply the changes and close the Properties Editor.

3.8.4 Parameters for the Wireless Subnets

For each of the two wireless subnets, set the properties as follows:

1. Go to Select mode by pressing the **s** key.
2. Double-click on the cloud icon to open the Wireless Subnet Properties Editor.
3. Go to the **Physical Layer** tab.

4. Click the **Open Channel List Editor** ... button in the **Value** field of **Listenable Channels**.

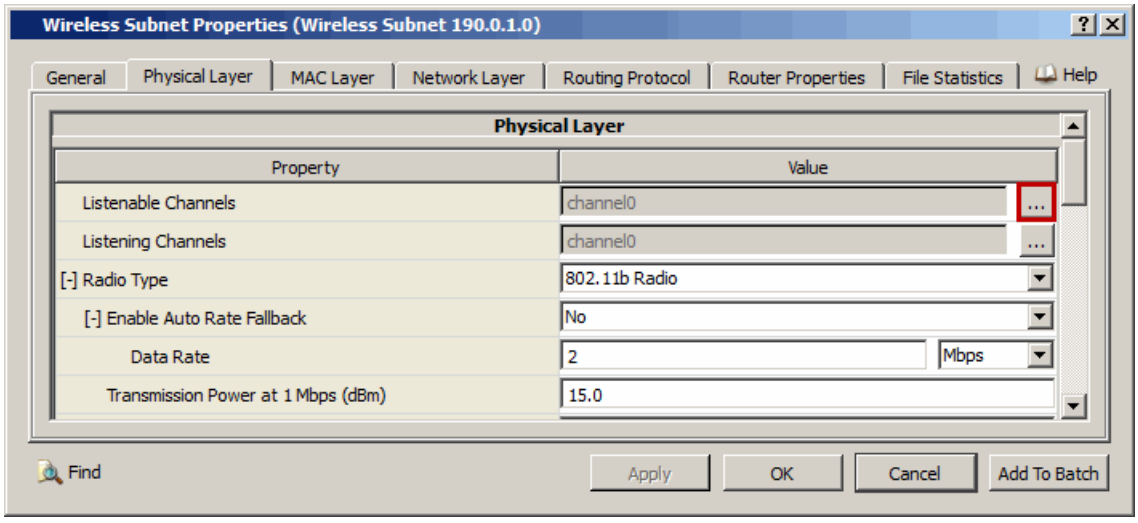


FIGURE 3-27. Setting Listenable Channel Mask

5. This opens the **PHY Channel List Editor**. Check both channels and click **OK** to close the **PHY Channel List Editor**.

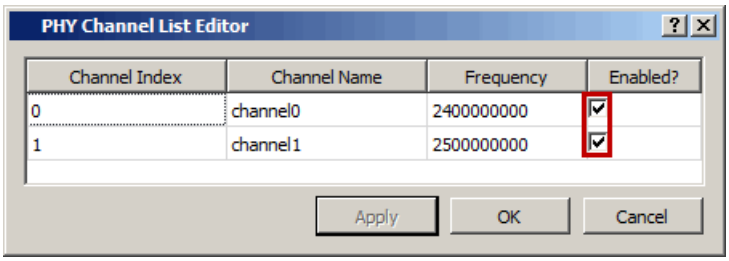


FIGURE 3-28. Channel Mask Editor for Listenable Channels

6. For the right subnet *only*, click the **Open Channel List Editor** ... button in the **Value** field of **Listening Channels**. (Do not modify the listening channels for the left subnet.)
7. This opens the **PHY Channel List Editor**. Uncheck channel0 and check channel1. Click **OK** to close the **PHY Channel List Editor**.

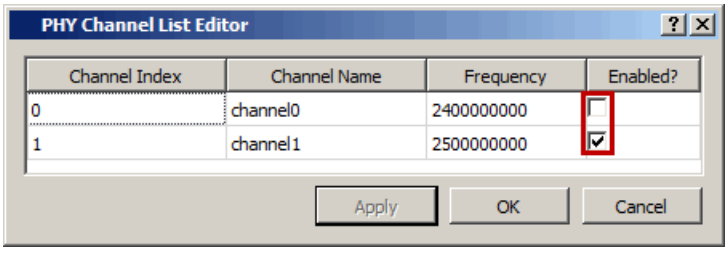


FIGURE 3-29. Channel Mask Editor for Listening Channels of Right Subnet

8. Then click **OK** to apply the changes and close the Properties Editor.

3.8.5 Application Properties

To set the properties of the CBR session, do the following:

1. Open the CBR Properties Editor by doing one of the following:
   - Enter select mode by pressing the **s** key and double-click on the CBR link on the canvas.

   OR
   - Click on **Table View** at the bottom of the canvas. Go to the Applications tab and double-click on the row labelled CBR.
2. In the CBR Properties Editor, set the parameters as shown below.

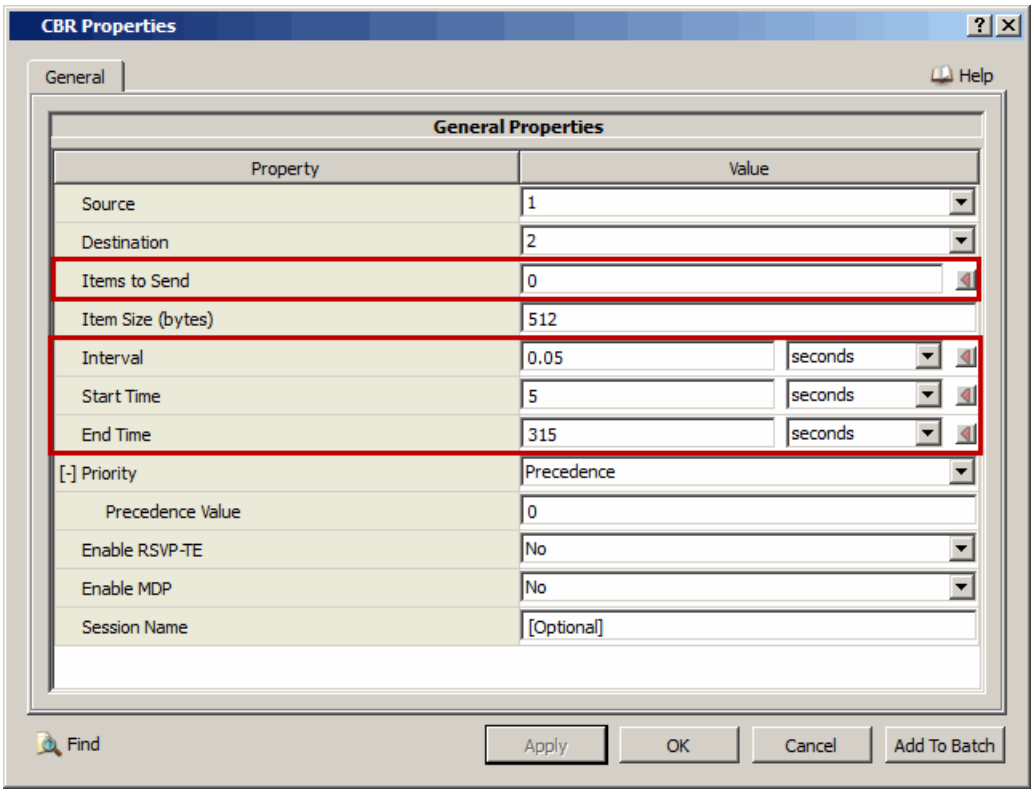


FIGURE 3-30. Setting CBR Application Properties

3.9 Saving and Running Scenario

Save and run the scenario by doing the following:

1. Save the scenario by clicking the **Save** button in the toolbar (see Figure 3-1).
2. Initialize the scenario by clicking the **Run SImulation** button. Start the simulation by clicking the **Play** button. See Chapter 2 for details of running the scenario and analyzing the results.