The GUI User Manual
for the EstiNet 9.0 Network Simulator

Release Date: August 3, 2015

Produced and maintained by EstiNet Technologies Inc.

Table of Contents

1. Environment Introduction

EstiNet provides a highly-integrated and professional GUI environment in which a user can easily conduct network simulations. EstiNet supports common wired and wireless network devices such as hub, switch, router, host, IEEE 802.11(a/g/n) ad hoc mode and infrastructure mode. EstiNet also supports VANET (IEEE 802.11(p), ITS) and OpenFlow. All nodes are shown as Figure 1.1.1.

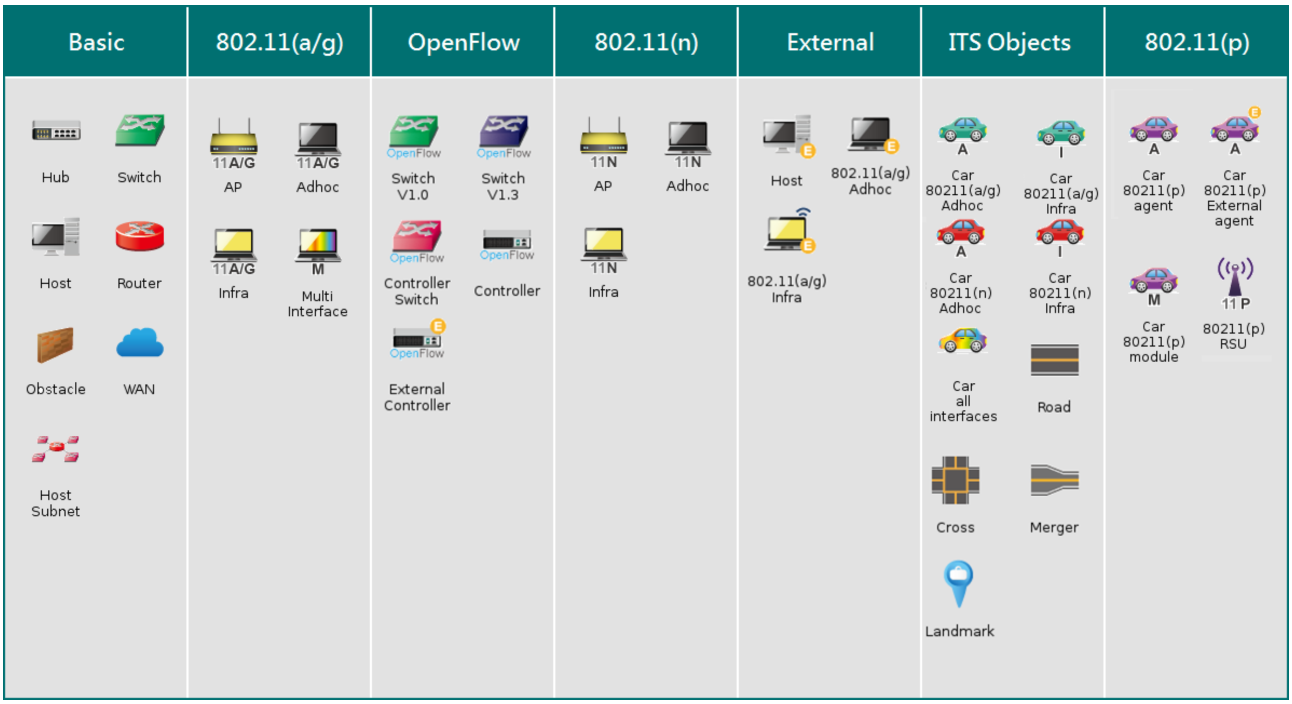


Figure 1.1.1 *EstiNet nodes*

In this manual, it introduces the node groups of Basic, 802.11(a/g) and 802.11(n). If you would like to understand the VANET and OpenFlow, please reference another two documents "EstiNet_9.0_OpenFlow1.3_GUI_Manual" and "EstiNet_9.0_VANET_GUI_Manual" for ITS Objects and 802.11(p).

There are five areas in EstiNet GUI (figure 1.1.2):

(1) **Menu Bar: The menus in the Menu bar allow user to access all available operations. There are 9 types of menus: File, Edit, G_Tools, N_Tools, G_Setting, N_Setting, Simulation, View, and Help.**

(2) **User Control: It provides icons of control functions to operate or modify network topology. The icons like zoom in or zoon out, four kinds of simulator operation modes [D, E, R, P].**

(3) **Network Node: It supports network nodes to draw network topology in working area.**

(4) **Working Area: The area that user could draw and modify network topology and setup the property of network node.**

(5) **PlayBack: It provides playback control buttons to display the simulation result in working area.**

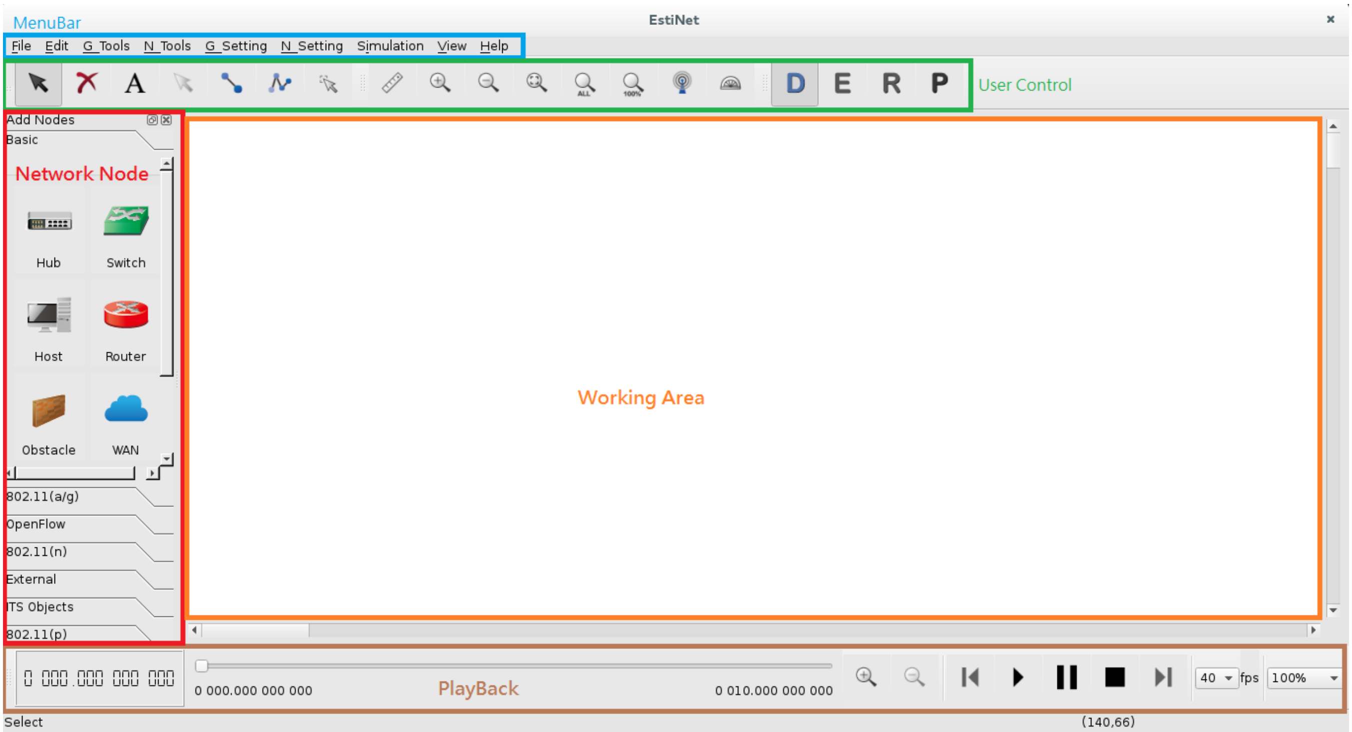


Figure 1.1.2 *EstiNet Startup Screen*

Besides, EstiNet adopts a distributed architecture. The main components show as figure 1.1.3. There are three main sections: GUI, Dispatcher and Simulation Server. GUI provides two main features: deploy network topology and playback the simulation results. Dispatcher transfers the simulation request or commands form GUI to Simulation Server and gets results from Simulation Server to GUI. Simulation Server receives the simulation request or commands via coordinator, and transfers the results to dispatcher via coordinator. GUI, Dispatcher and Coordinator use Inter- Process Communication (IPC, that is, sockets) to communicate for each other.

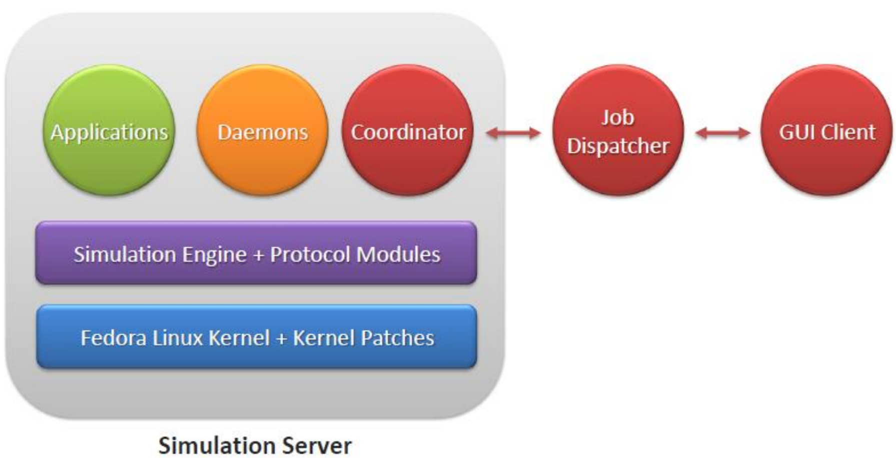


Figure 1.1.3 *EstiNet Architecture*

2. Operation Flow

Please follow the document EstiNet_9.0_InstallationGuide to open the EstiNet simulator. There are four modes in simulator operation flow: **D mode (Draw Topology), E mode (Edit Property), R mode (Run Simulation) and P mode (Playback)** as figure 2.1.5. The user could switch these four modes in toolbar (figure 2.1.6) or in menu bar **Menu → File → Operating Mode (**figure 2.1.7)**.**

Figure 2.1.5 *Flow diagram of EstiNet Simulator*

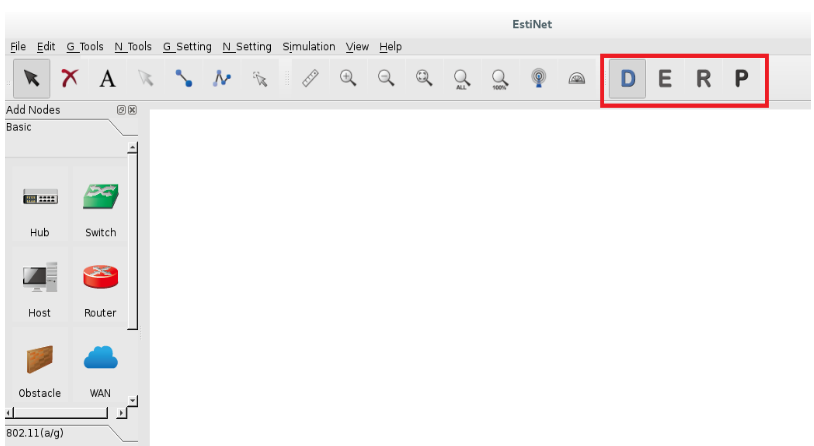


Figure 2.1.6 *Switch mode in toolbar*

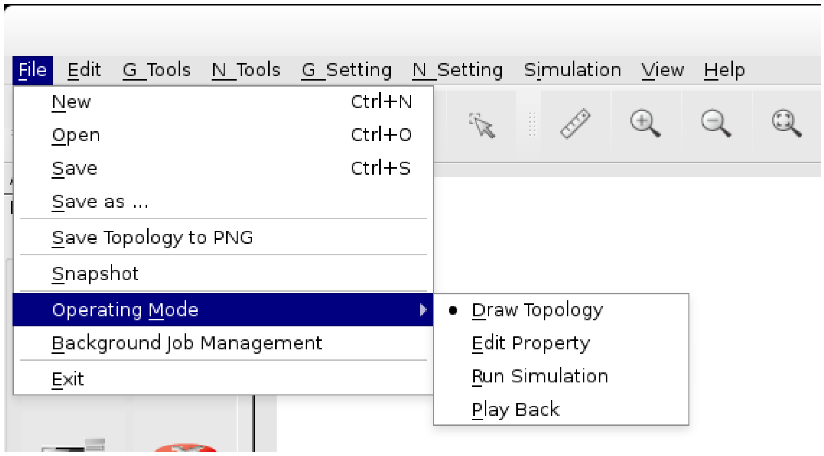


Figure 2.1.7 *Switch mode in menu bar*

It describes the usage of these four types as below.

D Mode  : Draw Topology

In this mode, user could deploy a network topology as what he/she like. We briefly describe how to draw a network topology step by step.

(1) Move the mouse cursor to the left area of Add Nodes. Then click tab of Basic.

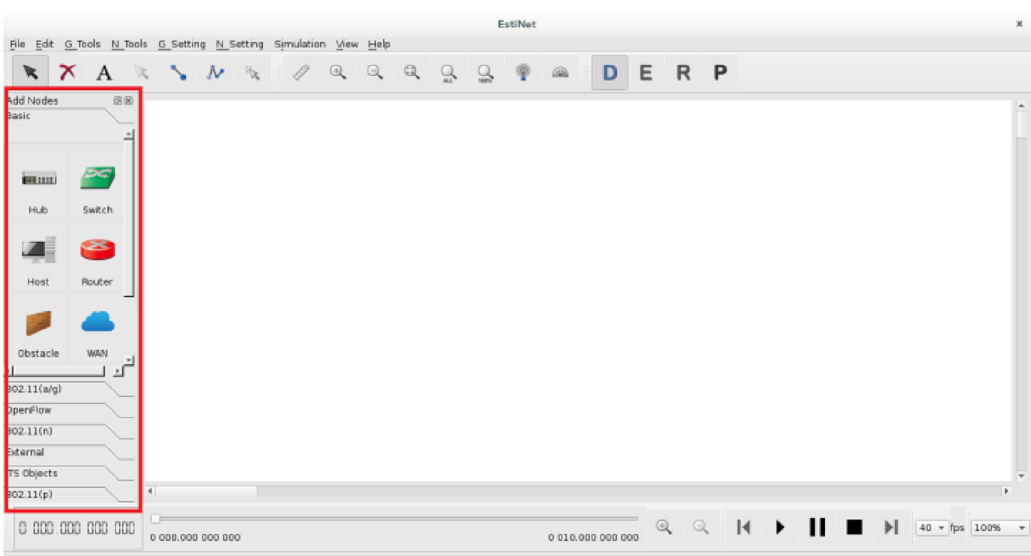


Figure 2.1.8 *Select Node*

(2) Click on the host node () and then left click at the desired location in the working area.

(3) To deploy another host node just clicking the left mouse button at the next desired location in the working area as figure 2.1.9.

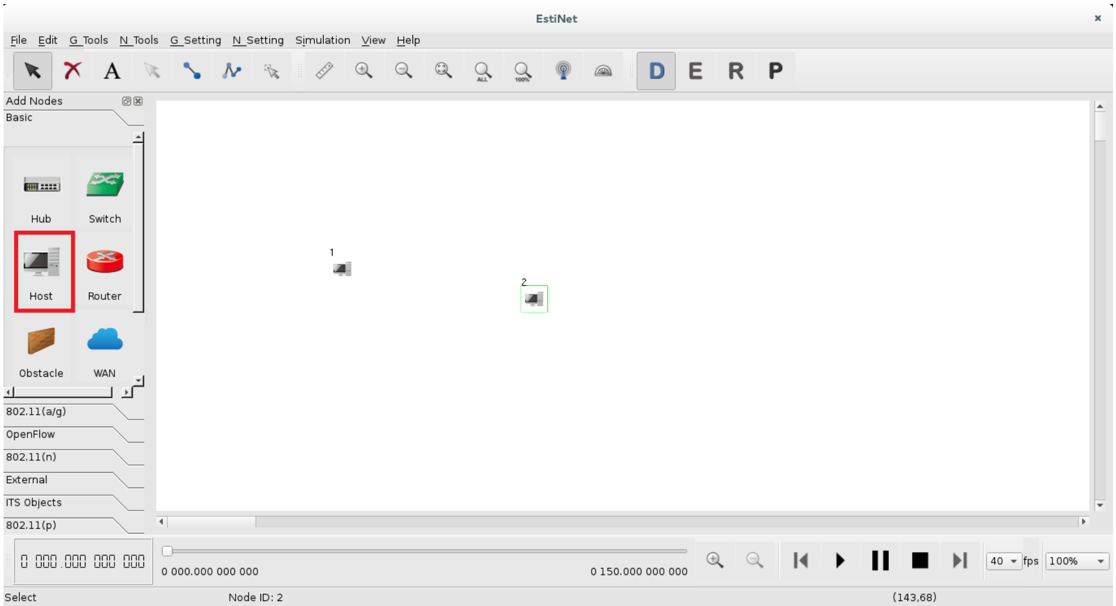


Figure 2.1.9 *Add two hosts into working area*

(4) Click on the **Create a point-to-point** link icon ( )as figure 2.1.9.Click and hold the left mouse button on the *host node1* then drag the cursor to the *host node2* and release the button to complete the connection. The linkage would be created between node1 and node2 as figure 2.1.11.

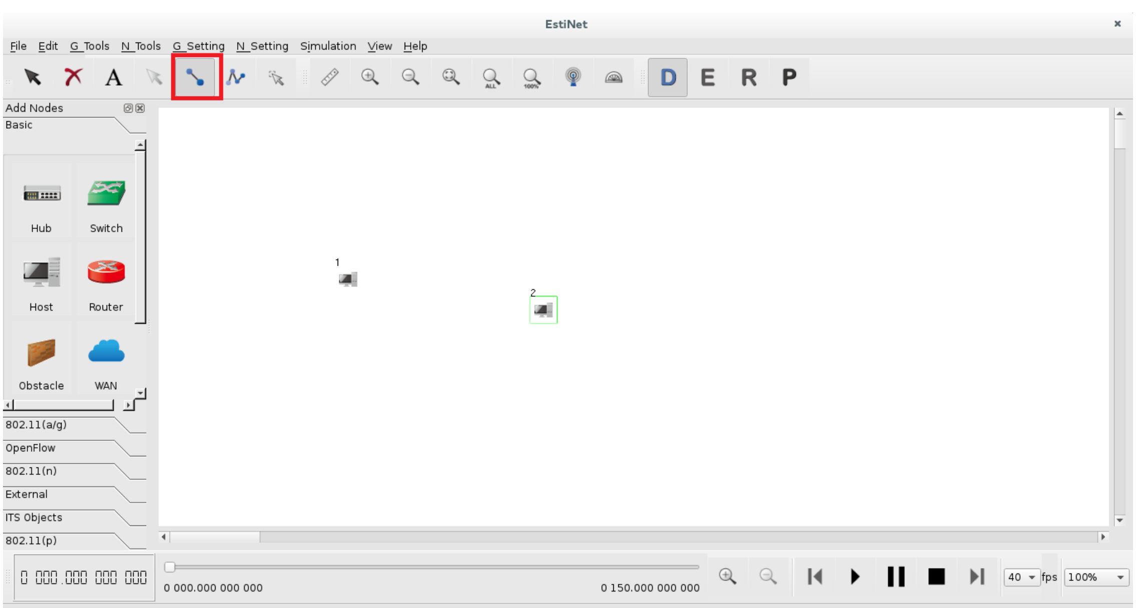


Figure 2.1.10 *Select "Link" icon*

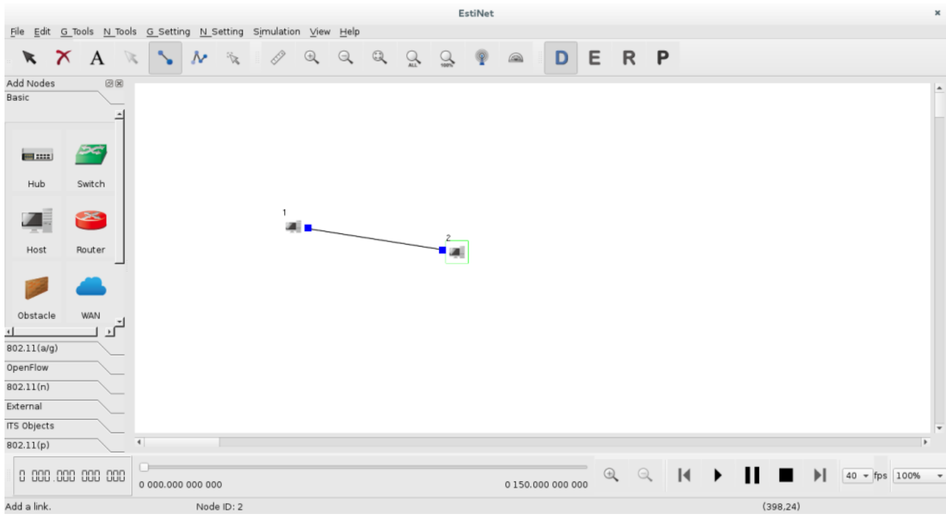


Figure 2.1.11 *Complete a simple network topology*

If user would like to change the location of *node*, click the icon **select a node** ( ). Then click the node and hold the left mouse button to another desired location in the working area.

If user would like to delete a node or link, just click the delete icon ( ) on the toolbar. Then click the **nodes** or **links** one by one. All of them would be deleted. If user would like to delete a lot of nodes, there are two methods to do it. Firstly, click the icon **select a node** ( ). Secondly, click the nodes with pressing **ctrl** key or pressing left button of mouse then draw an area which you would like to delete it. Then click right button of mouse and select **Delete Selected Nodes** or choose **Menu** → **Edit** → **Delete Selected Nodes** to complete it.

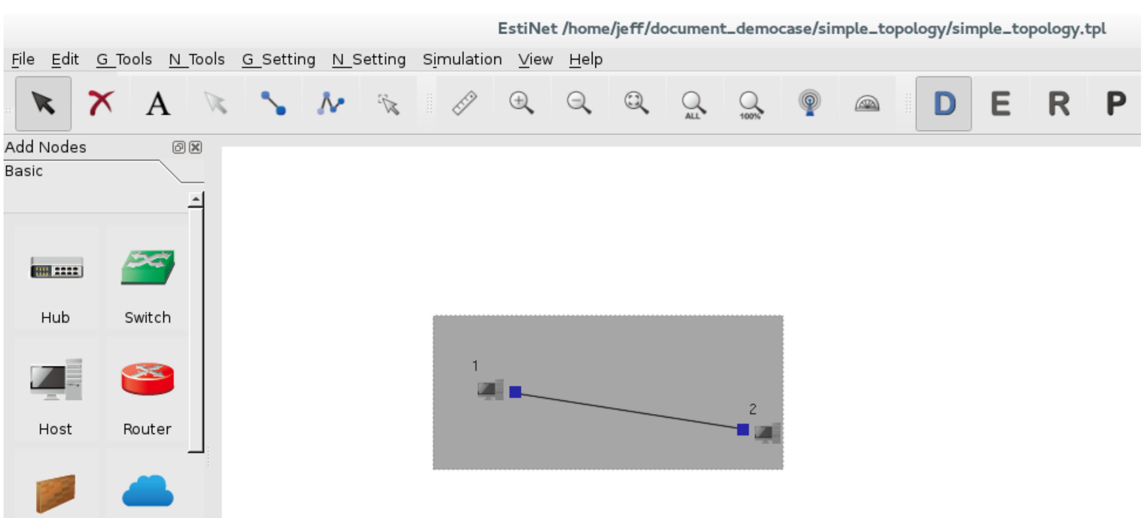


Figure 2.1.12 *Select nodes by dragging an area*

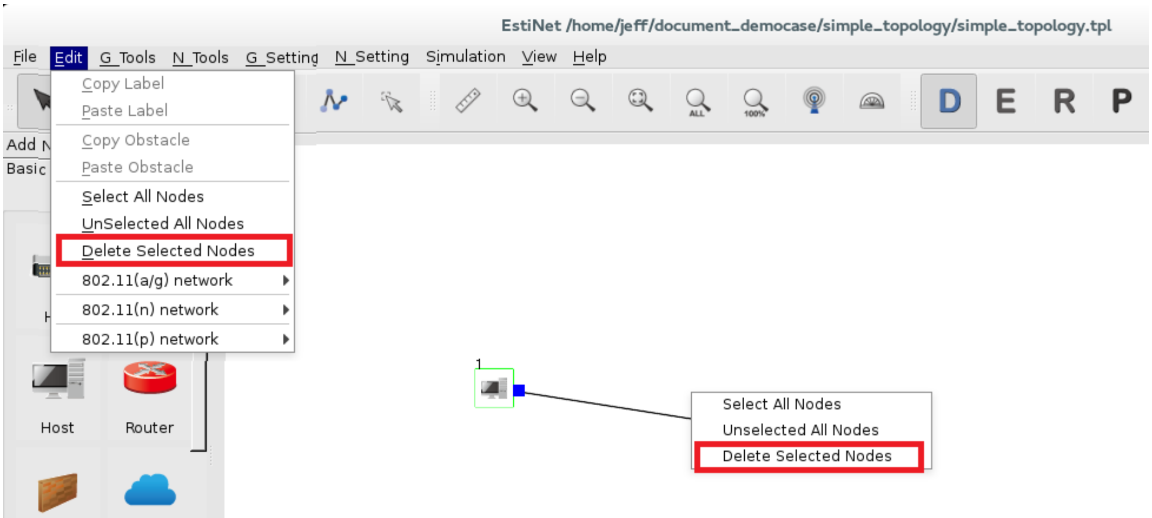


Figure 2.1.13 *Delete selected nodes*

(5) Remember to save this network topology by choosing **Menu** → **File** → **Save** or **Menu** → **File** → **Save As**. In this simple case, user could save this topology file as "simple_topology.tpl". Please notice that the file name could not using these special characters : space, *, ?, >, <, ;, &, !, [,], |, ", `, (,), {, } which are reserved for Linux system used. When user saves topology as file "simple_topology.tpl", GUI program will also create another file "simple_topology.xtpl" which stores the attribute values in XTML-like text format. User could check all attribute values in this file. User must save topology file when change operation mode from mode D Draw Topology to mode E Edit Property.

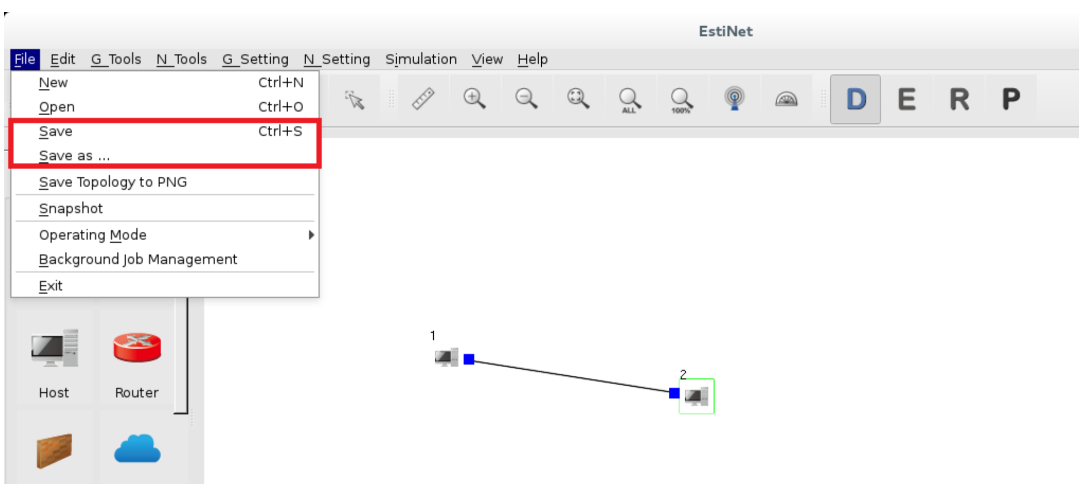


Figure 2.1.14 *Save network topology*

Please notice that only under **D Mode**, user could deploy a network topology.

E Mode E : Edit Property

When user switches to E Mode: Edit Property, the GUI program will automatically search subnets under a fixed network. Then it will generate and assign IP address, MAC address to all nodes which belong to layer-3 devices. It will also generate and assign MAC address to all nodes which belong to layer-2 devices. As you know, hosts, routers, or mobile nodes are layer-3 devices and switches or wireless LAN access points are layer-2 devices.

The format of IP address is **1.0.subnetID.hostNumOnThisSubnet** where **subnetID** and **host-NumOnThisSubnet** are assigned by GUI program. To get the IP information of node, please move the mouse curser onto a blue little square beside the node. The information of node *IP*, *Subnet ID* and *Port ID* would be shown as Figure 2.1.15.

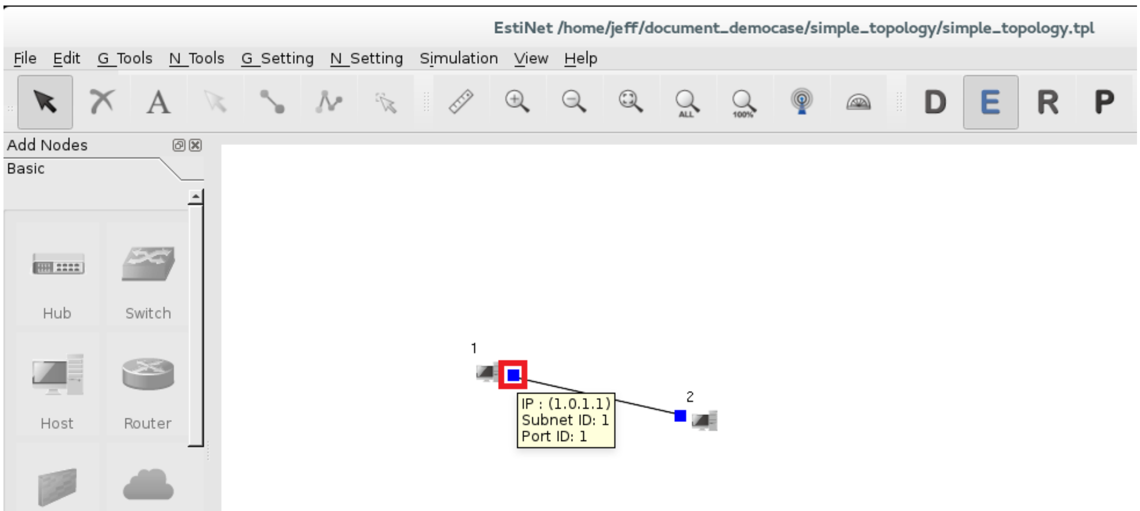


Figure 2.1.15 *Display the Information of Node*

Follow the format of EstiNet IP address, there are maximum 254 subnets in a topology. And there are maximum 254 nodes in a subnets. Please notice that 0 and 255 of the **subnetID** or the **host-NumOnThisSubnet** are occupied by network broadcast. To calculate the maximum number of layer-3 nodes in a topology, there are 254nodes/subnet * 254subnets = 64,516 nodes. Another limitation which related with layer-3 nodes is the numbers of tunnel interface 40,960 under Linux kernel. This limitation is designed by EstiNet. To conclude these two, user should understand the maximum layer-3 nodes in an EstiNet topology is 40,960.

One important task in E mode "Edit Property" is to specify which application program such as traffic generator should be run on which nodes during simulation. We briefly describe how to specify an application program step by step.

(1) Double click the host node1 then a dialog box would be pop up as Figure 2.1.16.

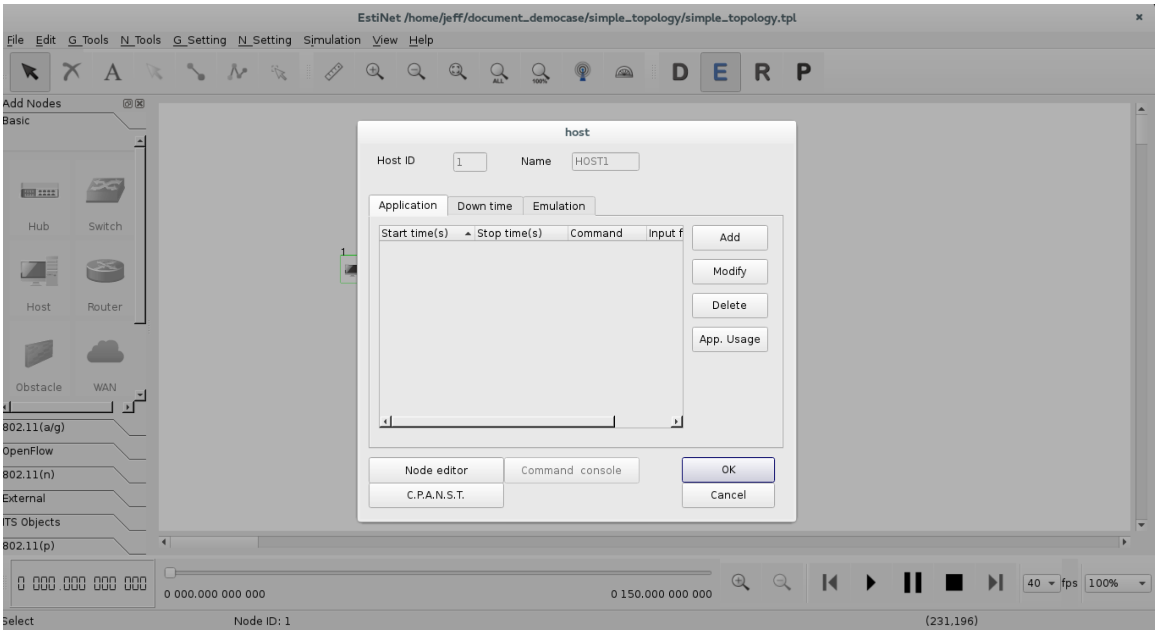


Figure 2.1.16 *A dialog box of host node1*

(2) To add an application traffic program just click "Application" tab then click "Add" button.

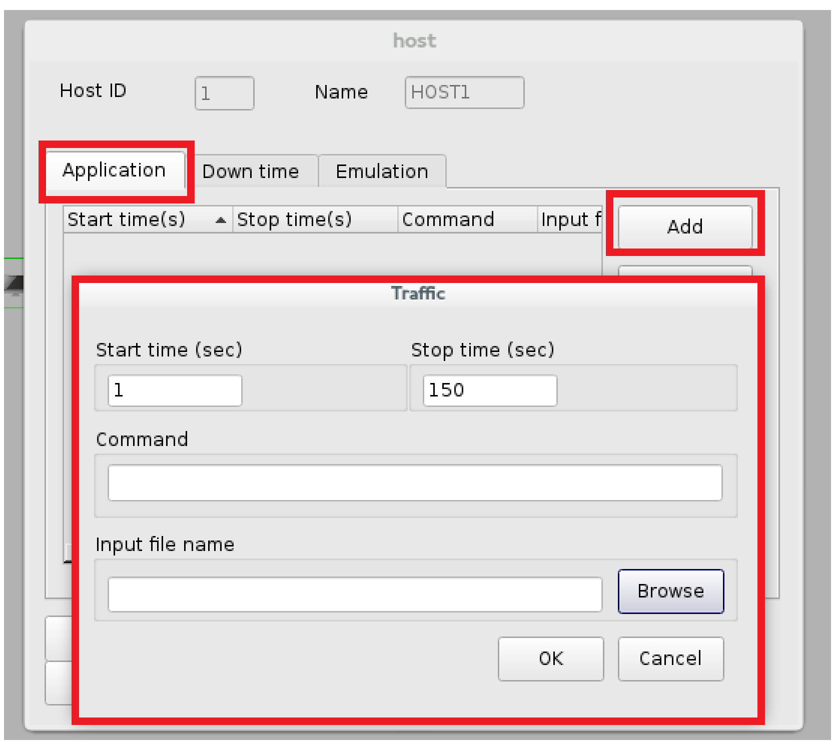


Figure 2.1.17 *Add an application program*

Please notice that the default value of an application's start time is set to 1 second. It's because in the simulation world, the operating system's clock is controlled to start from 0 second. In other words, the time of an application ticks from 0 second if it's start time is specified to be 0 second. However, in the real world, the operating system's clock never ticks from 0 second. For this reason, some real-world applications (e.g., Java programs, OpenDaylight controller, etc.) do not expect the time to be less than 1 second and are not programmed to handle this situation. These applications will behave abnormally when they run in the simulation world with their start time being specified less than 1 second. To avoid encountering this abnormal situation, we suggest that an application's start time be greater than or equal to 1 second. This is why the default value of an application's start time is set to 1 second.

(3) Set host node1 (1.0.1.1) as a sender which send packets to host node2 (1.0.1.2)

The host node1 (1.0.1.1) is configured to send TCP packets to host node2 (1.0.1.2) by application "stcp" from the 1st second to the 150th second. User should input "stcp 1.0.1.2" in command field as Figure 2.1.18. After click "**OK**" button, user could see the application list as Figure 2.1.19.

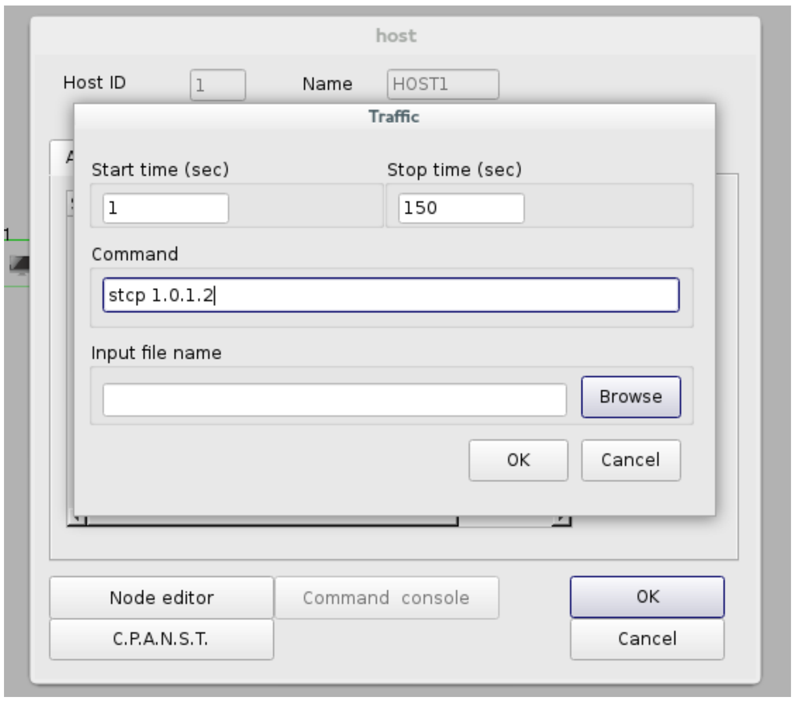


Figure 2.1.18 *Add command stcp in host node1*

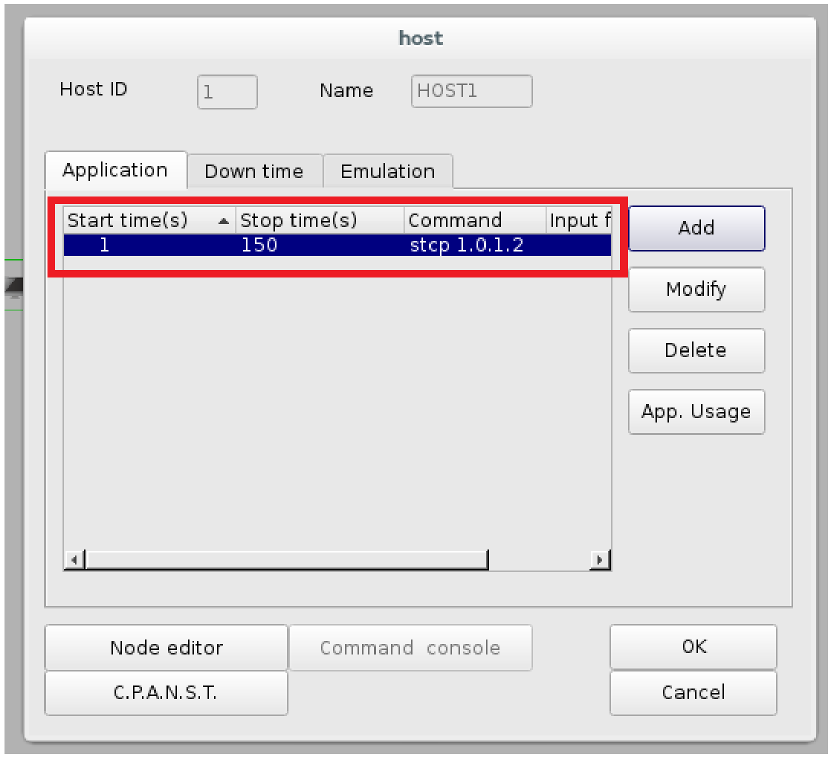


Figure 2.1.19 *The list of application program*

If user click the button "App. Usage", it will display a dialog of "*Application programs' user manuals*" as Figure 2.1.20. There are 7 kinds of pre-installed application which user could use them directly without installation.

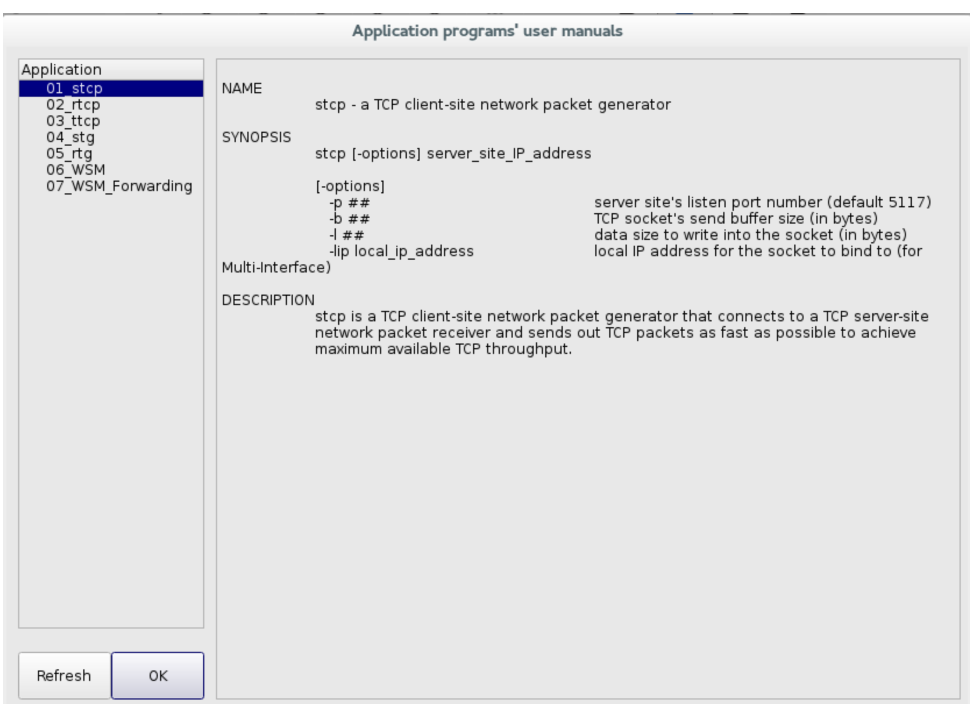
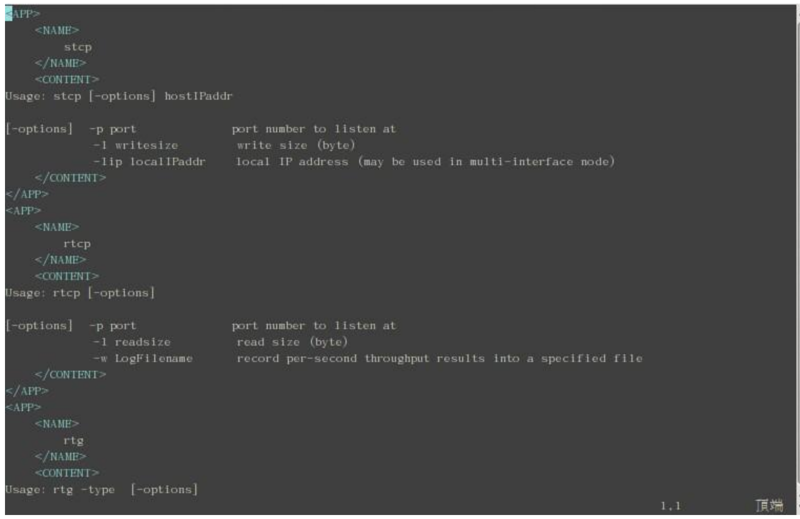


Figure 2.1.20 *Application programs' user manuals*

If user would like to install another application program, please copy the program into directory /usr/local/estinet/tools. When the program was input in command field as Figure 2.1.18, the simulation engine could execute it directly. If user would like to add another application usage information into the dialog of "*Application programs' user manuals*", please edit file app.xml under directory /usr/local/estinet/etc/ as figure 2.1.21.



```
<APP>
    <NAME>
        stcp
    </NAME>
    <CONTENT>
Usage: stcp [-options] hostIPaddr

[-options]  -p port             port number to listen at
            -l writesize         write size (byte)
            -lip localIPaddr     local IP address (may be used in multi-interface node)
    </CONTENT>
</APP>
<APP>
    <NAME>
        rtcp
    </NAME>
    <CONTENT>
Usage: rtcp [-options]

[-options]  -p port             port number to listen at
            -l readsize          read size (byte)
            -w LogFilename       record per-second throughput results into a specified file
    </CONTENT>
</APP>
<APP>
    <NAME>
        rtg
    </NAME>
    <CONTENT>
Usage: rtg -type  [-options]
                                                          1,1          頂端
```

Figure 2.1.21 *The content of the program usage information file (app.xml).*

(4) Set host node2 (1.0.1.2) as a receiver which receives packets from host node1 (1.0.1.1)

The host node2 (1.0.1.2) is configured to receive TCP packets from host node1 (1.0.1.1) by application "rtcp" from the 1st second to the 150th second. User should input "rtcp" in command field then click "**OK**" button as Figure 2.1.22.

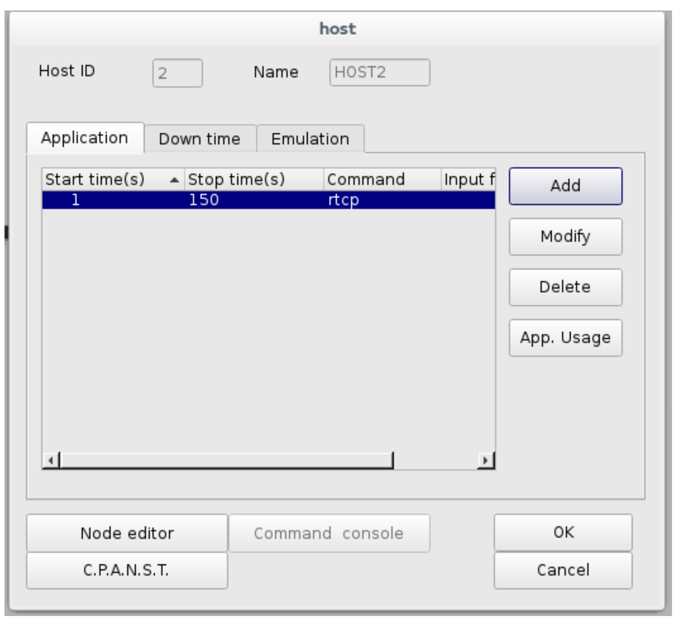


Figure 2.1.22 *Add command rtcp in host node2*

If user would like to change the network topology again, user should switch mode from E mode to D mode. Please notice that user must check all settings are correct or not when switch mode back from D mode to E mode. The GUI program will re- generate and re-assign IP and MAC addresses for all devices.

R Mode R : Run Simulation

When user switches to this R mode, the GUI will create “XXX.sim” directory and export many simulation configuration files that collectively describe the simulation case. After the simulation is finished, the GUI will create “XXX.results” directory to store the generated simulation results. The file name “XXX” is the same as current topology file name that user assigned and saved. In this sample, “XXX” is “simple_topology”.

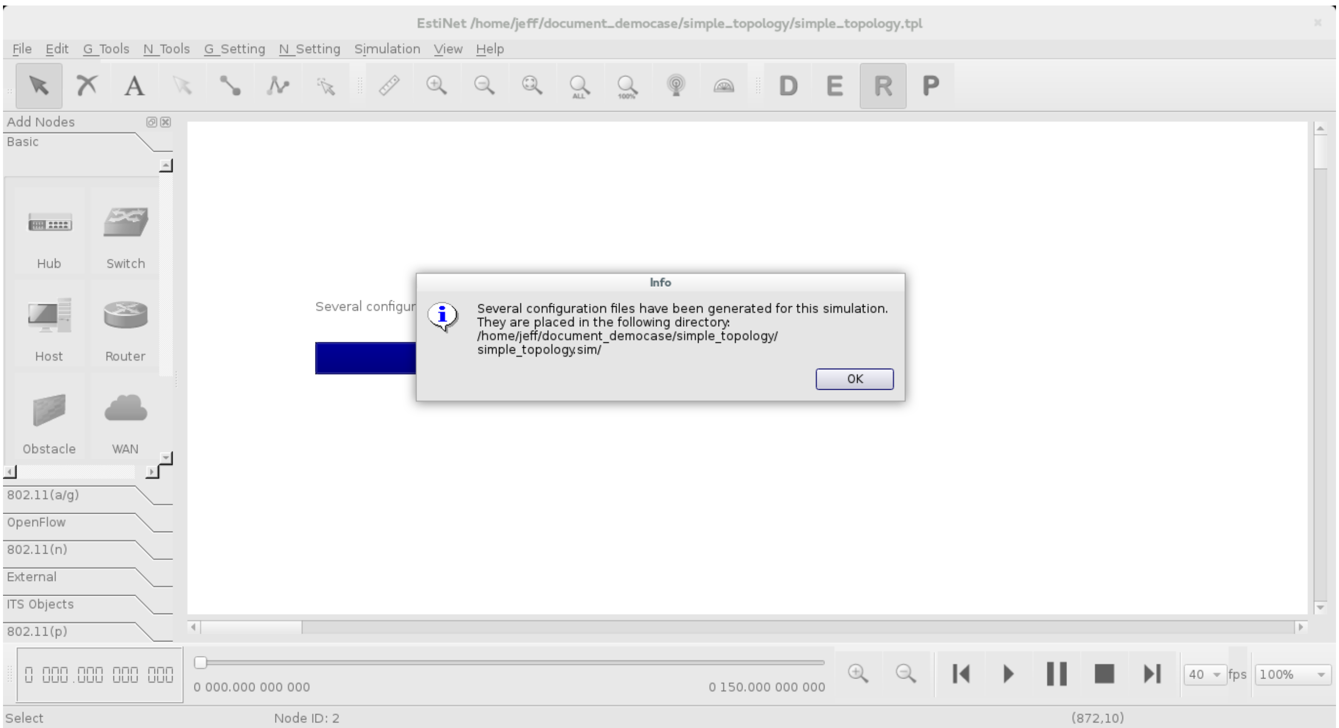


Figure 2.1.23 *The information dialog of configuration files*

Please choose **MenuBar Simulation → Run** to execute simulation. When simulator runs, it submits the current simulation job to simulation server which is managed by dispatcher. User could see the time knot is moving. The moving speed of time knot is related with simulation time.

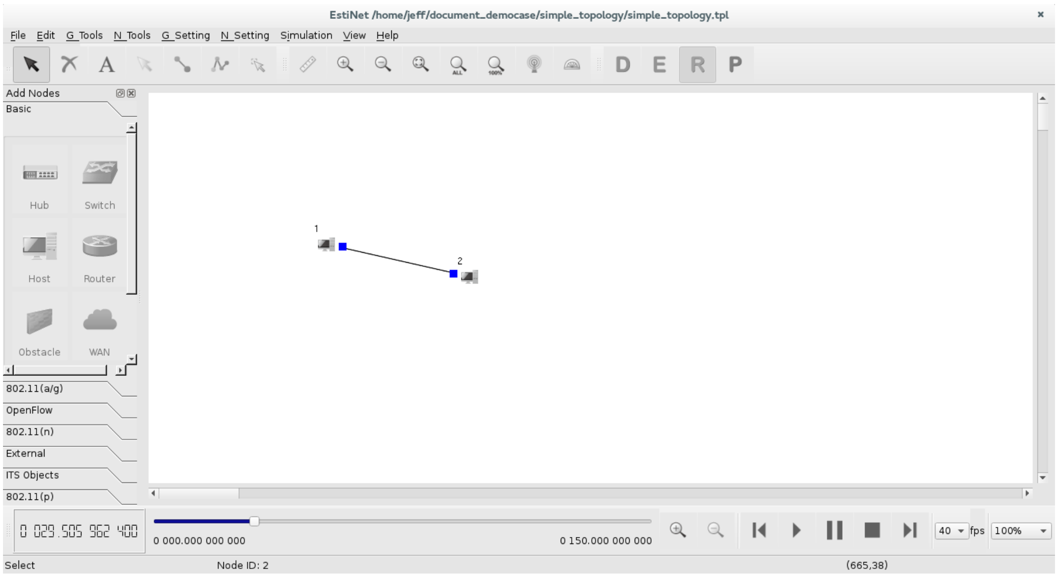


Figure 2.1.24 *Running simulation*

P Mode : Play Back

The simulation server will send back the simulation result files to GUI program after simulation is finished. The GUI program will store these files into directory "XXX.results" and automatically switch to "**Play Back**" mode.

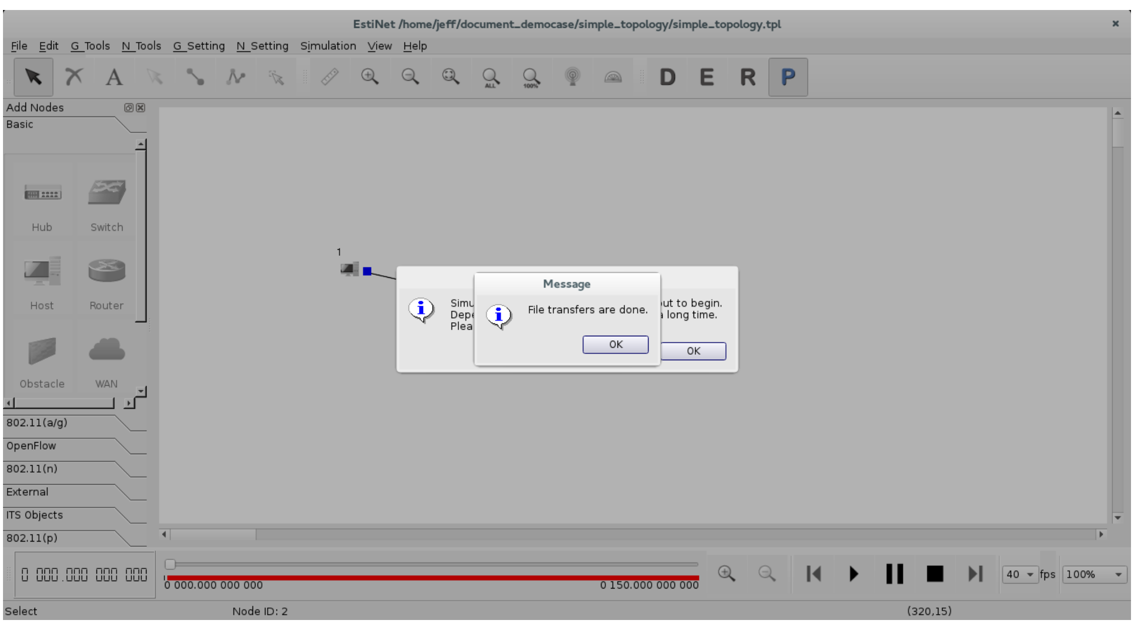


Figure 2.1.25 *Switch to the P mode after simulation finished*

To start the playback, user could click the play button ( ) in time bar which located at PlayBack area. The animation player will start to play the recorded packet animation.

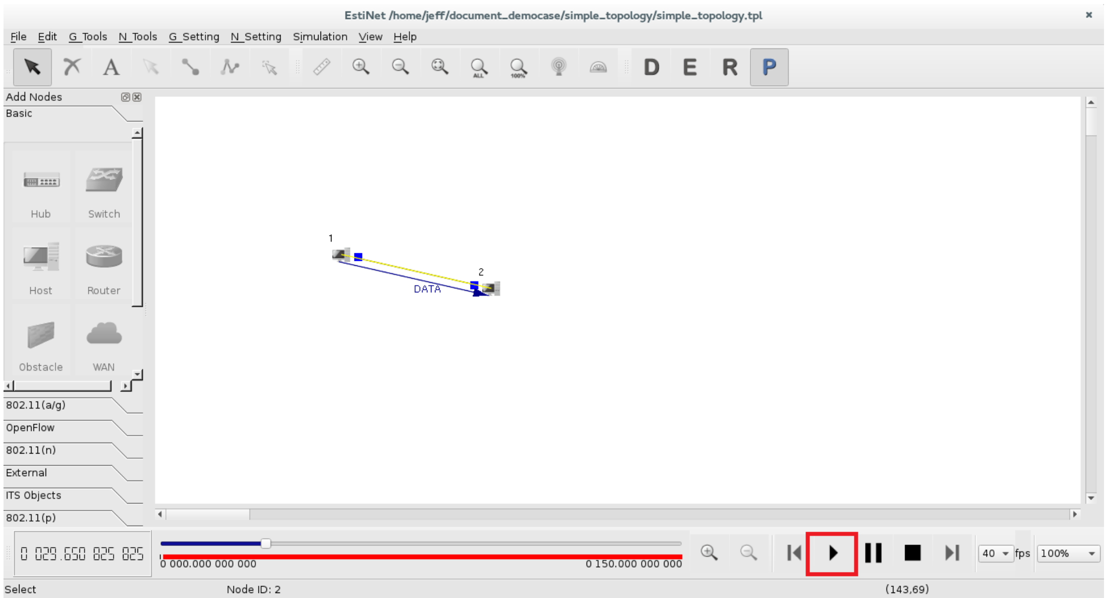


Figure 2.1.26 *Play the recorded packet animation*

Animation playing buttons are provided to alter playing sequences.

Figure 2.1.27 *Animation control buttons*

Play button ( ) plays the animation. If user clicks it while the animation player is idle, the player will start to play the packet animation trace. If user clicks it while the animation player is running, the time knot will jump directly to the nearest future time where there is a packet transmission. This feature is very useful when the traffic is sparse.

Pause button ( ) pauses the animation.

Stop button ( ) stops the animation.

Previous button ( ) moves the animation time window by one time window size in the backward direction.

Next button ( ) moves the animation time window by one time window size in the forward direction.

Figure 2.1.28 *The pull-down menus of animation playing speed*

The 40fps (frames-per-sec) selection box controls the display quality of the animation. It defines how many frames should be played in a second under real time clock. Using a smaller value can increase the animation speed because fewer CPU cycles are needed to refresh the screen. However, the resulting animation may become rigid and not smooth.

The 100% selection box controls the progress of the animation. It affects the time advancement quantity of the playback clock. During playback, the playback clock is advanced by a fixed time quantum in each of the playback loop. After the playback clock is advanced, all of the packet transfers whose transmission period (i.e., the period between the transmitting and receiving times) covers the current playback clock time are selected to be displayed on the screen. Choosing a larger value for this parameter will advance the progress of the animation playback more quickly. However, more packet transfers will be skipped and not displayed in the animation playback. Therefore, if packet transmission times over the links in a simulated network are tiny (e.g., tiny 58-byte TCP ACK packets transmitted on a 10 Mbps link or 1500-byte TCP data packets transmitted on a 1 Gbps link), it is better to use a smaller value for this parameter to see them.

The default value for this parameter is 100%. Other values such as 200% and 50% are provided. If the ratio of the selected value to the default value is X, the used time advancement quantity will be X times of the default time quantity.

3. GUI Simulation Settings

Before we briefly illustrate how to simulate by using GUI, you must understand how to setup various global parameters for each simulation case. The command for setting the various global parameters of each current simulation case is located in **Menu → G_Setting→ Simulation.**

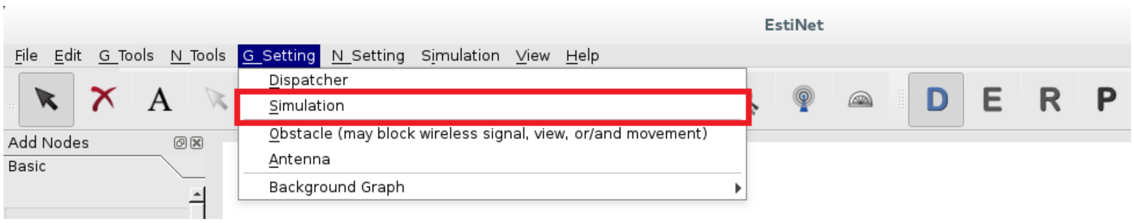


Figure 3.1.1 ***Menu → G_Setting→ Simulation***

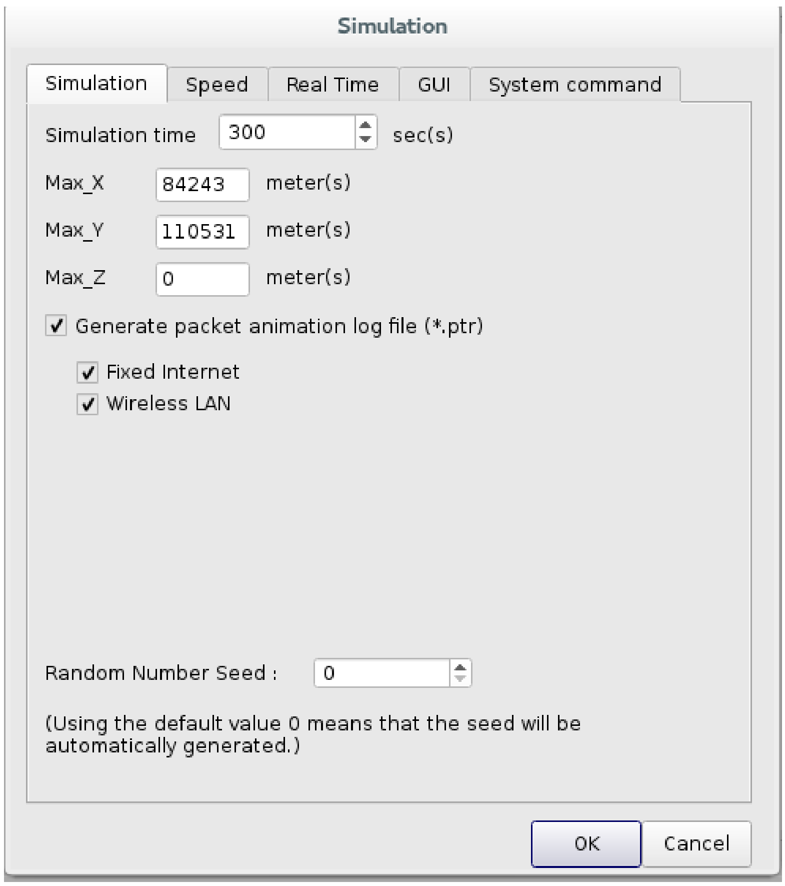


Figure 3.1.2 *The tab page of* ***Simulation***

Under the **[Simulation]** tab, there are several parameters to setup, as shown as below:

Simulation time: Specify the total simulation time in virtual time for simulation, and the maximum time that can be simulated is 99,999 seconds which close to 27 hours in virtual time.

Max_X: Specify the maximum value for X coordinate.

Max_Y: Specify the maximum value for Y coordinate.

Max_Z: It is not used right now, because GUI does not support three dimensions.

Generate packet animation log file (*.ptr): User has a selection whether the .ptr file (packet transfer trace) should be generated or not and further types of network transfer packets (Fixed Internet or Wireless LAN) should be logged into the ptr file. The default values are all enabled.

Random Number Seed: User could specify the value given to the simulation engine for this simulation. Using the default value of 0 indicates to the simulation engine that it can choose a random number for the random number seed. That is, each time the same simulation case is run, a different random number seed will be used. If the random number seed is fixed to a value greater than 0, EstiNet can generate repeatable results for each run of the same simulation case.

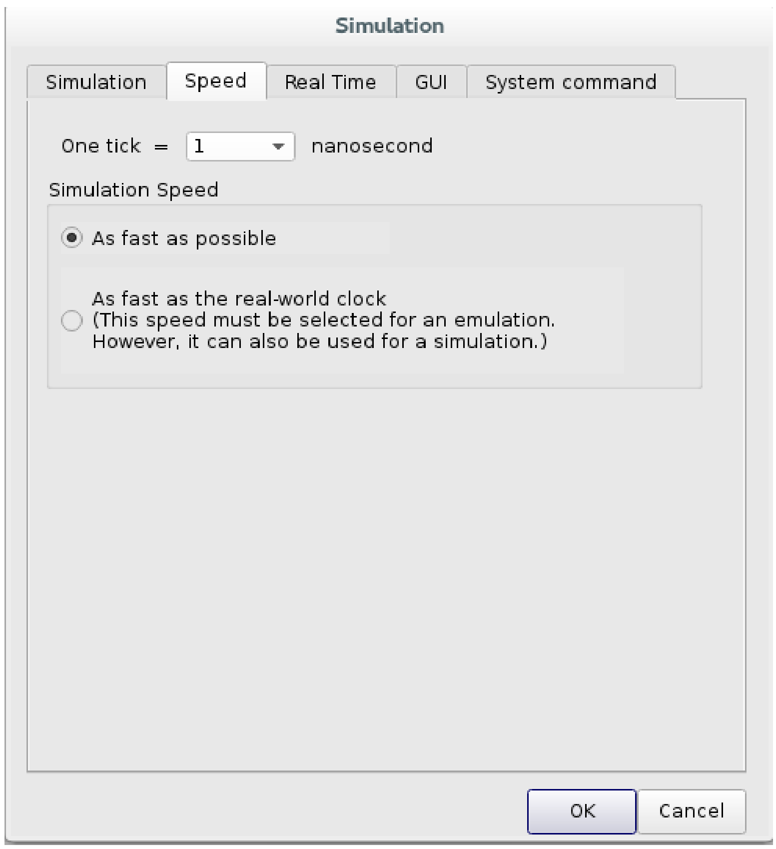


Figure 3.1.3 *The tab page of* ***Speed***

Under the **[Speed]** tab, one tick represents 1 nanoseconds in virtual time under a simulation.

As fast as possible: If user would like a simulation case to be finished as soon as possible, please choose this option. And it's the default setting.

As fast as the real-world clock: If user would like to do emulation or query some information in real-time clock, user must choose this which sync simulation clock with real-world clock.

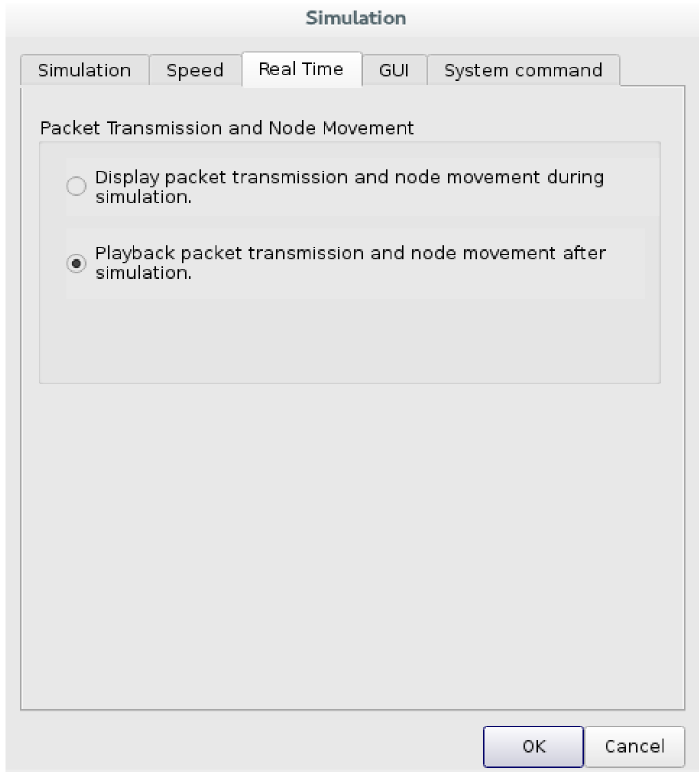


Figure 3.1.3 *The tab page of* ***Real Time***

Under the **[Real Time]** tab, if user want to see the display of packet transmission and node movement during a simulation, the option "**Display packet transmission and node movement during simulation**" has to be chosen. Otherwise, choose the other option "**Playback packet transmission and node movement after simulation**" to see the animation after simulation. This is the default selection.

Figure 3.1.4 *The tab page of **GUI***

Under the **[GUI]** tab, the parameters are shown as below:

Proportion: Changing this setting is useful for a network topology which loads a very large scale of background map. Using a large value can allow the whole network to be clearly shown on the screen without using the “Zoom Out” function. Although the user can use the default value (i.e., 1) and the “Zoom Out” function to display the whole network topology on the screen, node icons will become too tiny to be seen on the screen and it will be very difficult for the user to manipulate these tiny icons (e.g., to select them).

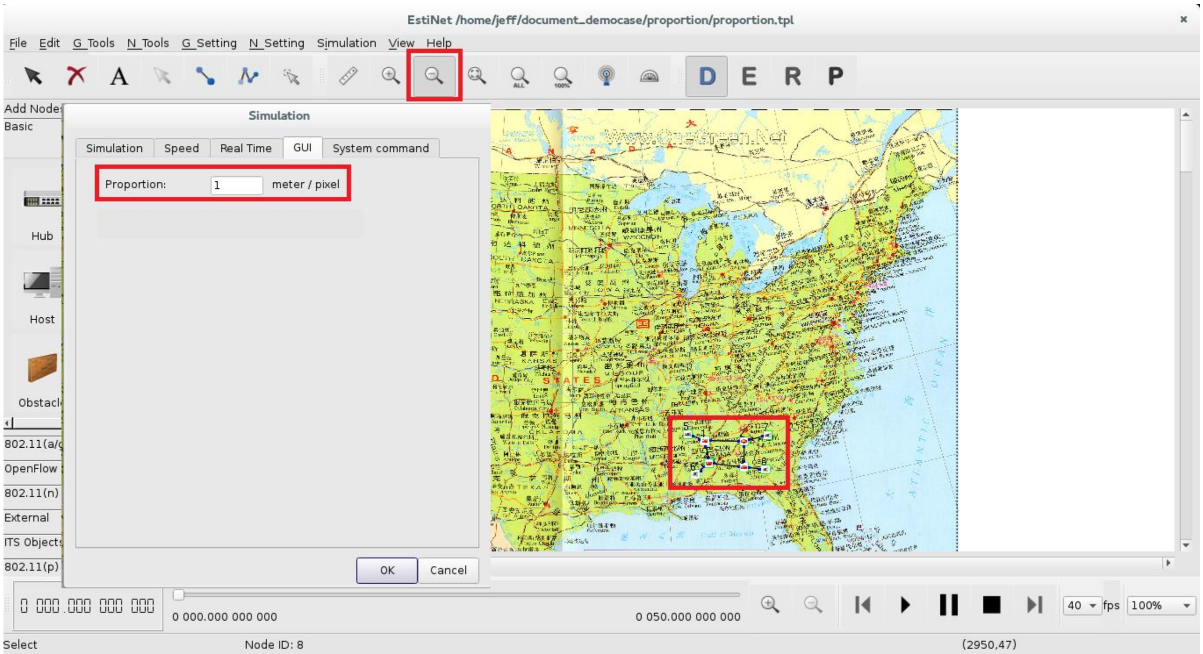


Figure 3.1.5 *A large background map with Proportion = 1*

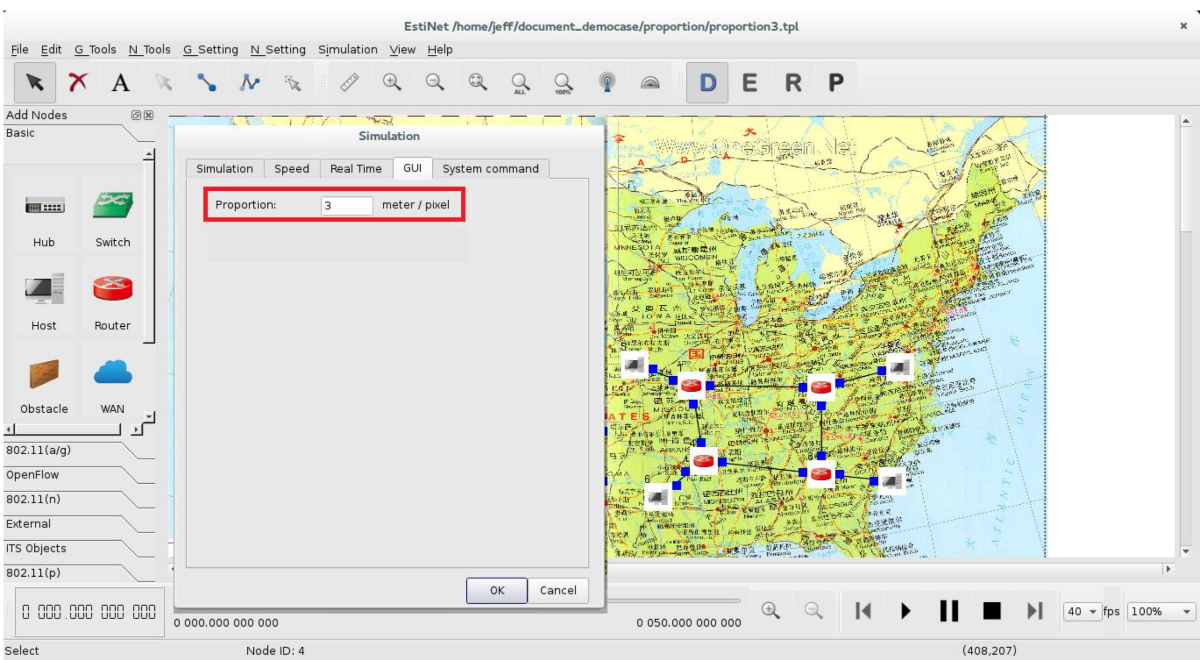


Figure 3.1.6 *A large background map with Proportion = 3*

Figure 3.1.7 *The tab page of* ***System command***

Under the **[System command]** tab, to-be-executed system commands and their output file names can be specified here. A system command is a command that, when executed, will get or set an object's value at the specified time. The output of the command will be saved to the specified output file, which will later be transferred back to the GUI program when the simulation is finished.

In addition to getting or setting a single object's value, the values of all objects of the same kind can be get or set at the same time to take a global snapshot of the whole network. For example, this function can be used to take the snapshot of the current arp tables of node1 as Figure 3.1.8. This global snapshot information can help a researcher study the convergence of a protocol.

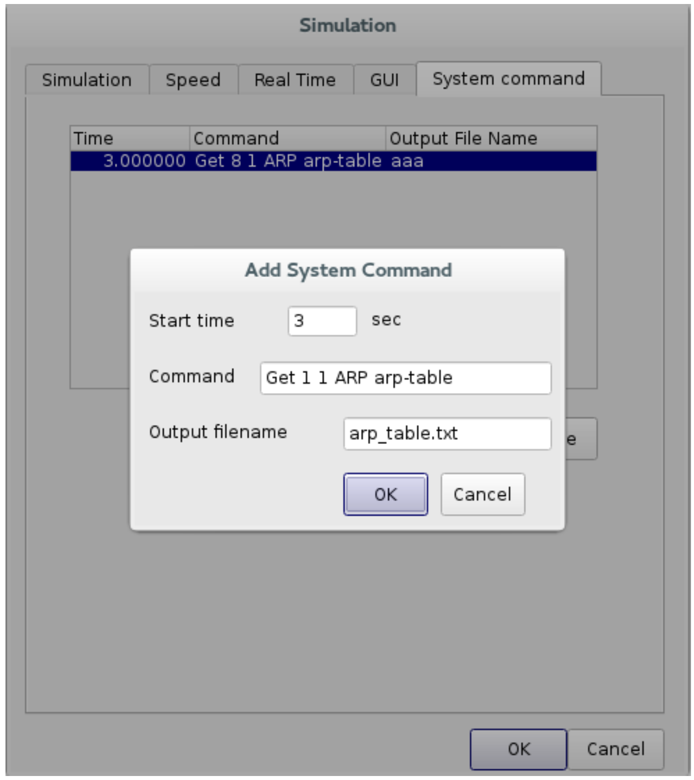


Figure 3.1.7 *Add System command*

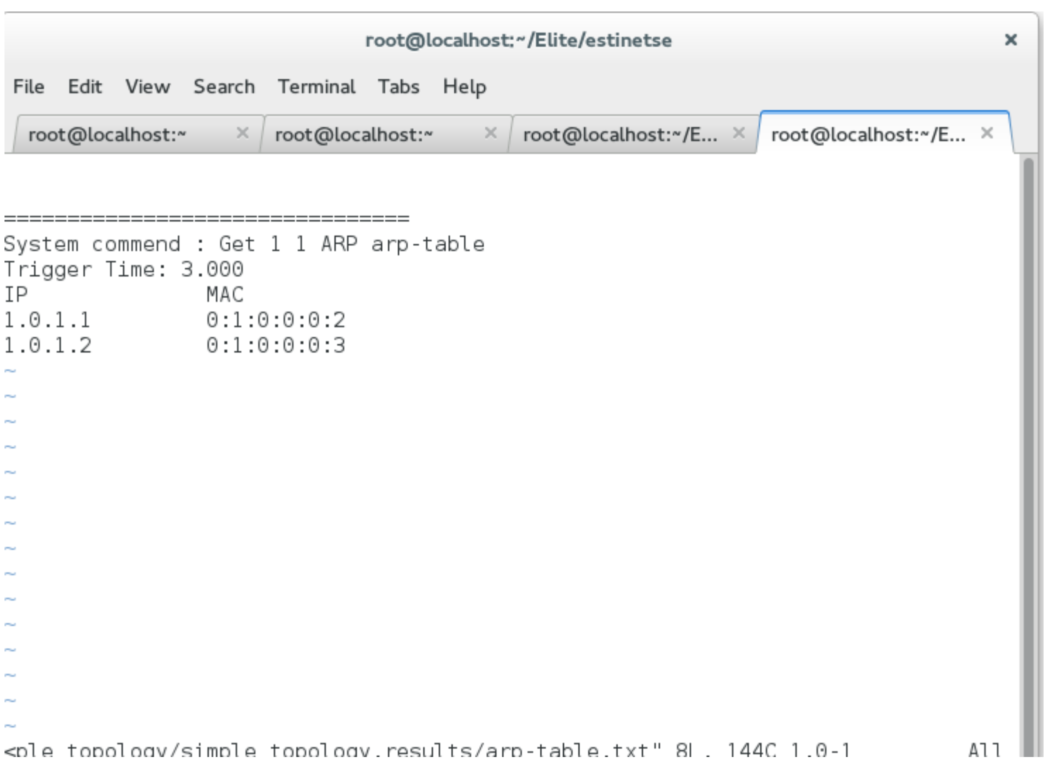


Figure 3.1.7 *Add System command Output file arp_table.txt*

Below is the usage description for **Add System command.**

Time: the starting time for executing this command

Command: the system command string

Output file name: the name of the output file

The system commands provided by EstiNet are listed below. Their syntax and meanings are also explained below.

Set: set a value to a variable in a module: Set {node} {0} {module} {tag} {value}

Get: get the value of a variable from a module: Get {node} {0} {module} {tag}

Note that a tag is a string associated with a particular variable declared and used in a protocol module. A variable here can be a single-value variable (e.g., a FIFO queue's maximum queue length) or a multi-column multi-row table (e.g., an arp table that has multiple [IP address, MAC address] mapping entries). It is the protocol module developer's job to write a command() method in his (her) module to recognize a tag and then get/set the value of its associated variable.

More information about the get/set command can be found in "Estinet 9.0 Protocol Developer Manual".

4. Ethernet (IEEE 802.3)

In this chapter, we will introduce wired internet with hub, switch and router. And pass through this case, let users know how to quickly and expediently operate GUI and run simulation.

(1) Draw topology of wired internet

In D mode D , we use these nodes: host, hub, switch and router in "Basic" group to draw the simple topology as figure 4.1.1 and save the topology file as "wired_internet.tpl."

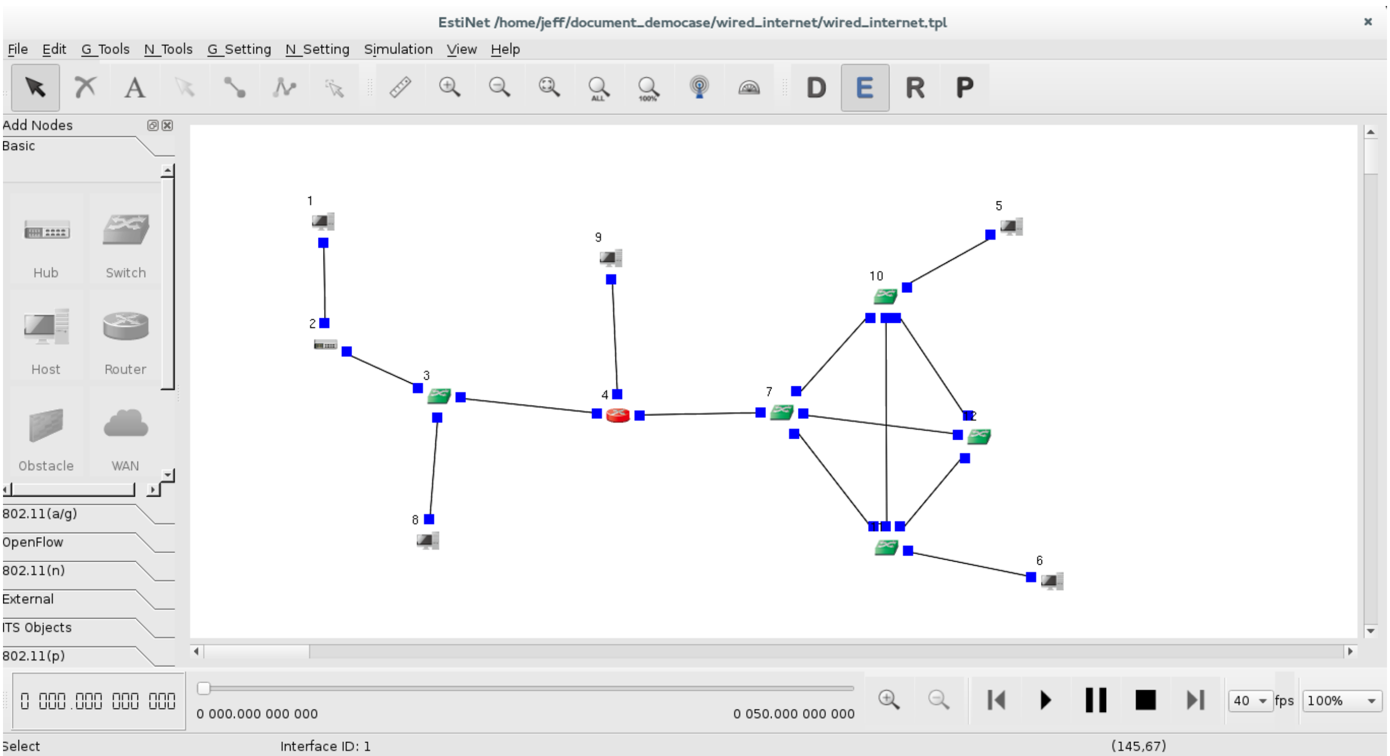


Figure 4.1.1 *The Ethernet topology*

(2) Setup Properties

Switch to E mode E and GUI automatically subdivides three subnets and assigns IP and MAC addresses as figure 4.1.2. If user want to get a node's IP address, user can move the mouse to a little blue square and GUI will show the node's IP and subnet information as figure 4.1.2. If user want to get a node's MAC address, user can right click on the node and select "Node Editor". It will pop up a dialog that shows the node's protocol stack. Double click MAC8023 module MAC8023 , it will pop up a dialog that shows MAC address as figure 4.1.3.

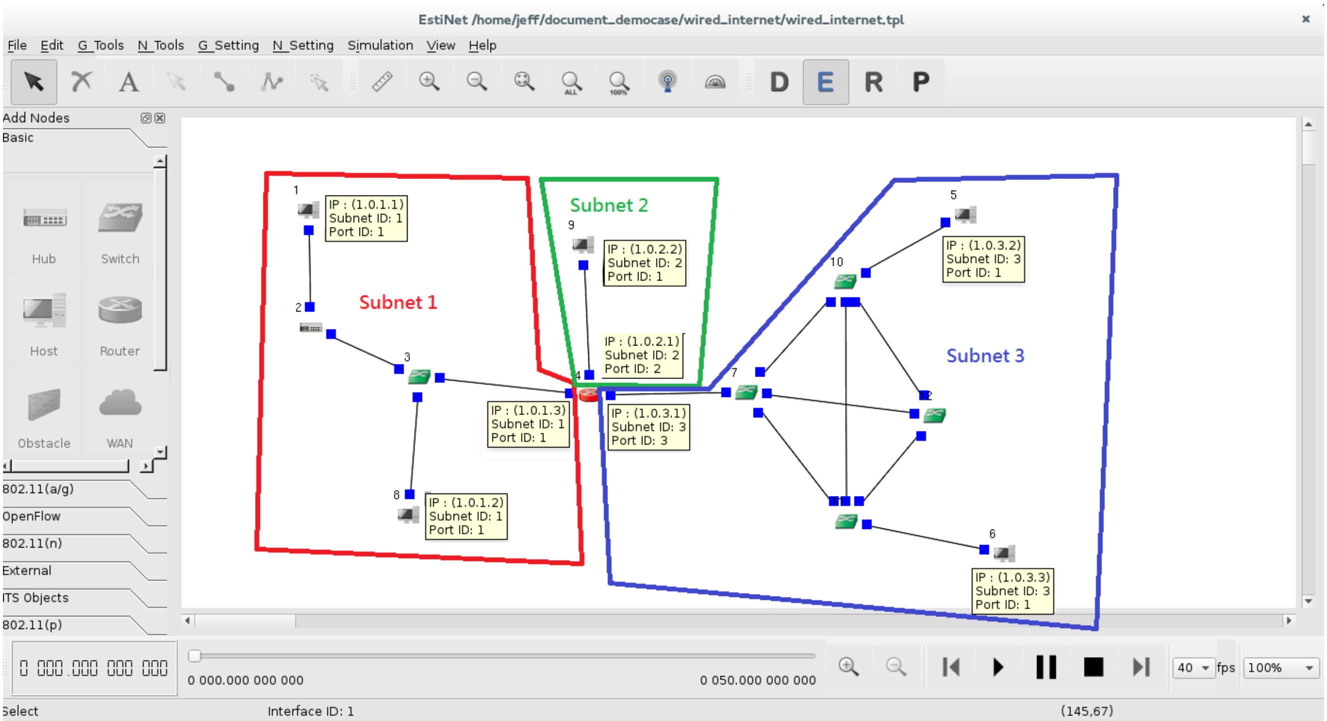


Figure 4.1.2 *The node information of IP address and subnet*

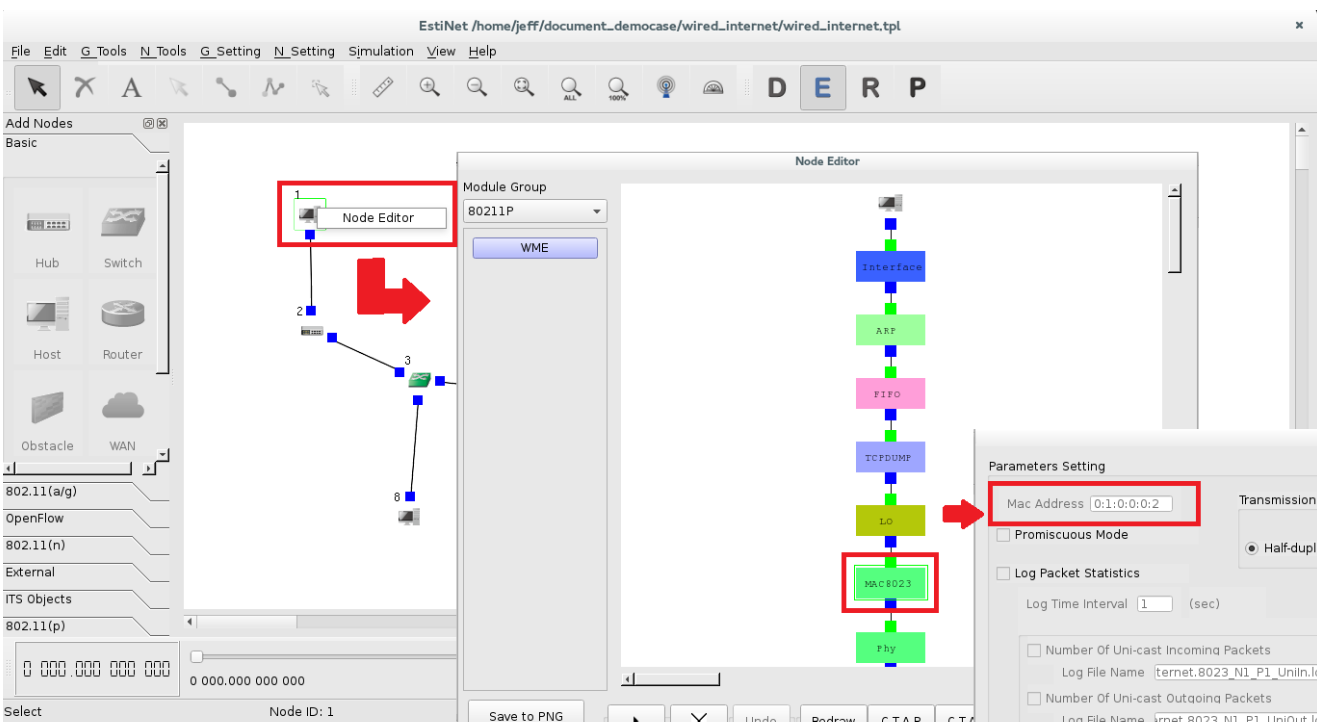


Figure 4.1.3 *MAC address of node1*

A. Set Network Traffic:

In this diagram, node1 send TCP packets to node5 and node9 send UDP packets to node6 and node8.

Node 1 is configured to send TCP packets to node 5. The TCP sender program is configured by double clicking on the node 1. Then add the command “ttcp -t -s -p 8000 1.0.3.2” with the 1st second to the 50th second in the “Application” tab.

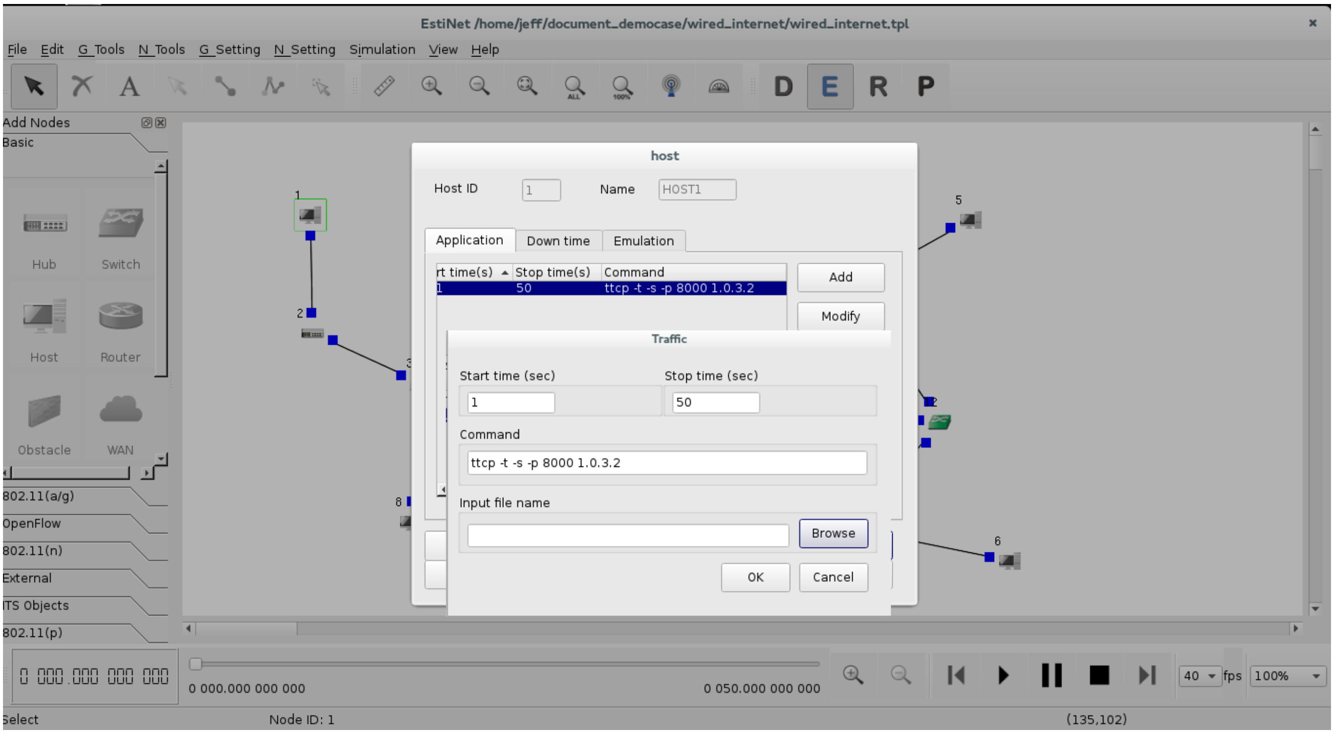


Figure 4.1.4 *Add application to node1*

Double click node5 and add the command "ttcp -r -s -p 8000" in the "Application" tab.

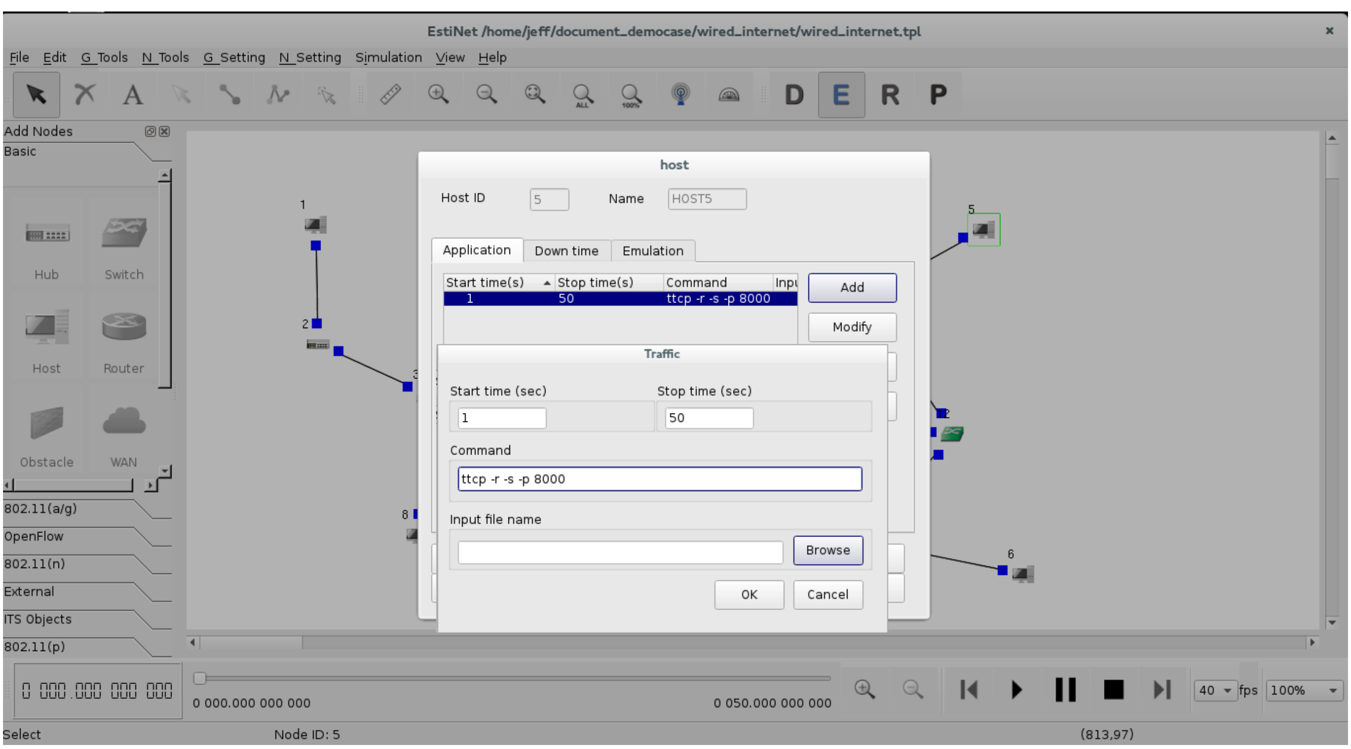


Figure 4.1.5 *Add application to node5*

Double click node9 and add these two commands: "ttcp -t -s -u -p 8001 1.0.1.2", "ttcp -t -s -u -p 8002 1.0.3.3" in the "Application" tab.

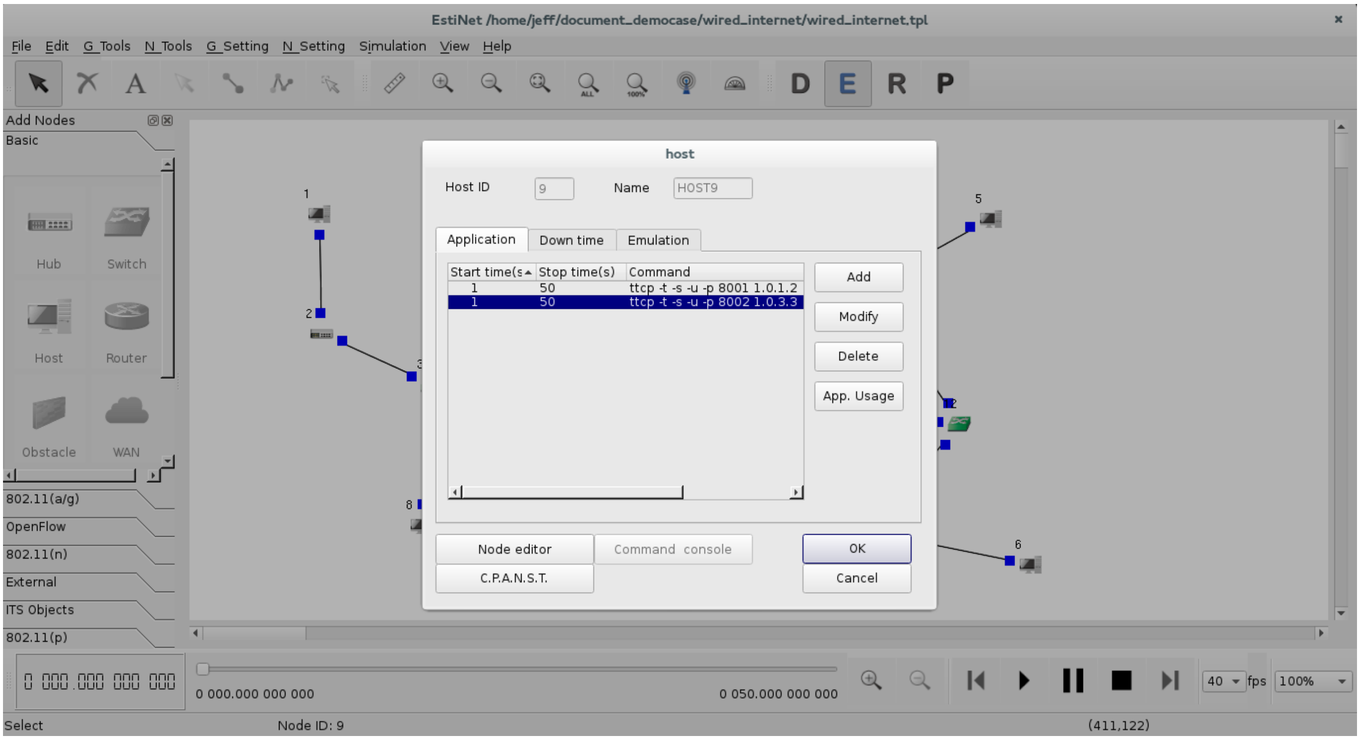


Figure 4.1.6 *Add application to node9*

Double click node6 and add the command "ttcp -r -s –u -p 8002" in the "Application" tab.

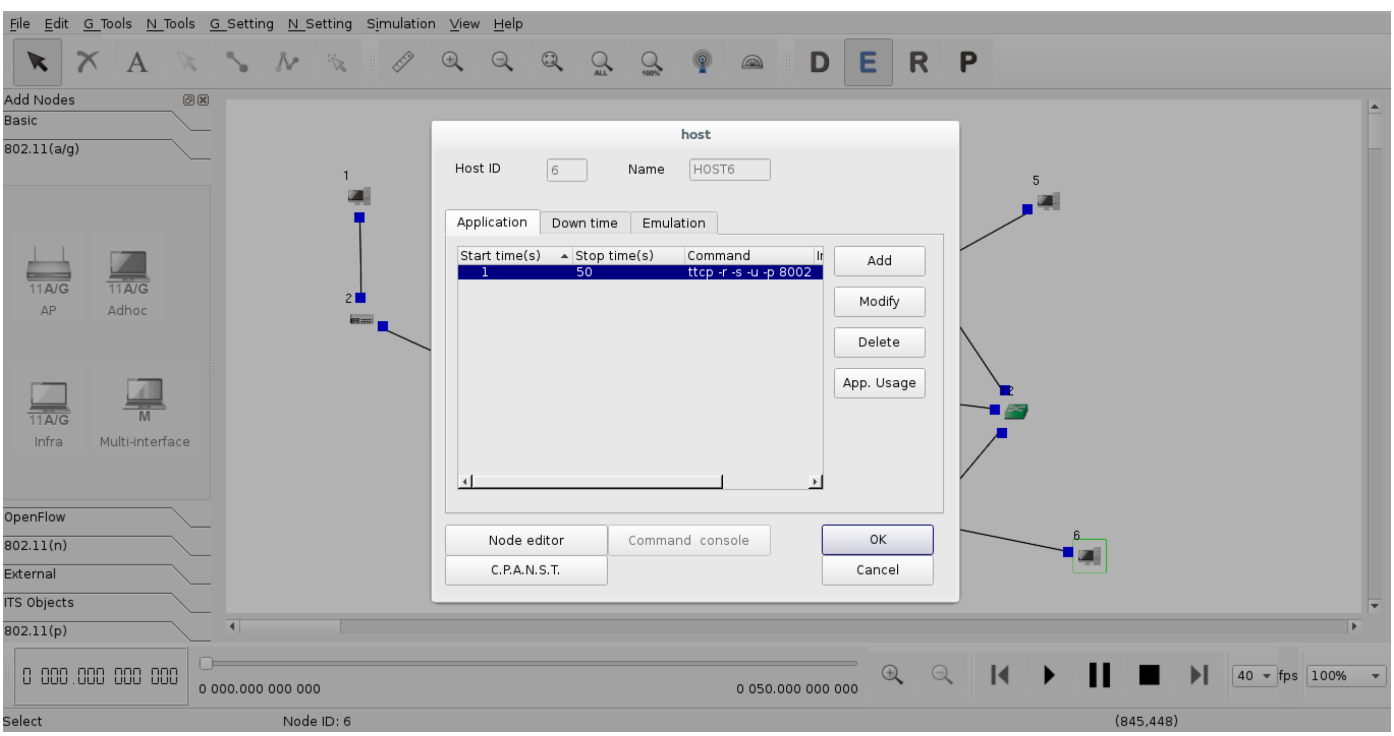


Figure 4.1.7 *Add application to node6*

Double click node8 and add the command "ttcp -r -s –u -p 8001" in the "Application" tab.

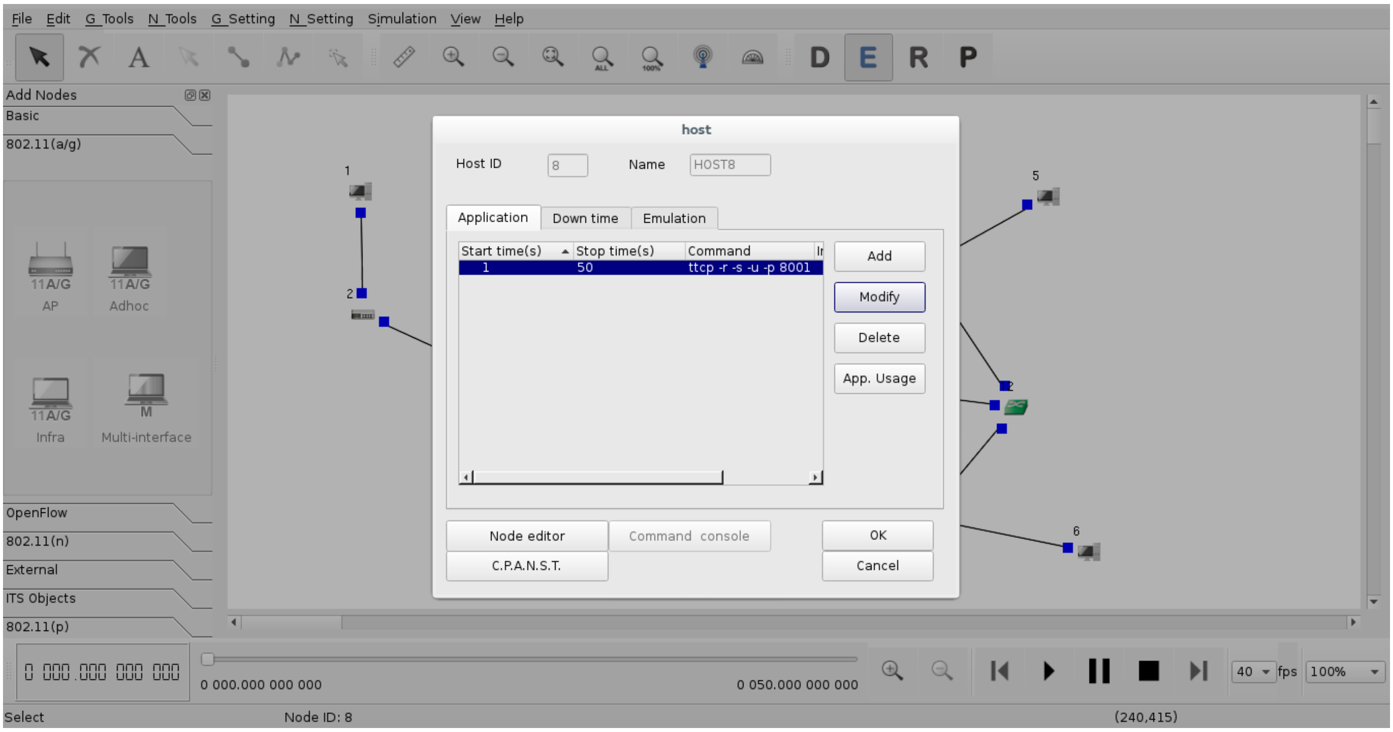


Figure 4.1.8 *Add application to node8*

If user would like to understand how to use application commands, please click “App. Usage” button to get more information.

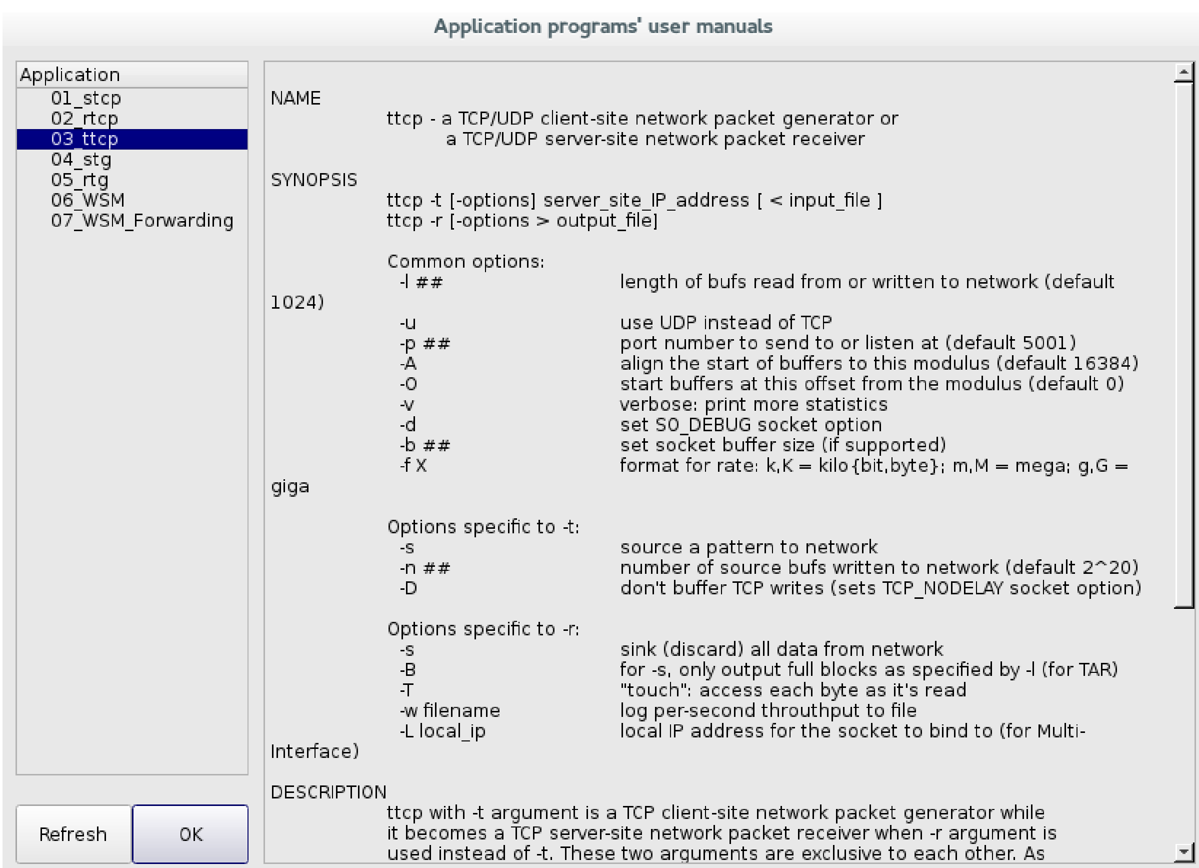


Figure 4.1.9 *Application programs' user manuals*

B. Set Hub node2 property:

Double click hub node2 then pop up the setting property dialog.

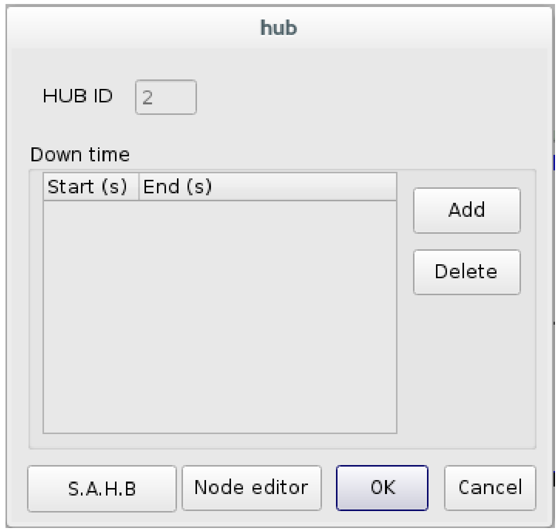


Figure 4.1.10 *The setting property dialog of hub*

Down time: "Down time" means packet can't be send or received. The down time periods specified here will be propagated and set in the PHY (or WPHY) modules of all of the node's interfaces. This is because the PHY (or WPHY) modules are the right place to disable or enable the transmission and reception of packets. In this case, the hub node is down during the 40st second to the 45th second.

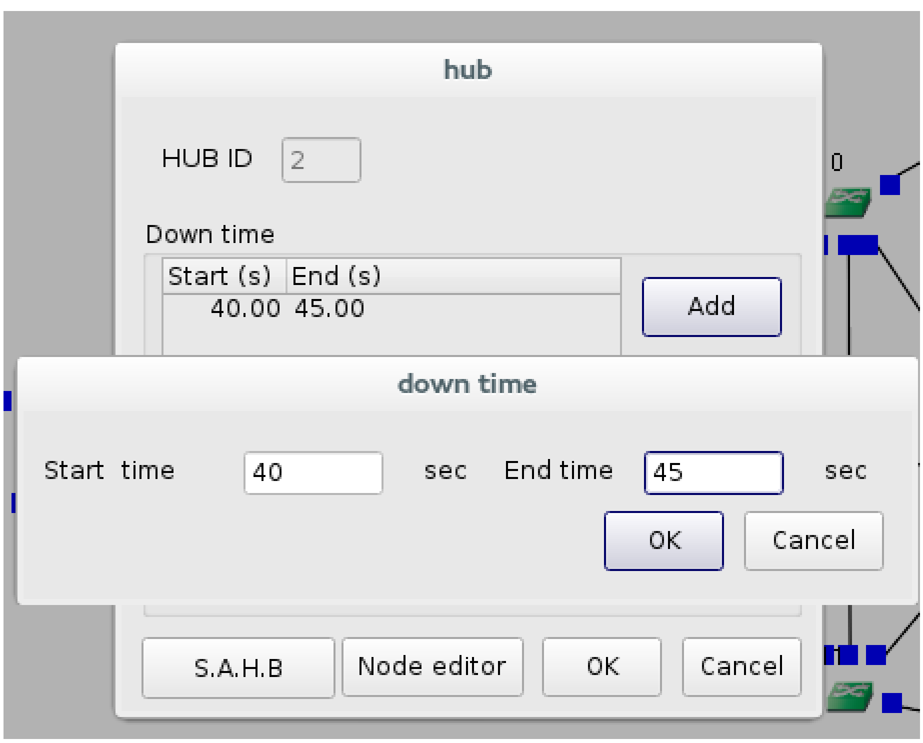


Figure 4.1.11 *The dialog of setting down time period*

"S.A.H.B": It means "Set All hub with the same bandwidth". User can use this function to set the same bandwidth which connected with hub in the same subnet. Click this button, it will pop up a setting dialog.Then input the bandwidth value. Please notice that the maximum bandwidth is100 Gbps (100000 Mbps). In this sample, using the default value: 10 Mbps.

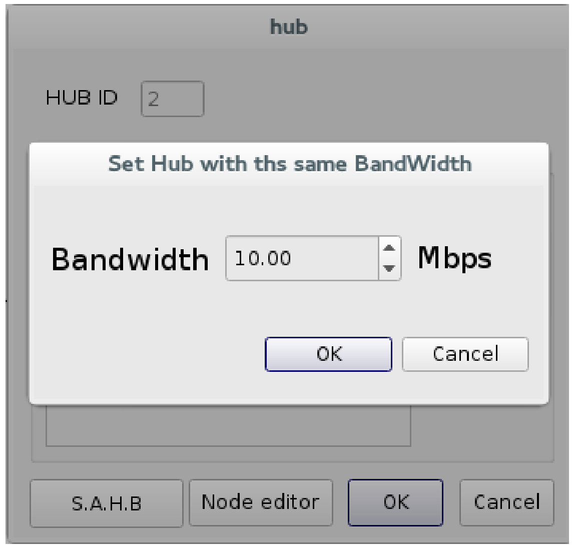


Figure 4.1.12 ***"S.A.H.B"*** *dialog*

Node editor: Click this button, it will display the protocol stack of hub. User could set attribute in Phy module.

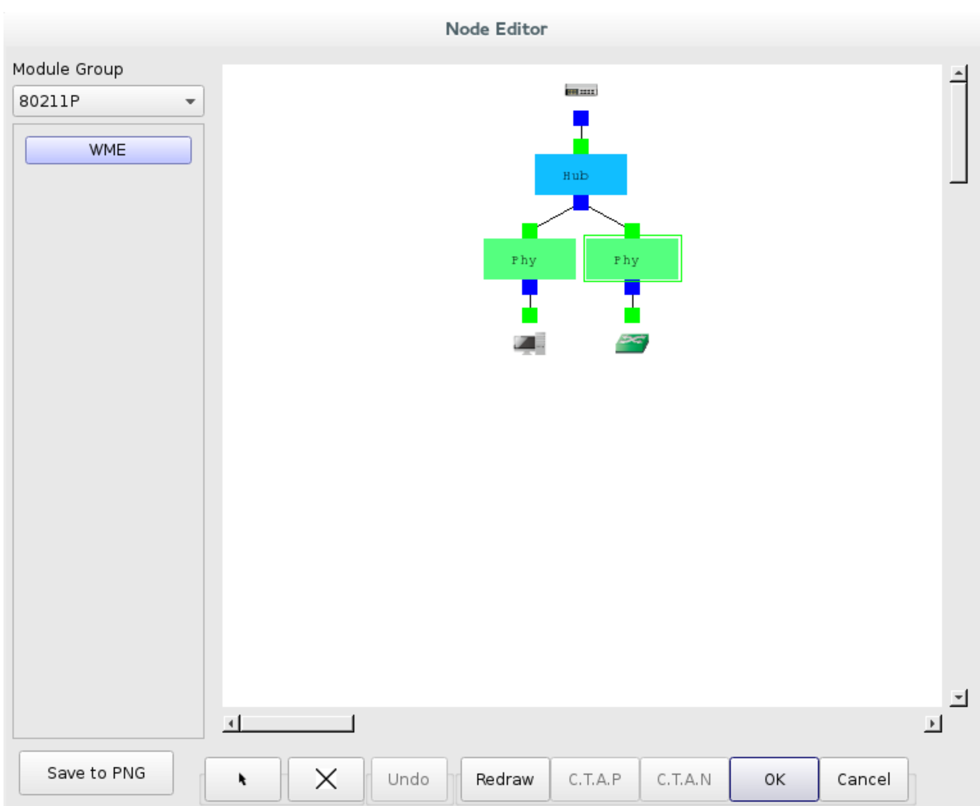
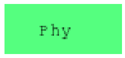


Figure 4.1.13 *The node editor dialog of hub*

Double click 

module, it will pop up a setting dialog. User can set the parameters of “Band-width”, “Bit Error Rate” (BER) and “Propagation Delay”. If user already set “Down Time”, the “Link Failure” check box will be enabled and the file name will be set automatically. To check current down time settings, user could click the “See Down Time Setting” button.

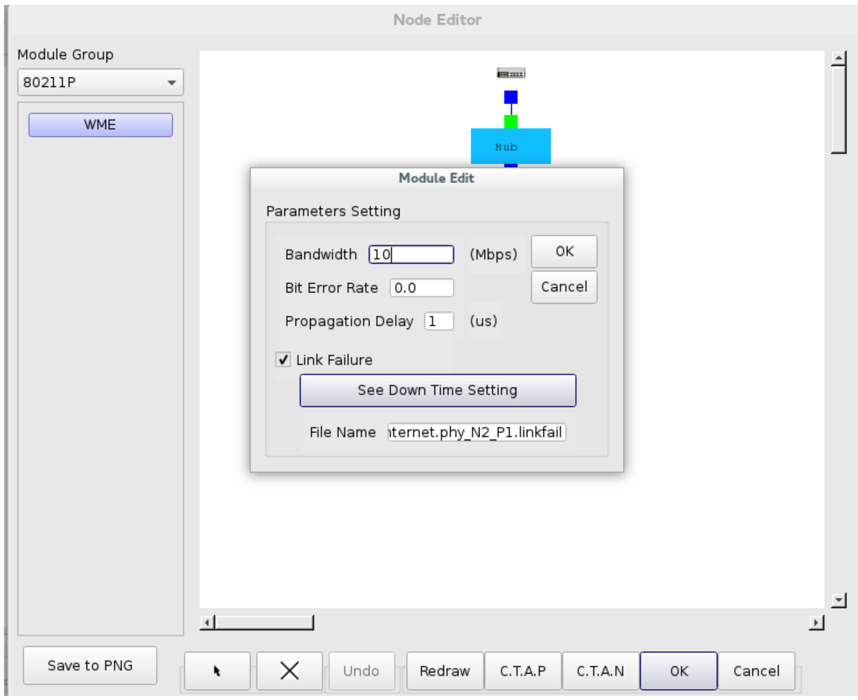


Figure 4.1.14 *The module edit dialog of phy*

C. Set Switch node7 property:

Double click Switch node7 and click “Node editor” button, it shows the protocol stack of switch.

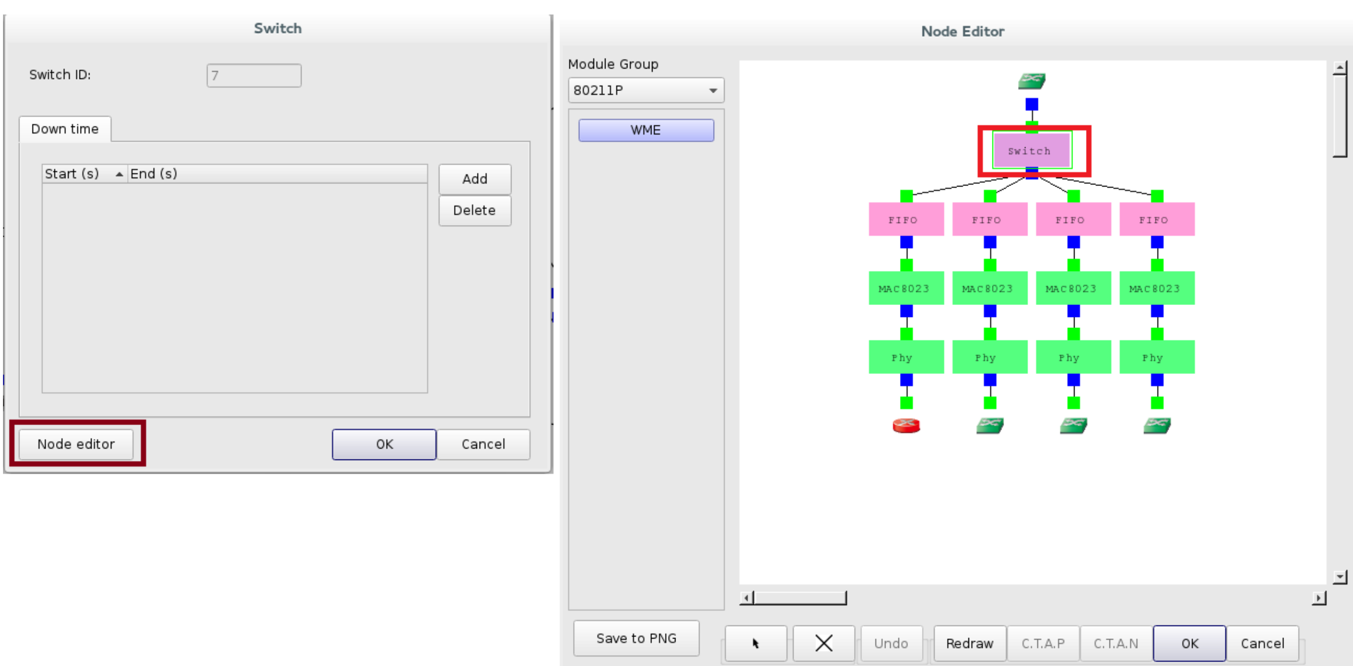
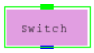


Figure 4.1.15 *The node editor diagram of switch*

Double click Switch  module then it will pop up a setting dialog. User can set the switch mode. The default mode is “Run Learning Bridge Protocol”.

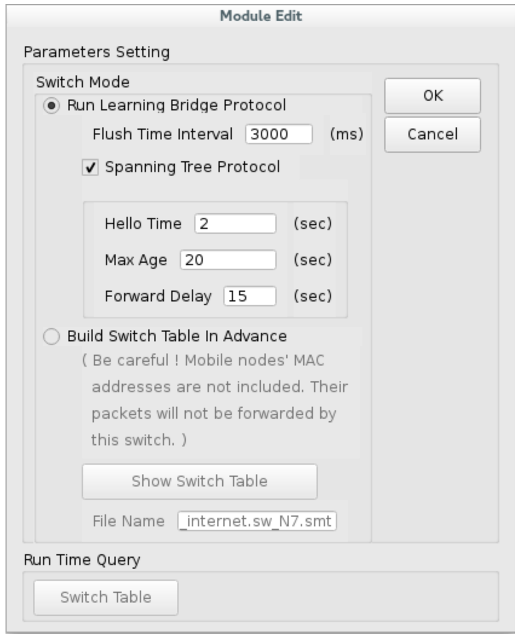


Figure 4.1.16 *The module edit dialog of switch*

In this sample, using the default mode and check “Spanning Tree Protocol” to ensure a loop-free topology.

If user want to copy the module settings to all other nodes with the same type, please use the button:”C.T.A.N” (Copy to all modules on all nodes with the same type).

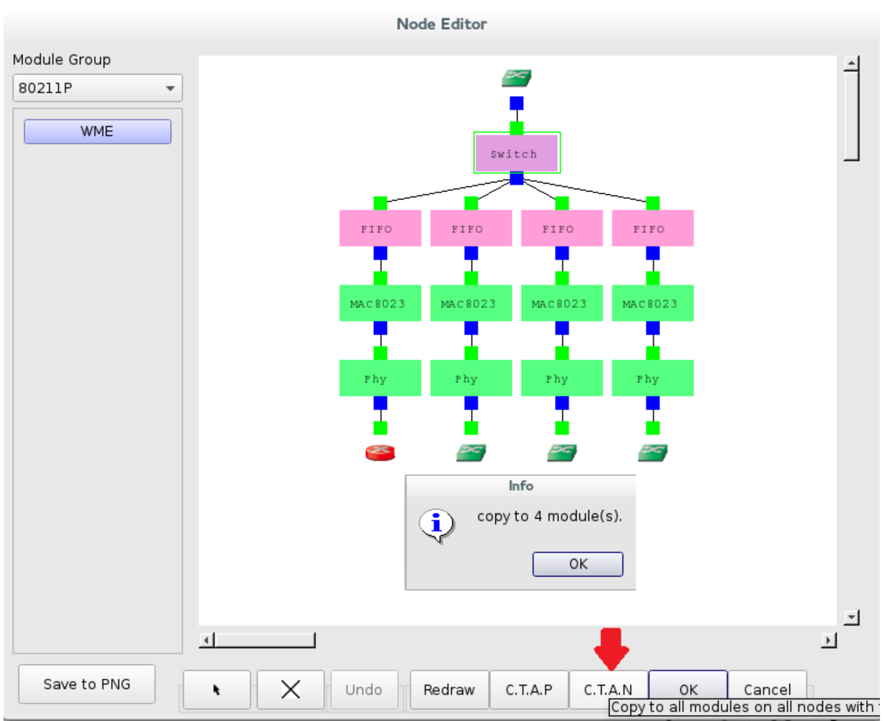


Figure 4.1.17 *Copy the settings of this switch module to other switch modules*

D. Set Router node4 property:

Double click router node4, it will pop up a setting property dialog.

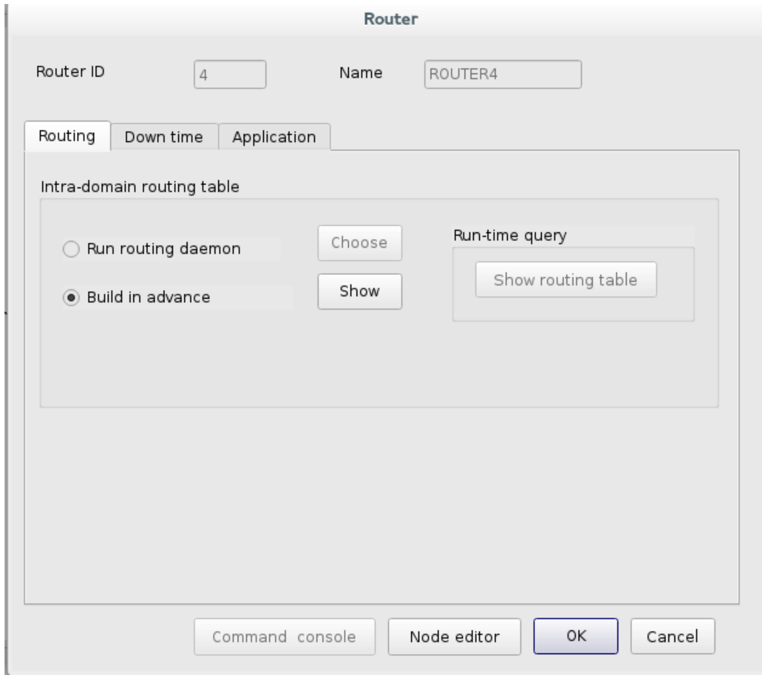


Figure 4.1.18 *The dialog of router*

In “Routing” tab, user can choose “Run routing daemon” or “Build in advance”. If user selects “Run routing daemon”, please click “Choose” button to select what kind of daemon to be used.

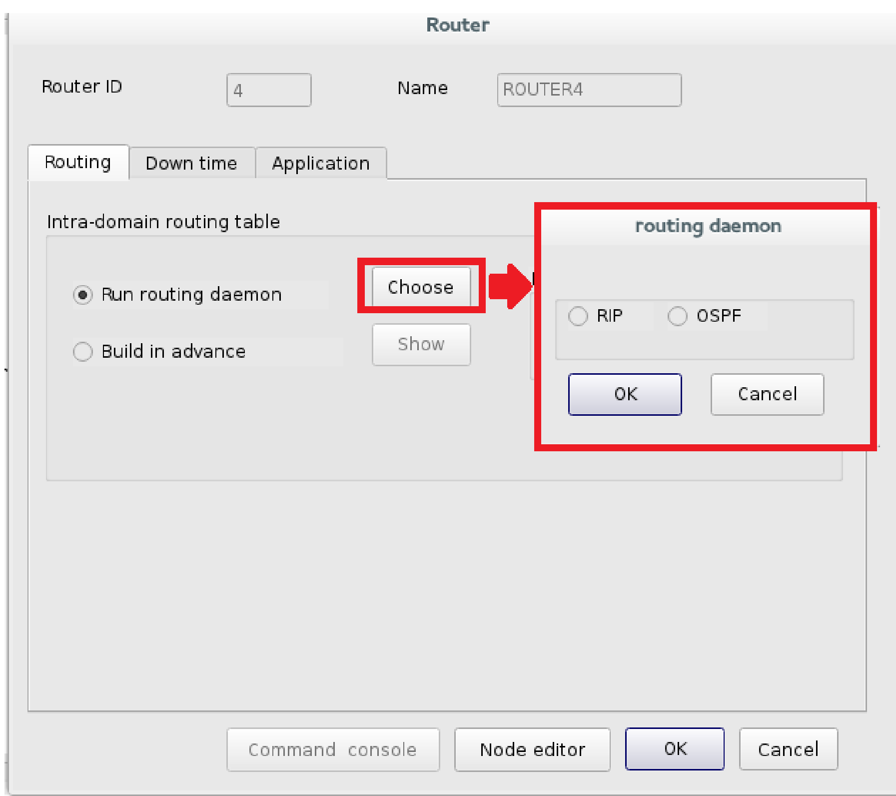


Figure 4.1.19 *To choose a kind of routing daemon*

If user select “Build in advance”, than, user could click “Show” button to see the routing table.

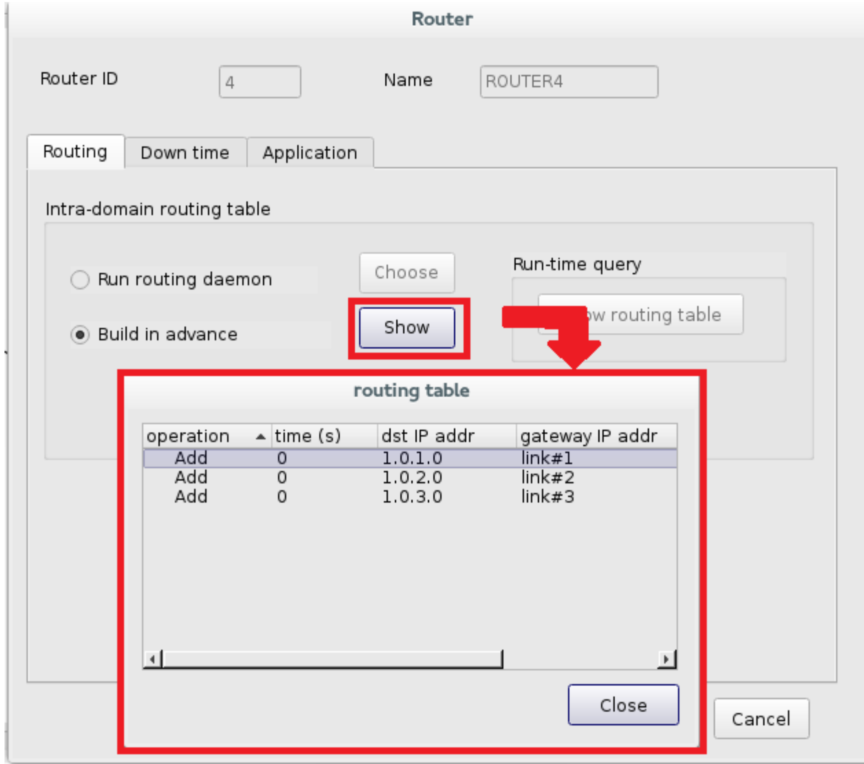


Figure 4.1.20 *To show routing table*

In this sample, using default value: “Build in advance”.

E. Set Link property:

Double click the link between host node9 to router node4, it will pop up the link property dialog box. In this dialog box, user can set the parameters of link bandwidth, signal propagation delay, bit-error-rate (BER), and down time periods for each direction of the link.

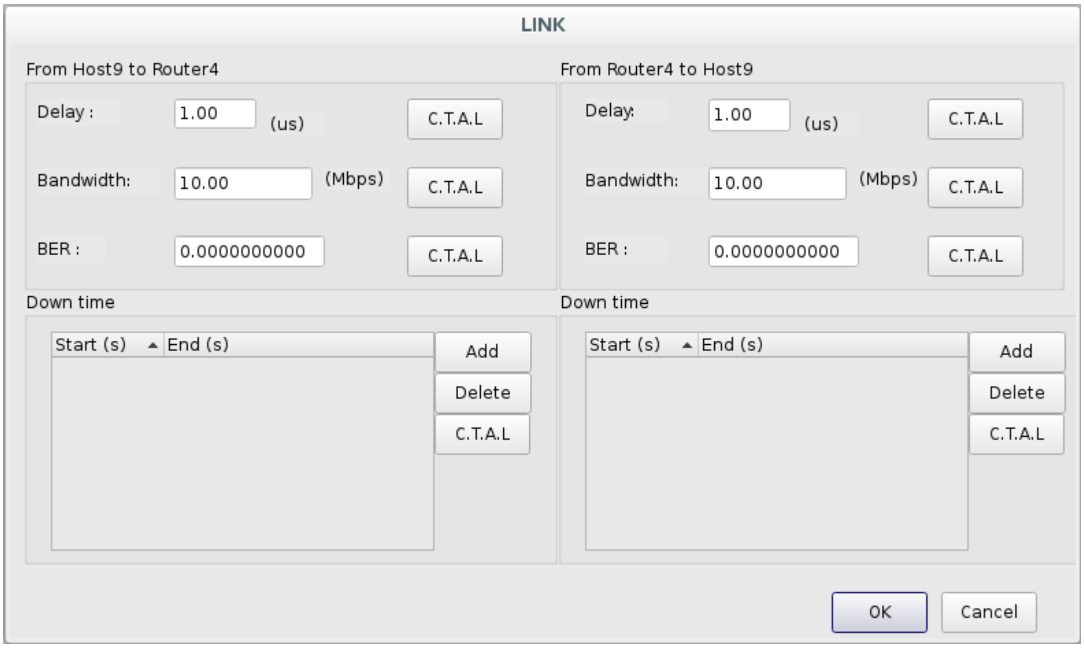


Figure 4.1.21 *The dialog of link property*

The "C.T.A.L" button stands for "Copy To All Links." Clicking this button will copy the field's current value to the same fields of all links in the simulated network. This button can save a lot of time. For example, if user want to set the bandwidth of all links to 20 Mbps, user can open up any link's property dialog box, change the bandwidth from 10 to 20 Mbps, and then click the "C.T.A.L" button next to the Bandwidth field. The bandwidth of all links in the simulated network will be changed to 20 Mbps.

Since links are bidirectional, user can separately specify the attributes and down time periods for each direction of a link. The attributes and down time periods specified for a direction of a link (say, from node A to node B) are automatically propagated to and set in the physical-layer module (e.g., PHY) of node A. In this case, setting "Bandwidth" to 15 Mbps and "Down time" between 40~45s for a direction of a link "From host node9 to router node4". User could check the property of the PHY module of the host node9 has been changed.

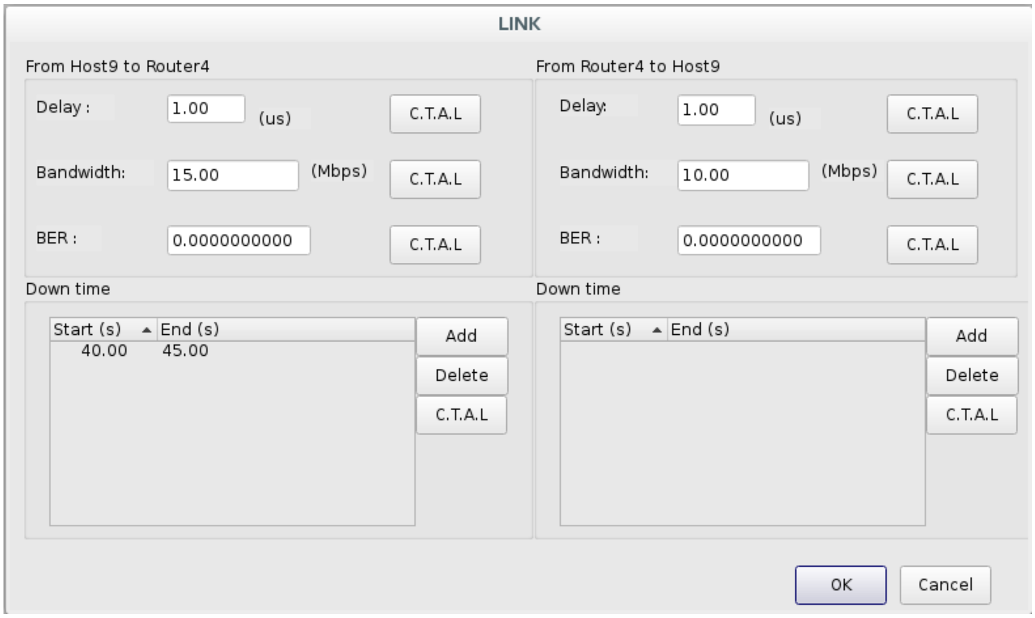


Figure 4.1.22 *To specify the attributes and down time periods "From Host9 to Router4"*

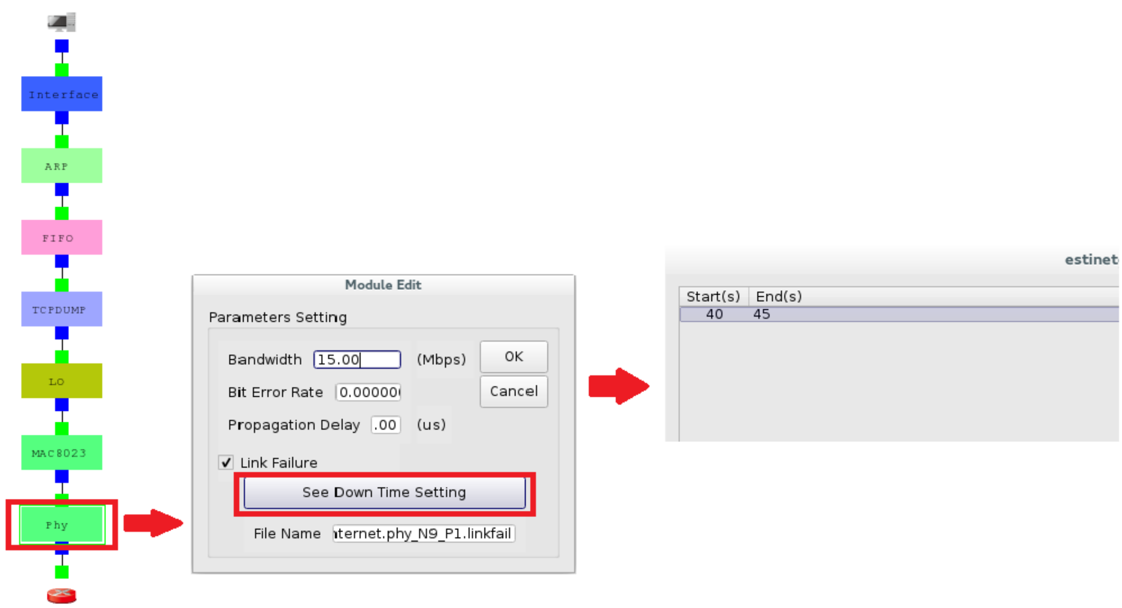


Figure 4.1.23 *The down time setting of phy module*

(3) Run Simulation

After above setting, user can switch to R mode  to start running the simulation. The command is located at **Menu → Simulation.**

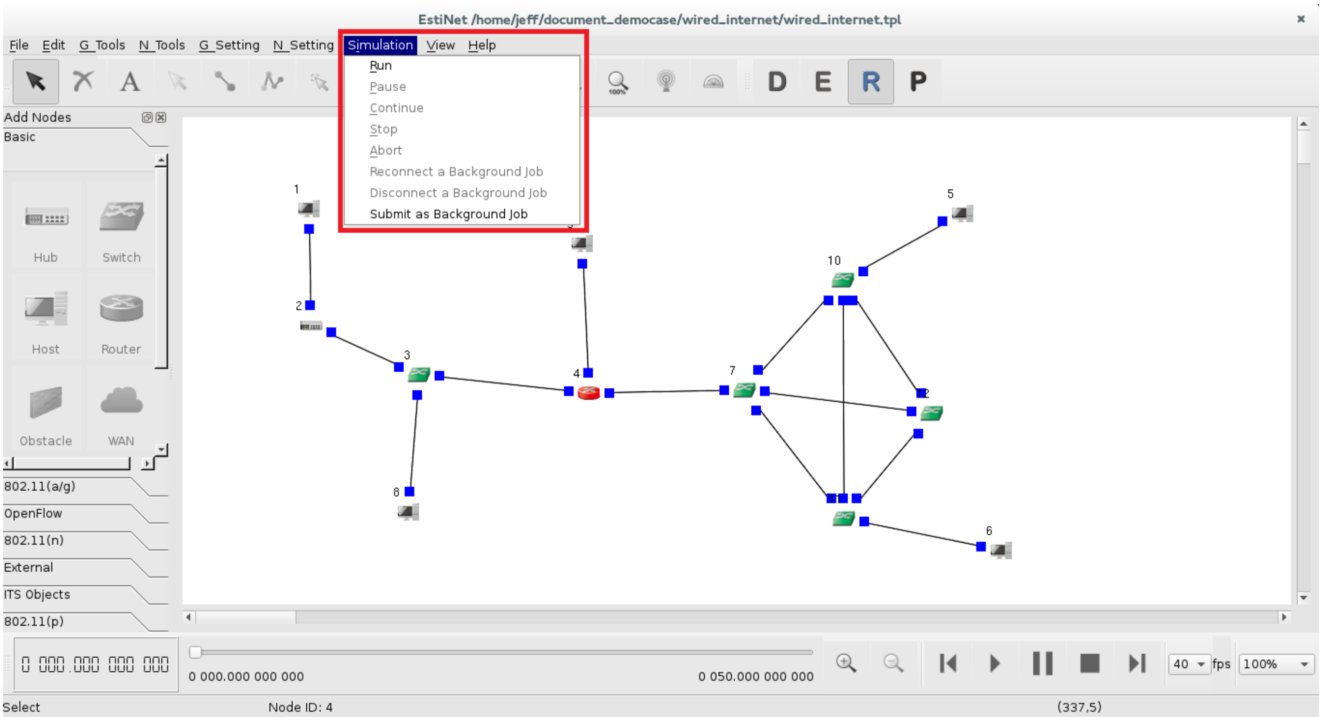


Figure 4.1.24 *Run simulation*

There are two ways to run a simulation: executed immediately "**Run**" or "**Submit as Background Job**"

Run: Choose this command to run the simulation immediately. And the other commands will be enabled.

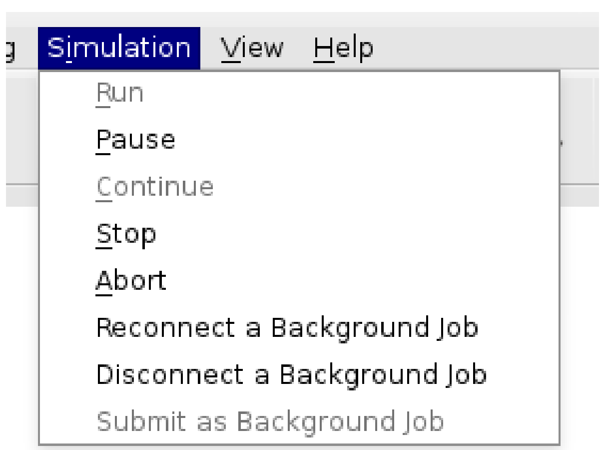


Figure 4.1.25 *Simulation Menu*

Pause: Pause the currently-running simulation

Continue: Continue the simulation that was paused

Stop: Stop the currently-running simulation

Abort: Abort the currently-running simulation. The difference between “Stop” and “Abort” is that a stopped simulation job’s partial results will be transferred back to the GUI program. However, an aborted simulation job’s partial results will not be transferred back. Instead, they will be immediately deleted on the simulation server to save disk space.

Disconnect a Background Job: Disconnect the GUI from the currently- running simulation job. The GUI can now be used to service another simulation job. A disconnected simulation job will be given a session name and stored in a session table.

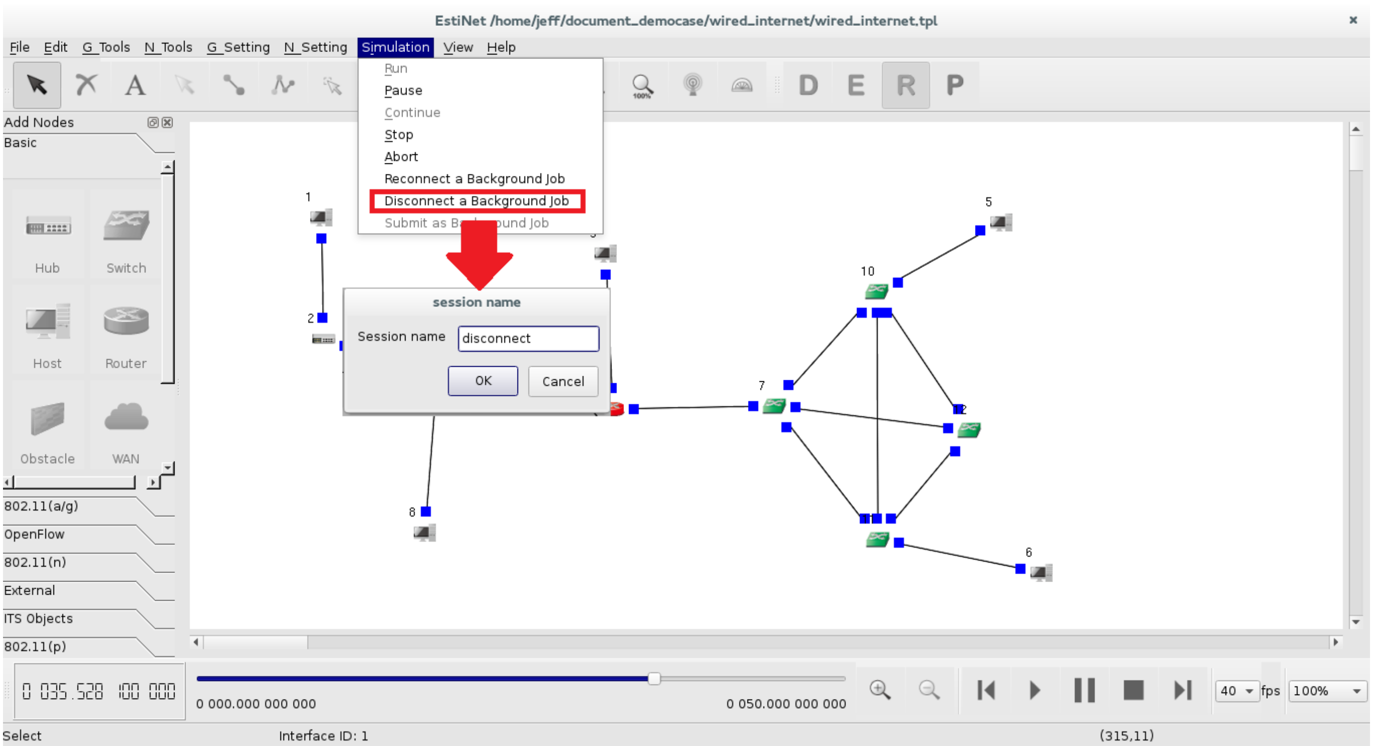


Figure 4.1.26 *Disconnect a background job*

Reconnect a Background Job: When executing the “Reconnect a Background Job” command, a user can choose a disconnected job to reconnect from this session table.

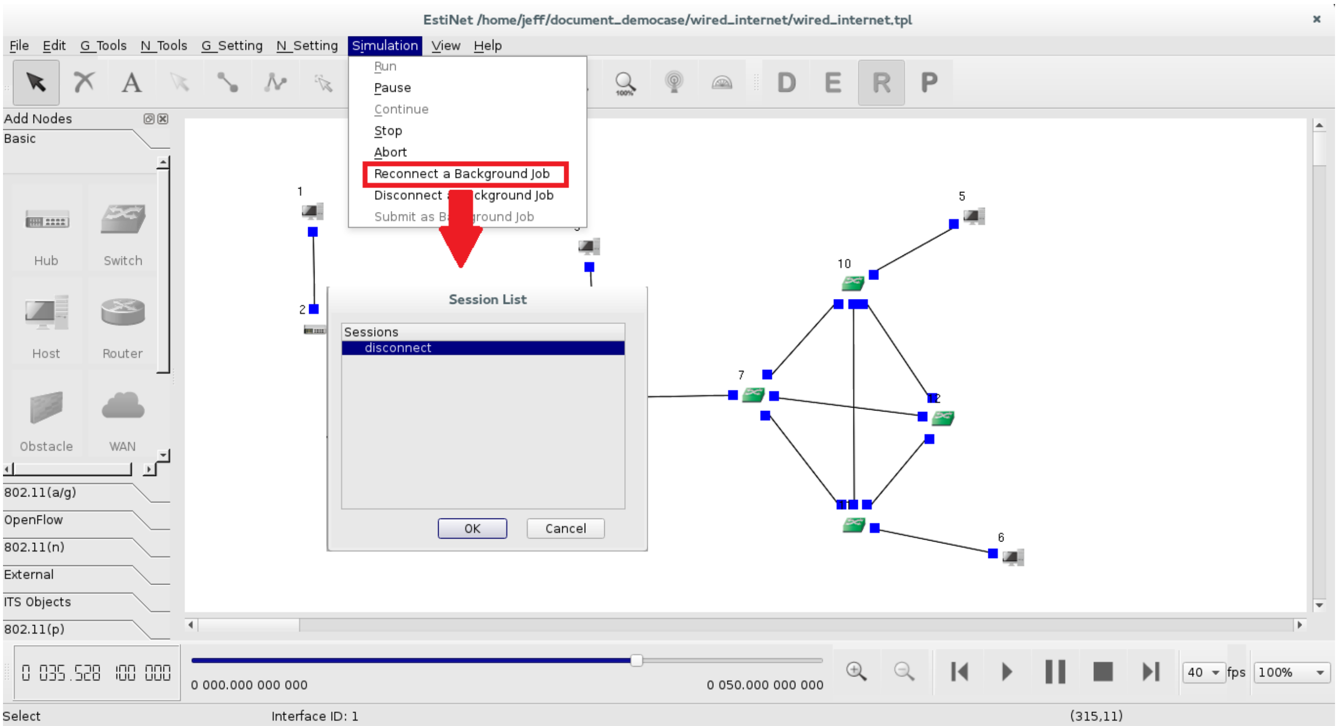


Figure 4.1.27 *Reconnect a background job*

Summit as Background job:

Choose this command to background execution the simulation. A background simulation job will be given a job name and manage by using **Menu → File → Background Job Management** command. A background job may wait in the dispatcher's job queue if currently there is no available simulation server to service this job. Whenever a simulation server becomes available, on behalf of the GUI program that submitted this background job, the dispatcher will automatically start the background job's execution on that simulation server.

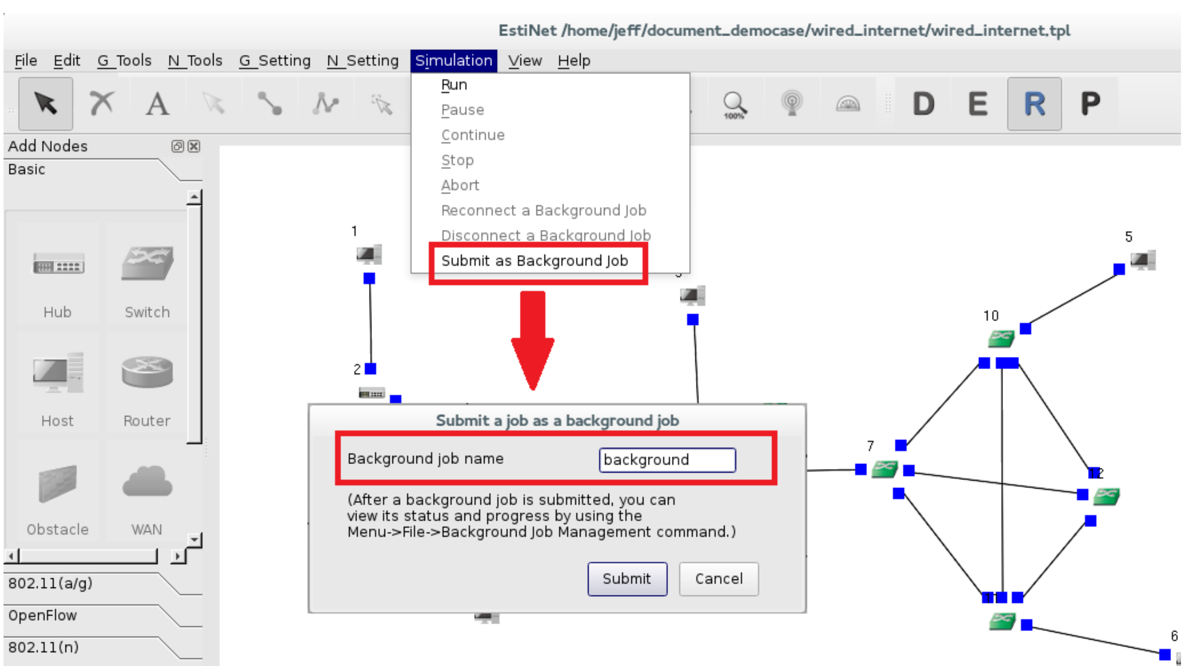


Figure 4.1.28 *Submit as background job*

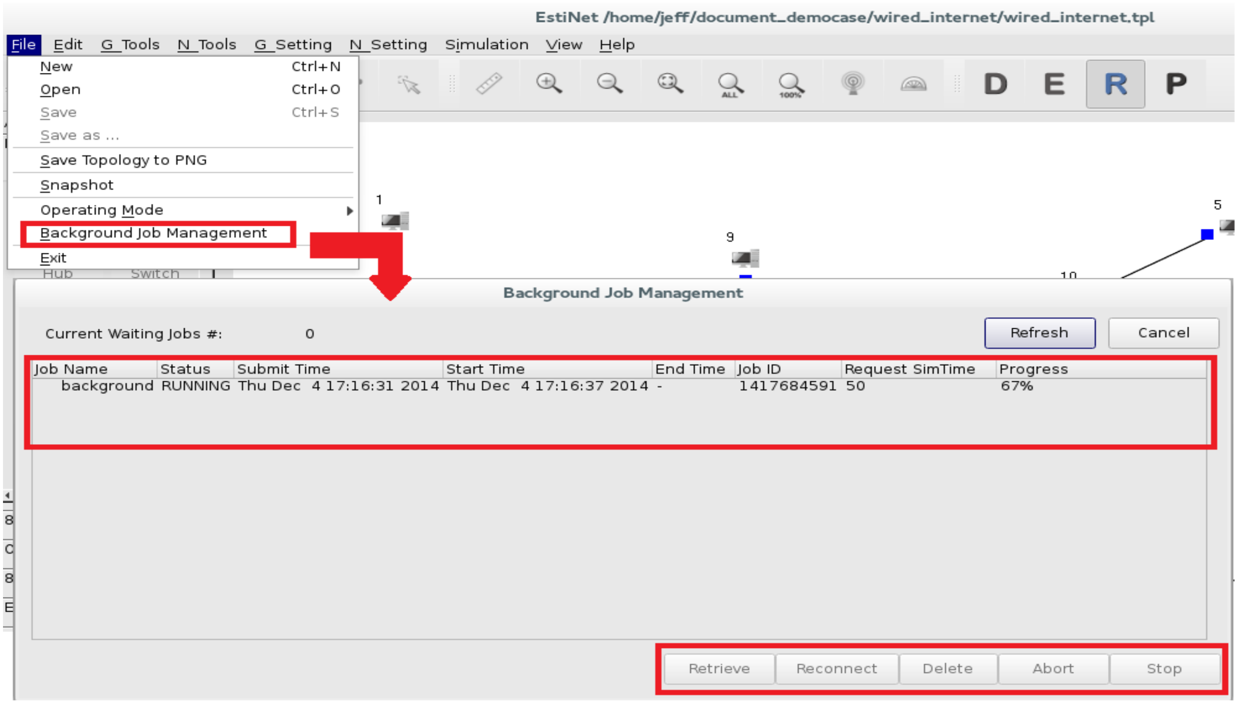


Figure 4.1.29 *Background job management*

When executing **Menu** → **File** → **Background Job Management** command, it will pop up a dialog that shows each background job's information. And you can delete, stop, abort, retrieve or reconnect it by using the bottom buttons.

(4) Playback

When simulation is done, GUI will get the simulation result automatically (for "Summit as Background job", user need to click "Retrieve" button to get the result.).

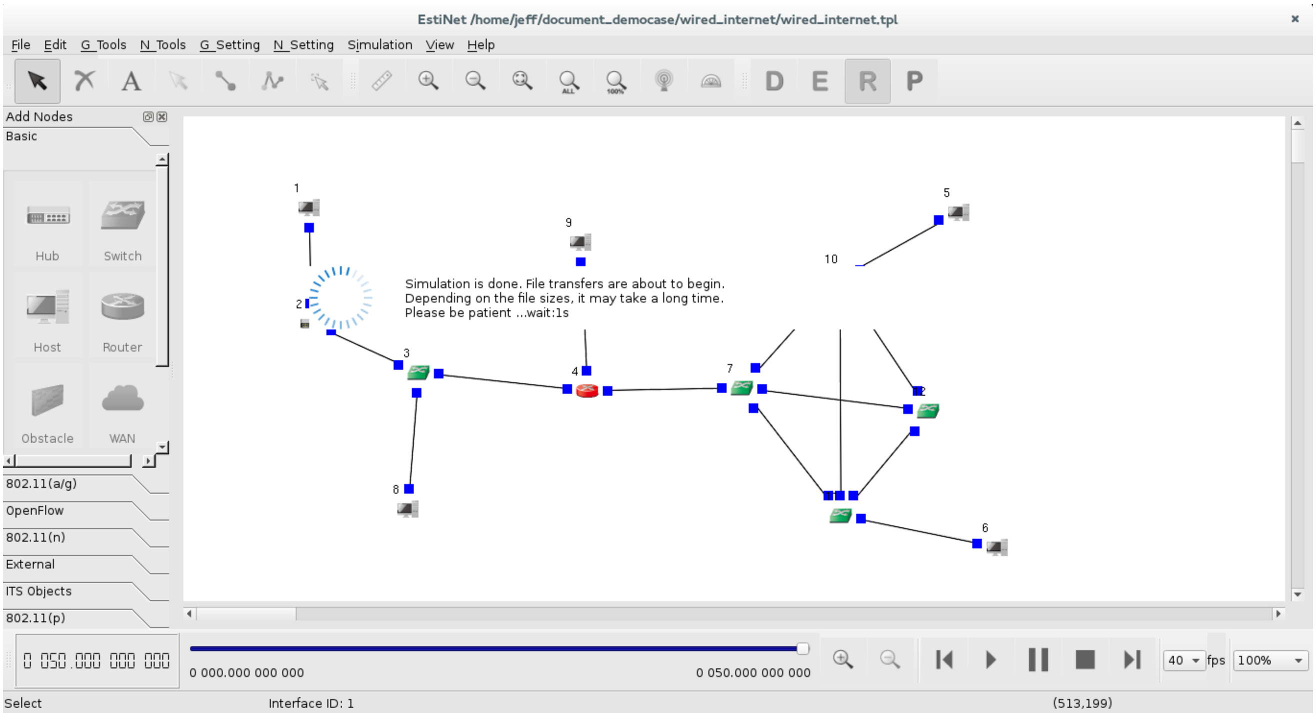


Figure 4.1.30 *Wait for getting simulation result*

GUI will switch to P mode after getting the result. User can click button to play the simulation result.

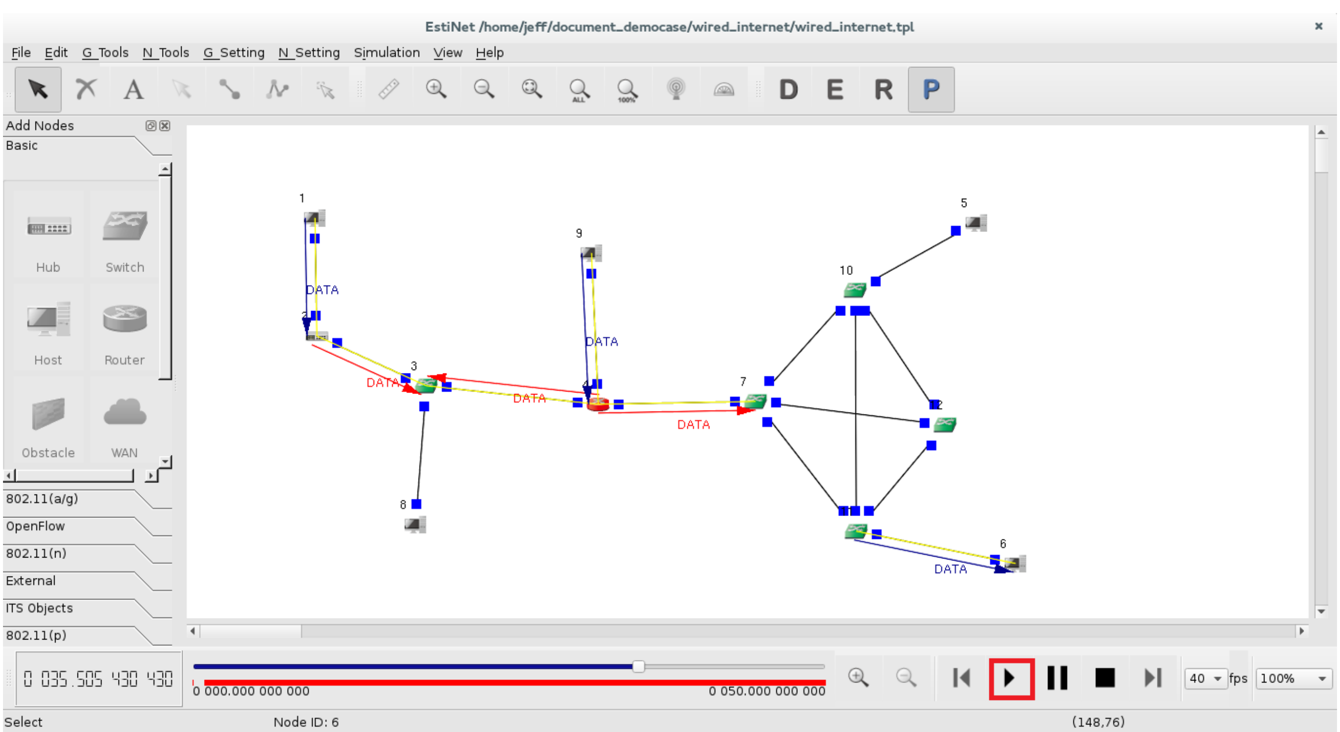


Figure 4.1.31 *Playback*

5. WLAN (IEEE 802.11)

EstiNet supports IEEE 802.11(a/g) and IEEE 802.11(n) wireless internet specifications. And both of them provide two modes: ad hoc mode and infrastructure mode. In this chapter, we will introduce these.

(1) Draw topology of wireless internet

In D mode, we use these nodes: host , router and obstacle in "Basic" group, IEEE 802.11(a/g) access point , adhoc , infra and multi-interface mobile nodes in "IEEE 802.11(a/g)" group and IEEE 802.11(n) access point , adhoc and infra mobile nodes in "IEEE 802.11(n)" group to draw a simple topology as figure 5.1.1. This topology was saved as file "wireless_internet.tpl."

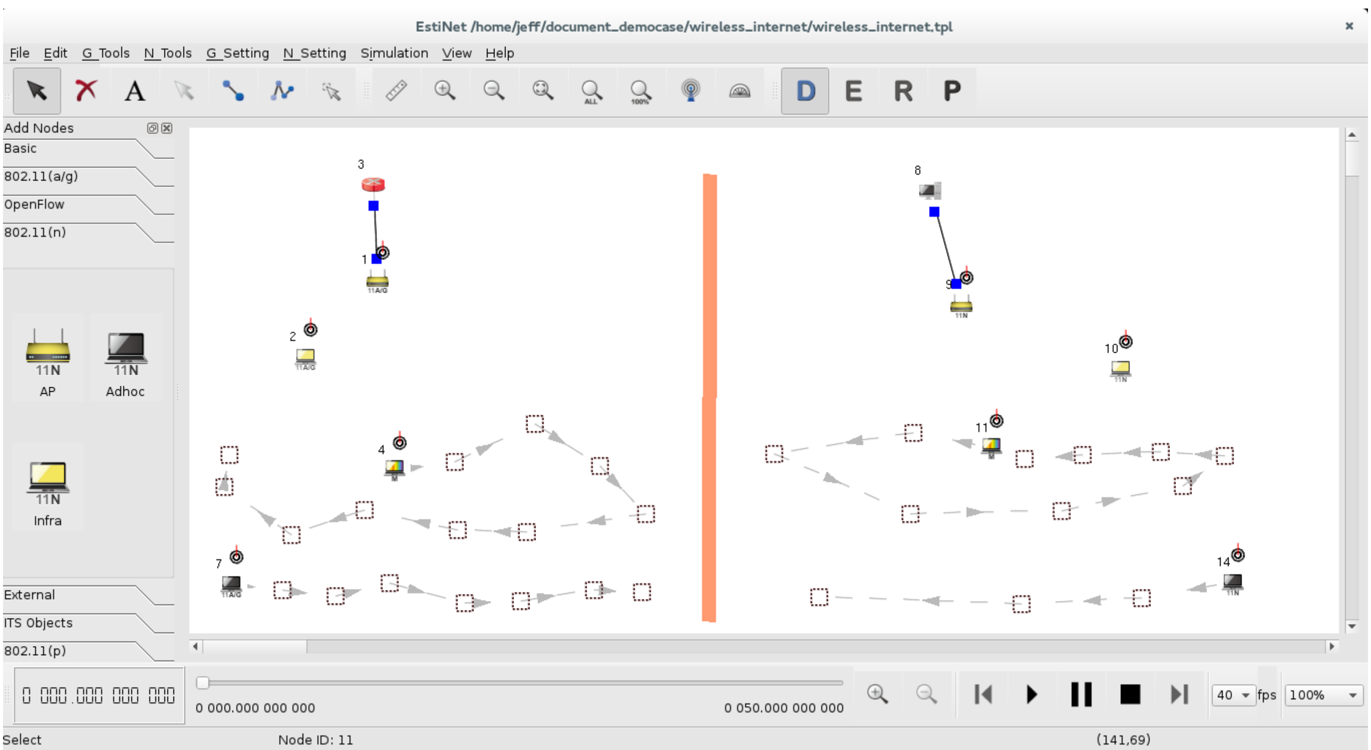


Figure 5.1.1 *The topology of WLAN*

When user add a multi-interface node, it will pop up a dialog to select how many and what kind of interfaces to be added as figure 5.1.2. There are IEEE 802.11(a/g) infrastructure mode interface, IEEE 802.11(a/g) ad hoc mode interface, IEEE 802.11(n) infrastructure mode interface, IEEE 802.11(n) ad hoc mode interface and IEEE 802.11(p) OBU interface in this multi-interfaces list. In this sample, node4 was set by one IEEE 802.11(a/g) infrastructure mode interface with one IEEE 802.11(a/g) ad hoc mode. And node 11 was set by one IEEE 802.11(n) infrastructure mode interface with one IEEE 802.11(n) ad hoc mode interface.

If user would like to add or delete any multi-interface in a mobile nodes, please double click on the multi-interface mobile node. Then click “Add” or “Delete” button to modify settings.

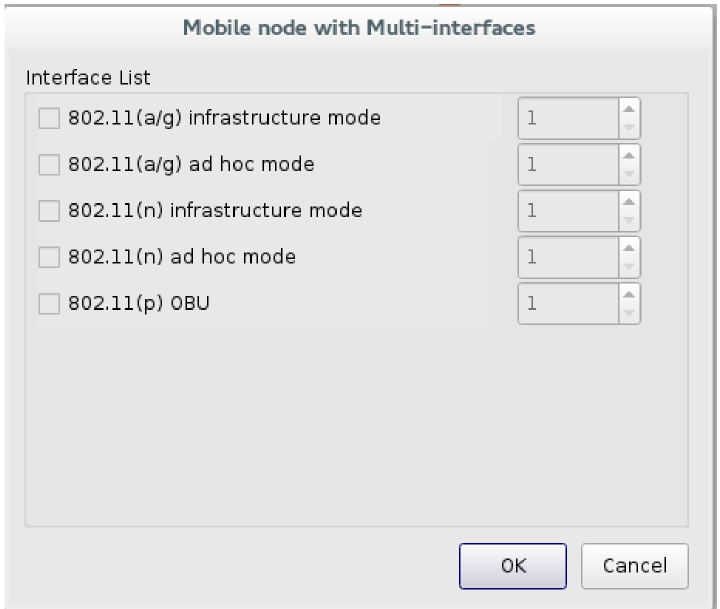


Figure 5.1.2 *Add interface in multi-interface mobile node*

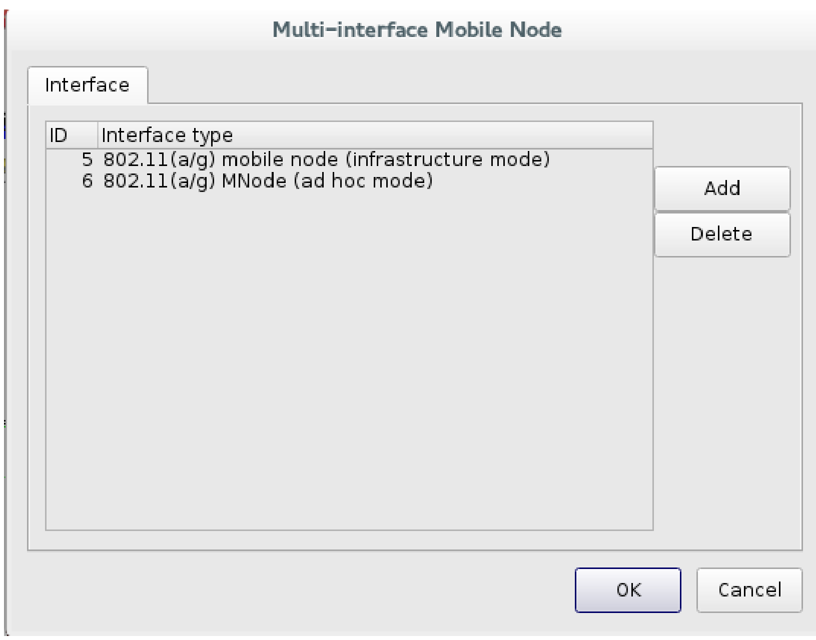


Figure 5.1.3 *Add or delete interface in multi-interface mobile node*

In a real-world network environment, a mobile node could move anywhere. If user want to add the moving path of these mobile nodes: node4, node7, node11 and node14. Firstly, click the moving path tool icon  on the tool bar. Then click the mobile nodes and dragging clicking the left button of mouse repeatedly to construct the whole moving path. Click the right button of mouse to stop setting moving path. In this sample, the moving paths of node4, 7, 11 and 14 are shown as figure 5.1.1.

In this sample, purposely to add obstacles to the open field which want to block/attenuate wireless signal at some places. Under EstiNet simulator, an obstacle is a rectangle which could block a mobile node's view, block a mobile node's movement, block wireless signal completely or just attenuate the power of wireless signal.

After drawing the obstacle, double-clicking on it. It will pop up a dialog to set the properties of the obstacle as figure 5.1.4. The default values for these properties can be set by executing the **Menu → G_Setting → Obstacle**. In this sample, using the obstacle to separate two areas wireless signal. Selecting "Block wireless signal" with "Block signal completely" to block the passing wireless signal completely with 15 pixels width. If user select "Block wireless signal" with "Attenuate by x dmb", the signal will be blocked by the amount strength specified. The other two selections "Block node view" and "Block node movement" are used in VANET.

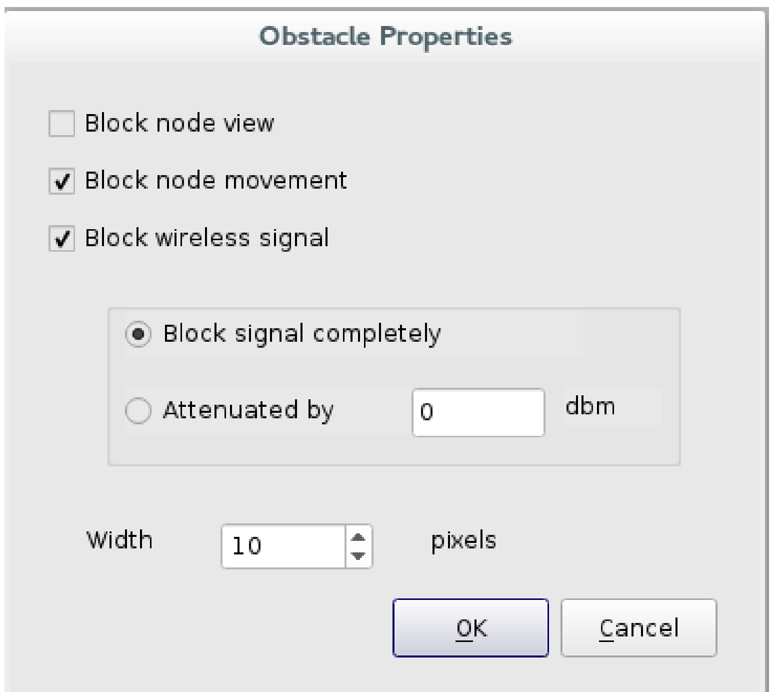


Figure 5.1.4 *The dialog of obstacle properties*

Before switch to E mode E , user need to use the "Form wireless subnet" ( ) tool to set which infrastructure mobile nodes belong to the same subnet. Because infrastructure mobile nodes need to use an access point to forward their packet, but GUI program does not have the intelligence to get which infrastructure mobile nodes and which access points should belong to the same subnet. So, user must gather the nodes of wireless mobile with their access point to form a subset. To do this, firstly, user selects this tool "Form wireless subnet" ( ) and then uses the left button of mouse to sequentially click all required nodes. When all required nodes have been selected, the user clicks the right button of mouse to end the selection process. In this sample, user selects node1 (IEEE 802.11(a/g) access point), node2 (infrastructure mobile) and node4 (the infrastructure interface of the multi-interface) to form a subnet. And user also selects node9 (IEEE 802.11(n) access point), node10 (infrastructure mobile) and node11 (the infrastructure interface of the multi-interface) to form another subnet. When user selects the multi-interface mobile nodes, it will pop up a dialog to ask which interface would be added to form a subnet.

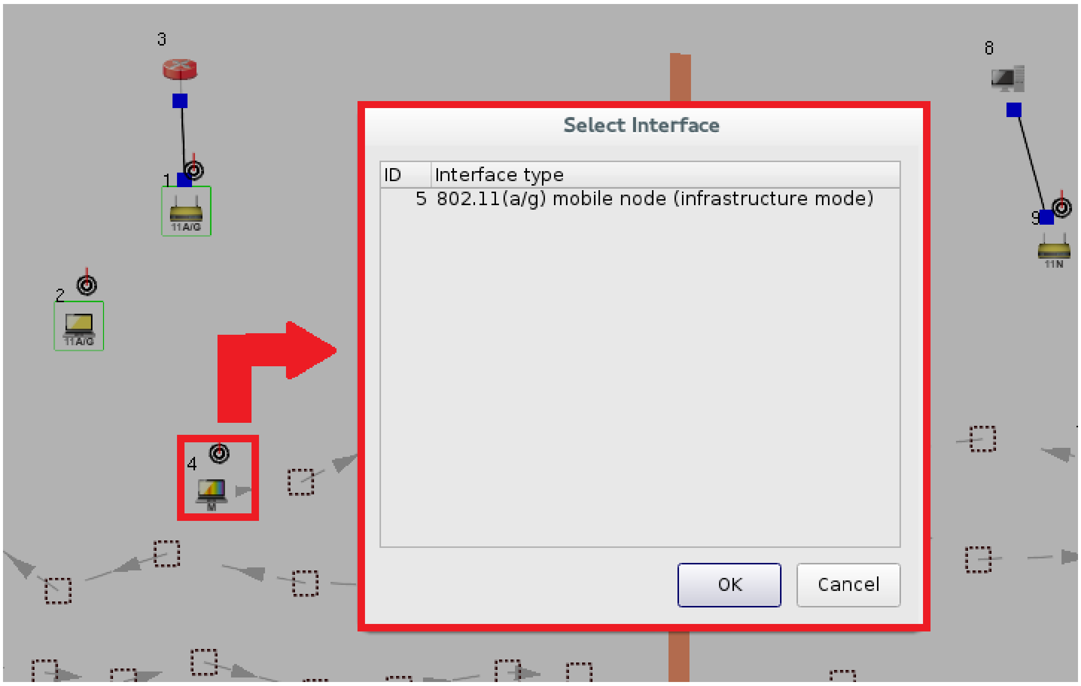


Figure 5.1.5 *Select interface of multi-interface mobile node4*

After clicking the right button of mouse to end the selection process, it will show the confirm dialog. Please click “OK” button, otherwise, click “Cancel” button to select again.

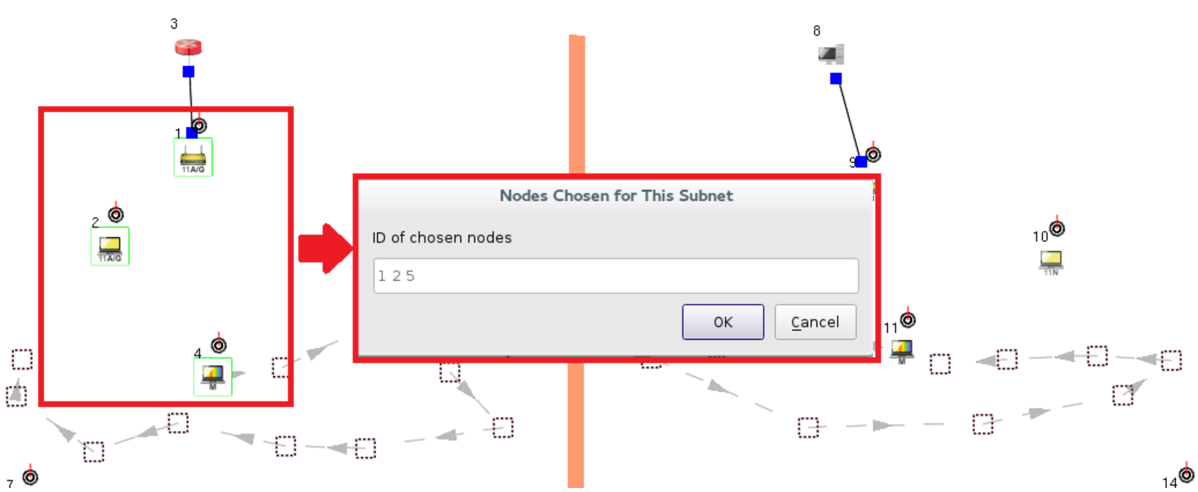


Figure 5.1.6 *Select node1 IEEE 802.11(a/g) AP and node2, node5 (include in node4) infrastructure mode to form a subnet*

After forming a subnet, if user want to check the subnet information, please execute “Menu → N_Tools → 802.11(a/g) Wireless Network→ Manage 802.11(a/g) Infrastructure Mode Subnets” and “Menu →N_Tools → 802.11(n) Wireless Network→ Manage 802.11(n) Infrastructure Mode Subnets”, and it will show the information dialog to check out these subnets. User could use “Delete Subnet” button to delete the subnet and do the subnet selection process again.

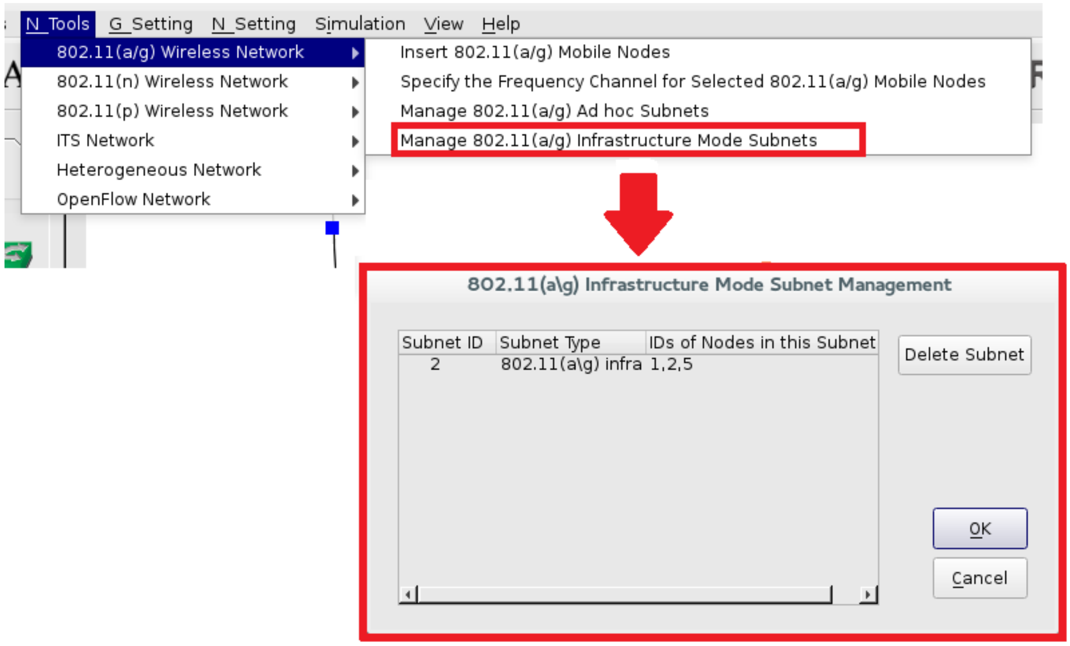


Figure 5.1.7 *Select IEEE 802.11(a/g) AP and infrastructure mode to form a subnet*

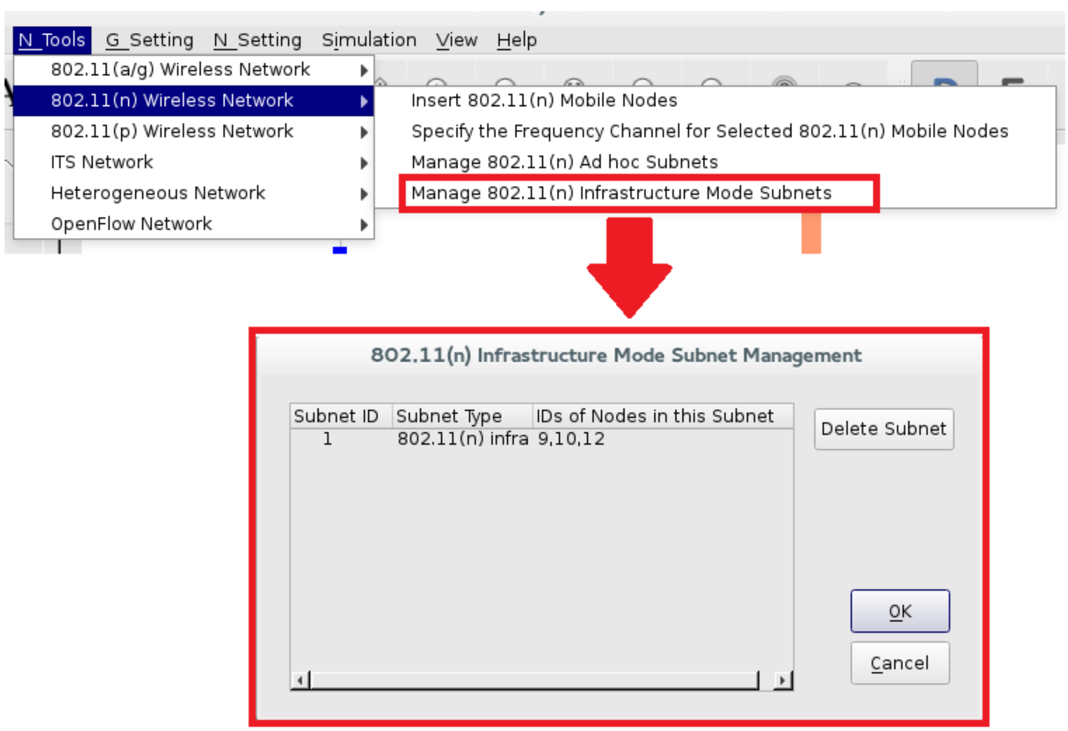


Figure 5.1.8 *Select IEEE 802.11(n) AP and infrastructure mode to form a subnet*

(2) Setup Properties

Switch to E mode E and GUI automatically subdivides various subnets and assigns IP and MAC addresses automatically as figure 5.1.9. To check IP address of a mobile node is moving the mouse to wireless signal propagation waves then GUI will show the mobile node's IP and subnet information. To check MAC address of a mobile node is the same as wired internet as figure 4.1.3.

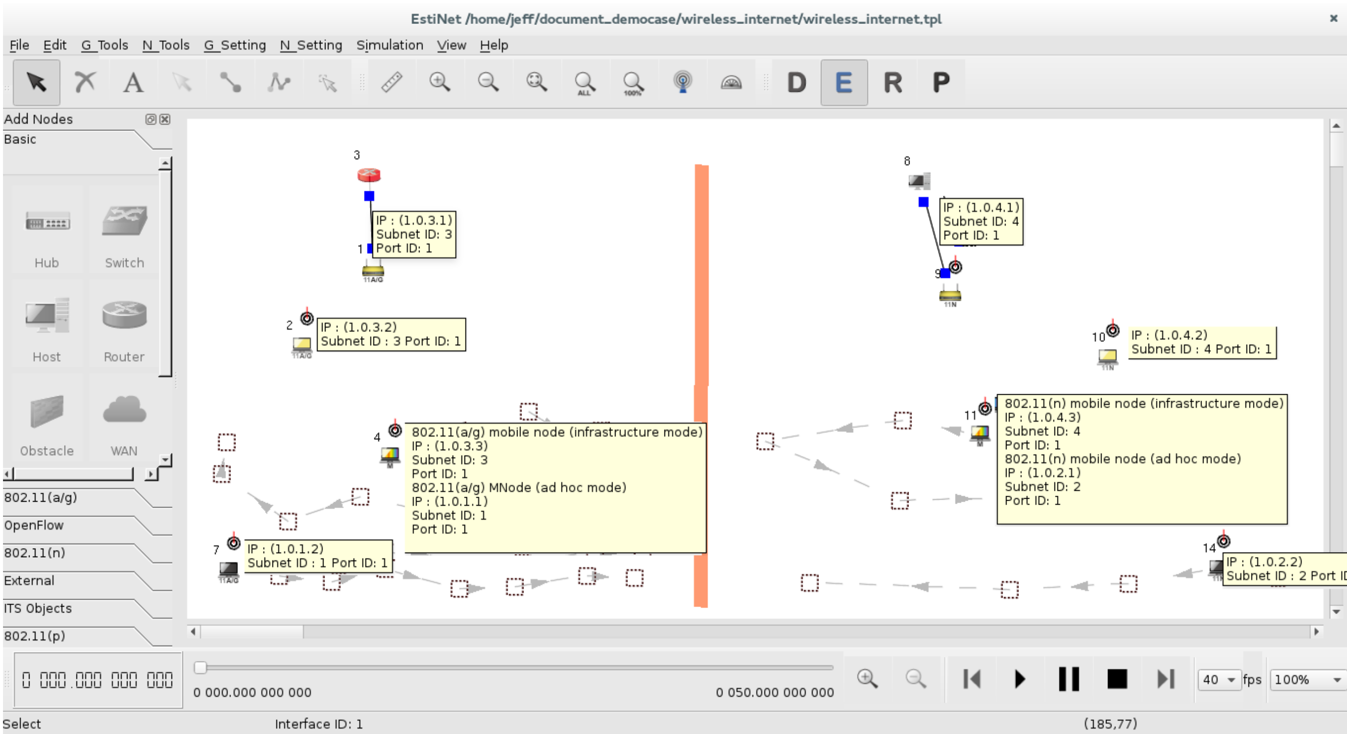


Figure 5.1.9 *The nodes information of IP address and subnet*

Set Network Traffic:

Using command “ttcp” to generate UDP and TCP packets in this sample. Node2 (IEEE 802.11(a/g) infrastructure mobile) and node11 (IEEE 802.11(n) infrastructure interface of multi-interface mobile) are expected as senders. Node2 sends UDP packets to node4 (IEEE 802.11(a/g) infrastructure interface of multi-interface mobile). At the same time, node11 (802.11(n) infrastructure interface) sends TCP packets to node10 (IEEE 802.11(n) infrastructure mobile).

Besides, using command “stcp” to generate TCP packets. Node4 (IEEE 802.11(a/g) adhoc interface of multi-interface mobile) and node14 (IEEE 802.11(n) adhoc mobile) are expected as senders. Node4 (IEEE 802.11(a/g) adhoc interface) sends TCP packets to node7 (IEEE 802.11(a/g) adhoc mobile). Meanwhile, node14 sends TCP packets to node11 (IEEE 802.11(n) adhoc interface of multi-interface mobile).

Double click node2 and add the command “ttcp -t -s -u -p 8000 1.0.3.3” in the “Application” tab as figure 5.1.10. The command “-t” means transmission, “-s” is the source, “-u” means UDP type, “-p port number” means port number, and with the destination IP address 1.0.3.3.

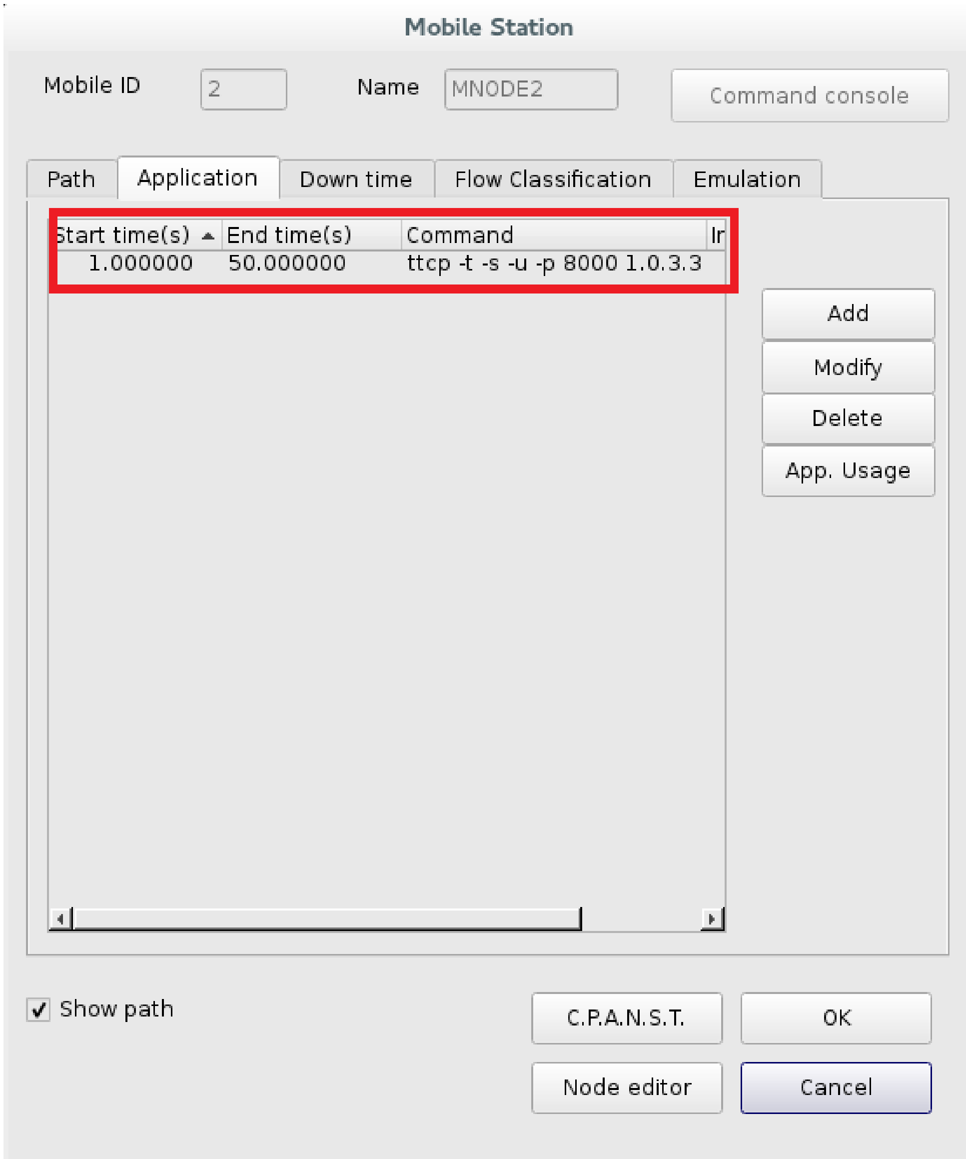
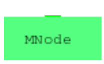


Figure 5.1.10 *Add application ttcp in node2*

i. Node2: Click “Node editor” button to see the protocol stack of node2 as figure 5.1.11.

Double click 

module and show the “Module Edit” dialog. It supports 25 channels for user to select “Channel Number”. The default channel number is 36 as figure 5.1.12.

Double click 

module and show the “Module Edit” dialog. There are two operation modes in mobile node. One is “Ad Hoc Mode” and the other is “Infrastructure Mode”. Either “Ad Hoc Mode” or “Infrastructure Mode” must be selected. When user selects “Infrastructure Mode” with “Passive Scan” (default setting), the mobile node will sequentially scan all channels (up to 25 channels in EstiNet IEEE 802.11 (a/g) infrastructure mode).Then the mobile mode will decide which access points is the best selection by listen beacons of each access points. Because it spends 0.3 second per channel for passive scan, the total scan time would be 0.3 * 25 = 7.5 seconds. When user would like to select “Infrastructure Mode” with “Active Scan”, the mobile node will send probe packet to search access point. Because it spends 0.1 second per channel for active scan, the total scan time would be 0.3 * 25 = 2.5 seconds. The total scan time could explains why the traffic for infrastructure mobile nodes should be start after 7.5 or 2.5 seconds. When user enabled option "Only Scan One Channel", the mobile module will only scan one channel which specify in the PHY module PHY80211a.

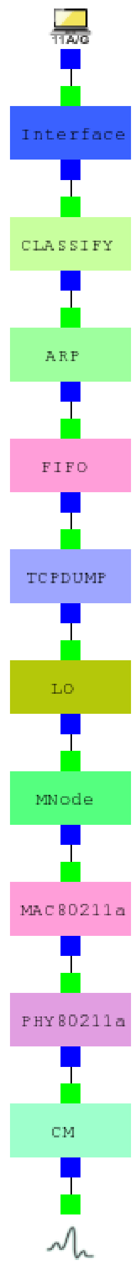


Figure 5.1.11 *The protocol stack of IEEE 802.11(a/g) infrastructure mode node2*

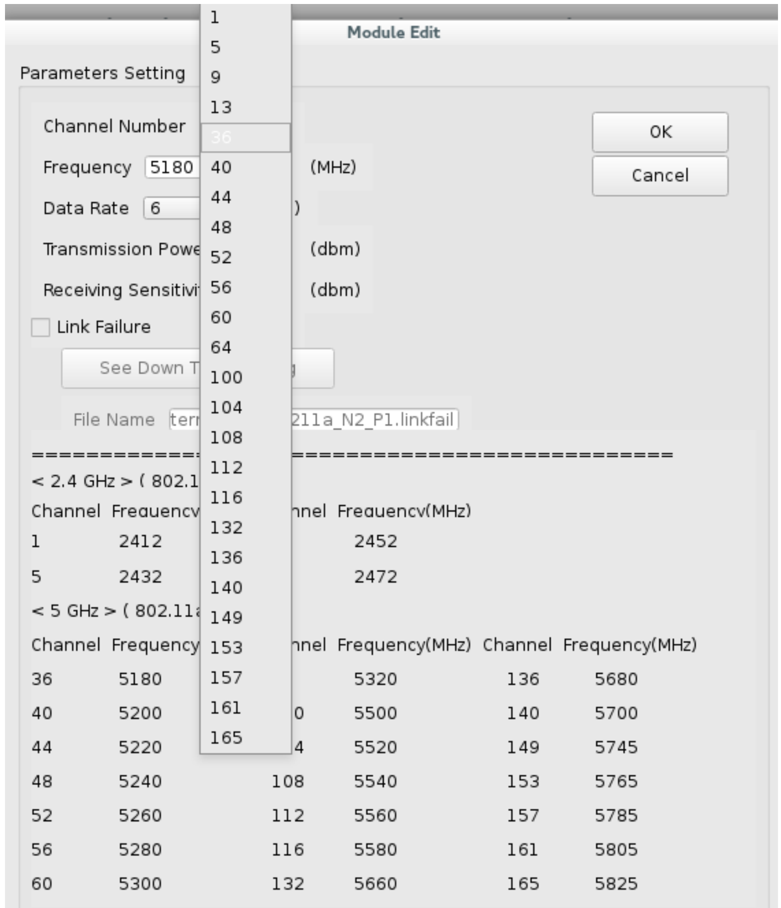


Figure 5.1.12 *Support 25 channels for wireless mobile node*

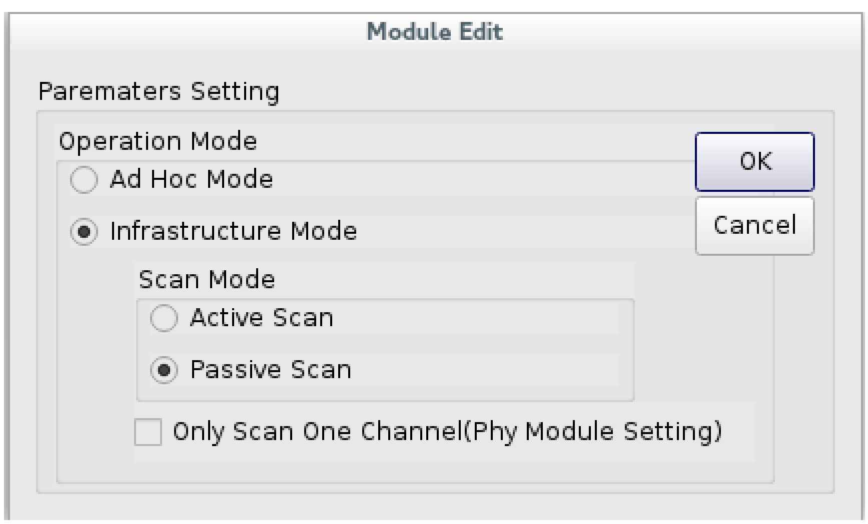
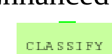


Figure 5.1.13 *The properties of MNode module*

Both IEEE 802.11(a/g) and IEEE 802.11(n) support EDCA (Enhanced Distributed Coordination Access). If user want to enable EDCA, just double click module 

and check "Enable Classification" as figure 5.1.14.

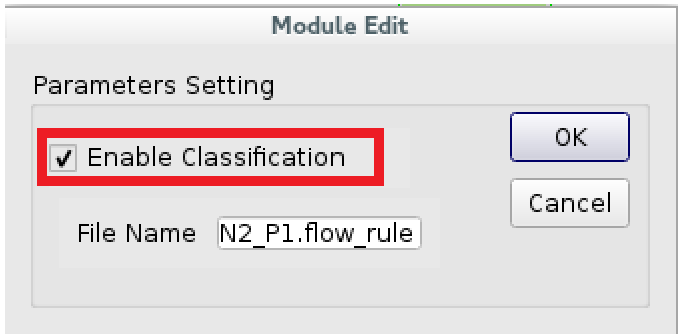
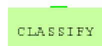


Figure 5.1.14 *Enable Classification in module CLASSIFY*

Then, return to the property dialog of node2 and add flow classification in tab “Flow Classification”. Setting flow rule to decide the packet which belong to what kind of user priority (0~7). When the packet through the CLASSIFY module  , it will match the packet with every flow rule. The match items of current flow rule are source IP, destination IP, source port, destination port, and protocol type of the packet. If the flow rule matched, the packet will be added the user priority information. Actually, the user priority information will attach with the packet. And the MAC module will use this information to execute EDCA mechanism. If none flow classification is added, it means there is no flow rules. The default user priority value of all packets are 0. In this sample, using default setting.

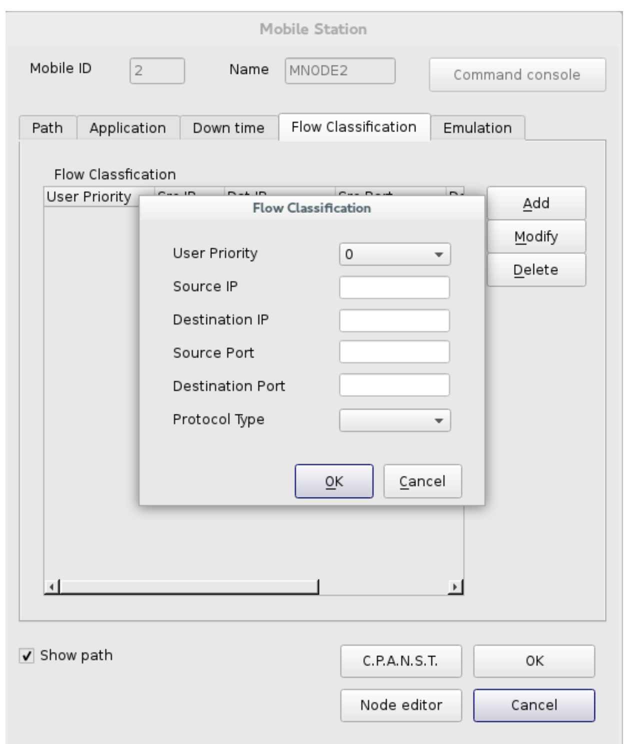


Figure 5.1.15 *Add flow classification to enable EDCA*

ii. Node4: Double click multi-interface mobile node4 and choose tab "Interface". There are two network interfaces such as figure 5.1.16. Select interface ID 5: IEEE 802.11(a/g) mobile node (infrastructure mode) with click “Edit” button, then add a command line: “ttcp -r -s -u -p 8000” which receive UDP packets from node2 as figure 5.1.17. The parameter "-r" means receive mode.

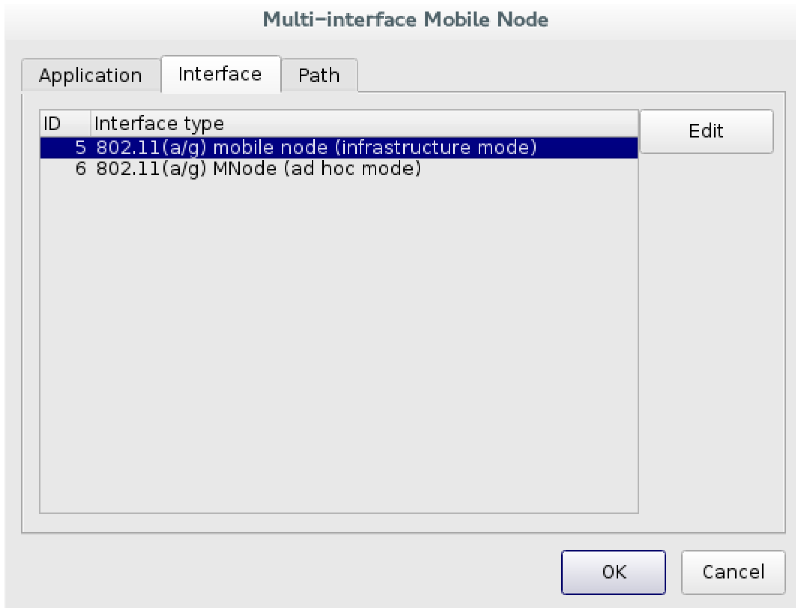


Figure 5.1.16 *The multi-interface mobile node4*

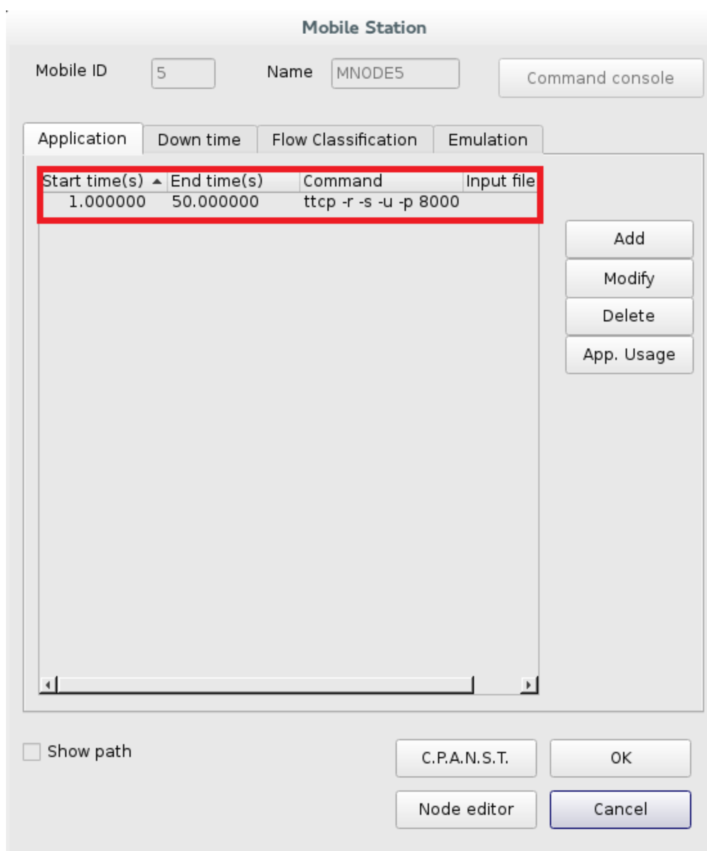


Figure 5.1.17 *Add application ttcp to interface id 5 node4*

Next, select the interface ID 6: 802.11(a/g) mobile node (ad hoc mode), click “Edit” button, to add a command line: “stcp -lip 1.0.1.1 -p 8001 1.0.1.2” which send TCP packets to node7 as figure 5.1.16. The command "-lip IP address" means binding the source IP address. Please notice that each interface should bind with its own interface IP address in multi-interface mobile node. To do this, the simulator could get the information of which interface should send packets under multi-interface mobile node.

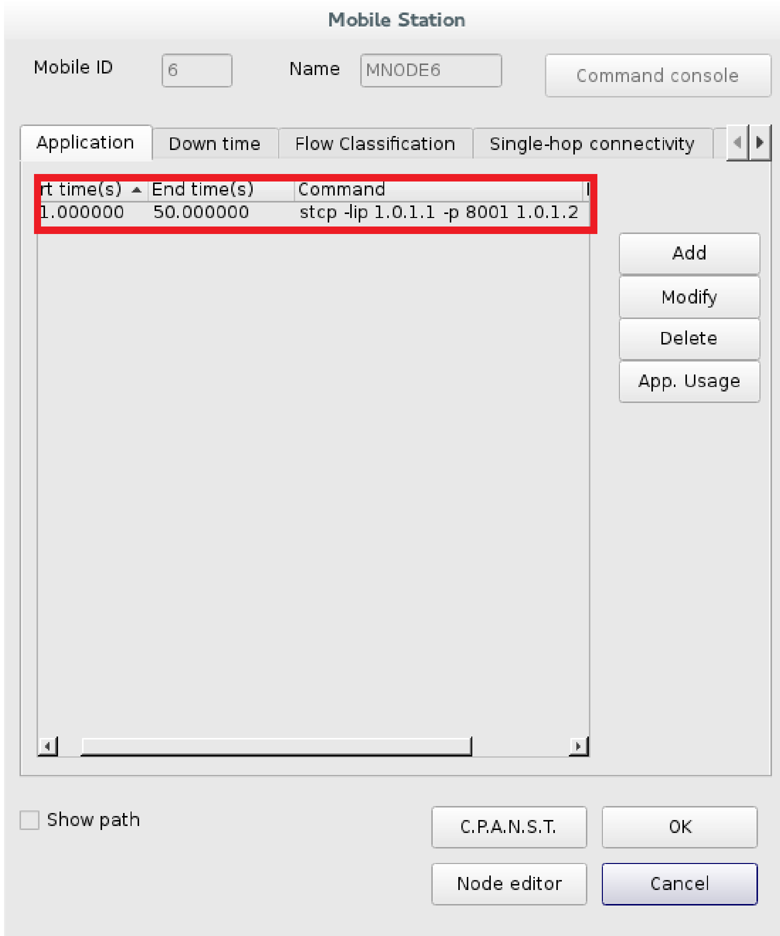


Figure 5.1.16 *Add application stcp to interface id 6*

Return to the property dialog of multi-interface mobile nod4 then choose tab “Path”. The path information is displayed as figure 5.1.17 which user specified by “Create a moving path” in D mode . User can change the path information by using “Add”, “Modify” and “Delete” buttons.

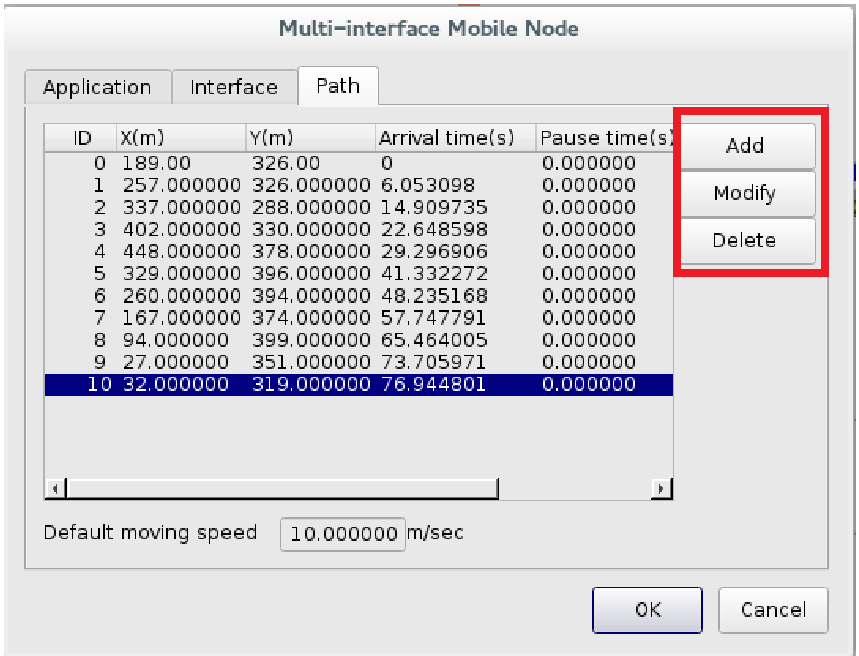


Figure 5.1.17 *The paths of node4*

iii. Node7: Double click 802.11(a/g) adhoc mobile node7 and select tab "Application". To add a command line: "rtcp -p 8001" which receive the TCP packet from node4 as figure 5.1.17.

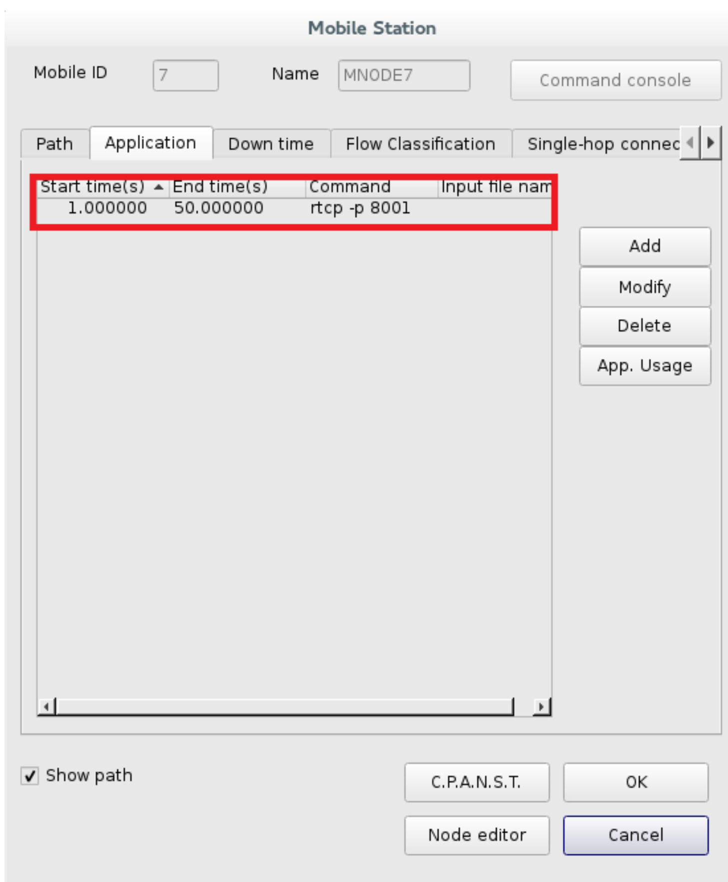


Figure 5.1.17 *Add application rtcp to node7*

Click “Node editor” button to see the protocol stack of node7. One of the difference between ad hoc mode and infrastructure mode is there is a GOD module  in ad hoc mode as figure 5.1.18.For GOD module, GUI provides a way to “Set the Frequency of Updating Mobile Node Routing Paths” . This **Menu → G_Tools → Set the Frequency of Updating Mobile Node Routing Paths (for the God Module)** command controls how frequently the routing paths should be recomputed for the GOD routing module. A higher frequency allows the GOD routing module to respond to link failures more quickly but at the cost of generating a larger file to store more information.

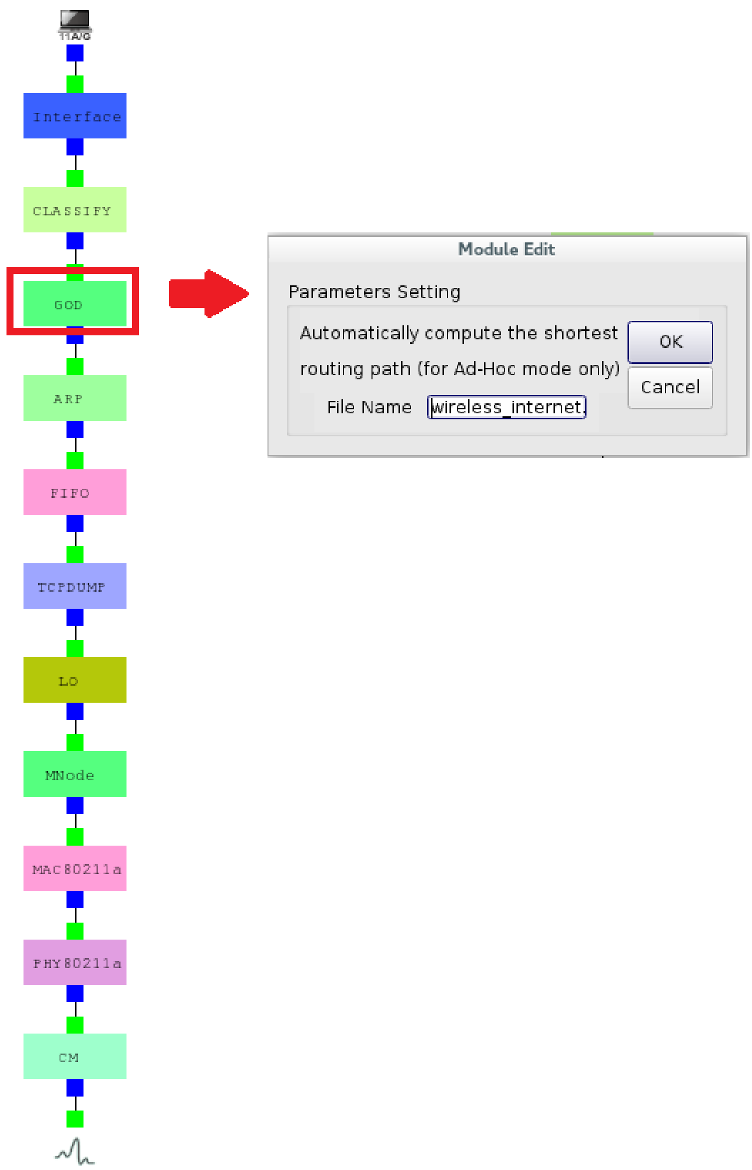
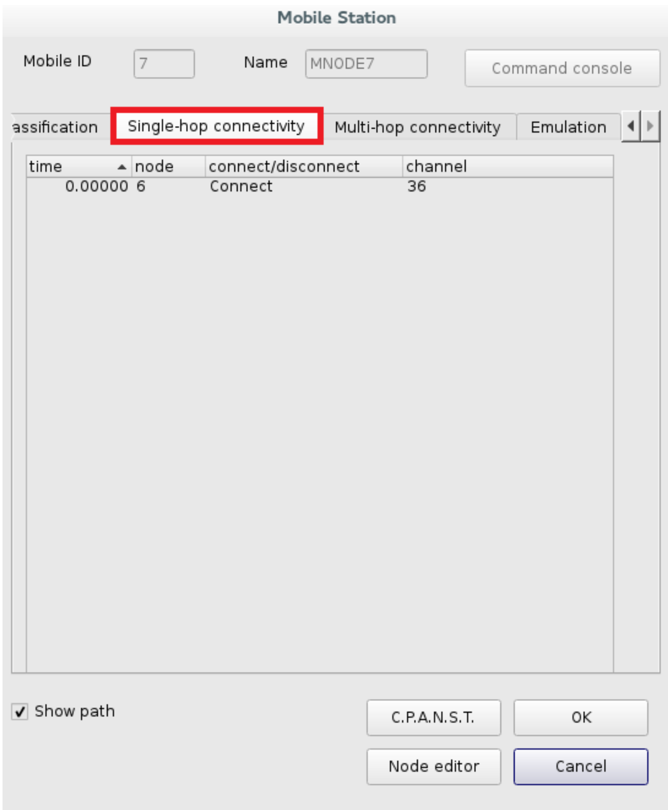


Figure 5.1.18 *The protocol stack of IEEE 802.11(a/g) adhoc mode node7*

The other difference is tab “**Single-hop connectivity”** and tab “**Multi-hop connectivity”** in the ad hoc mobile property dialog as figure 5.1.19. Under the “**Single-hop connectivity”** tab, the GUI program will calculate and show what time this mobile node can/cannot reach other mobile nodes in its single-hop transmission range. Another one, under the “**Multi-hop connectivity”** tab, the GUI program will calculate and show what time this mobile node can/cannot reach other mobile nodes via multiple hops, through the ad hoc mode forwarding. These two functions are provided for comparing research results with the ideal results. The outputs of these two functions represent the most accurate and ideal routing paths that only "God" can know and achieve at any time.

GUI program will automatically calculate the routing path according to the path that user specified.



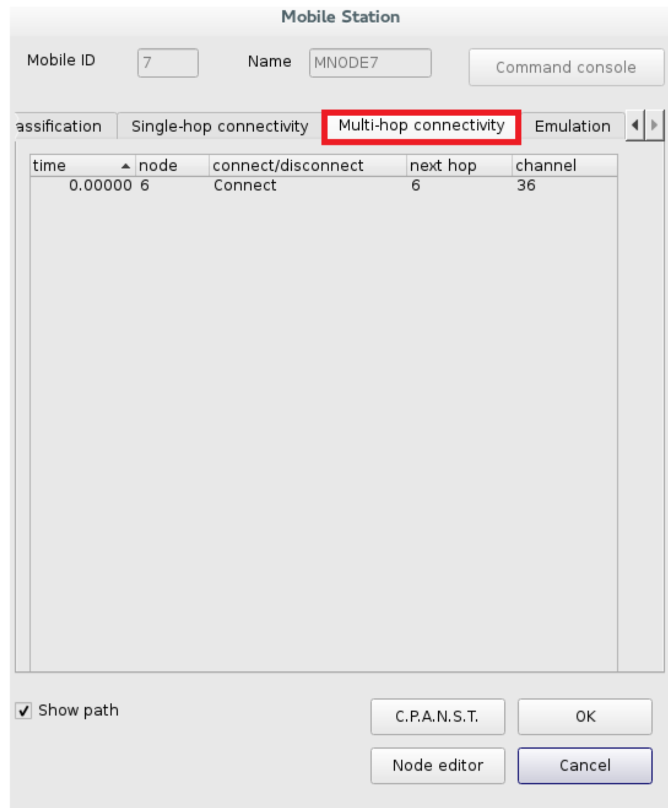


Figure 5.1.19 *The single hop and multi hop connectivity of IEEE 802.11(a/g) ad hoc mode node7*

iv. Node10: Double click IEEE 802.11(n) infrastructure mobile node10 and add the command: "ttcp -r -s -p 8002" which could receive TCP packet from node11.

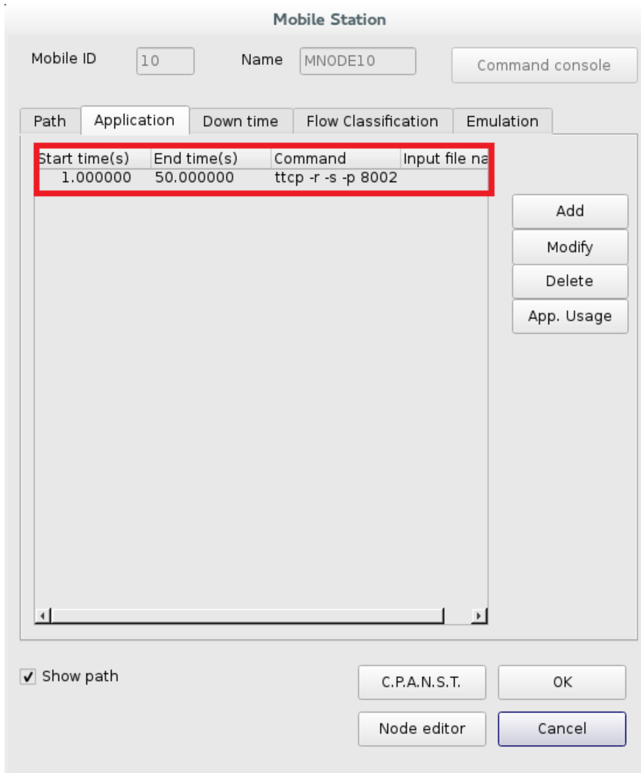


Figure 5.1.20 *Add application ttcp to node10*

Click "Node editor" button to see the protocol stack of node10. There is an AMSDU module 

in the protocol stack of IEEE 802.11(n). The purpose of AMSDU module is to aggregate MSDU frames to an AMSDU frame for IEEE 802.11(n). An AMSDU frame only contains MSDU sub-frames. And a MSDU sub-frame collects Ethernet frames to be transmitted to one or multiple destinations and wraps them in a single IEEE 802.11n frame. The default AMSDU module is not enabled as the following figure 5.1.21.

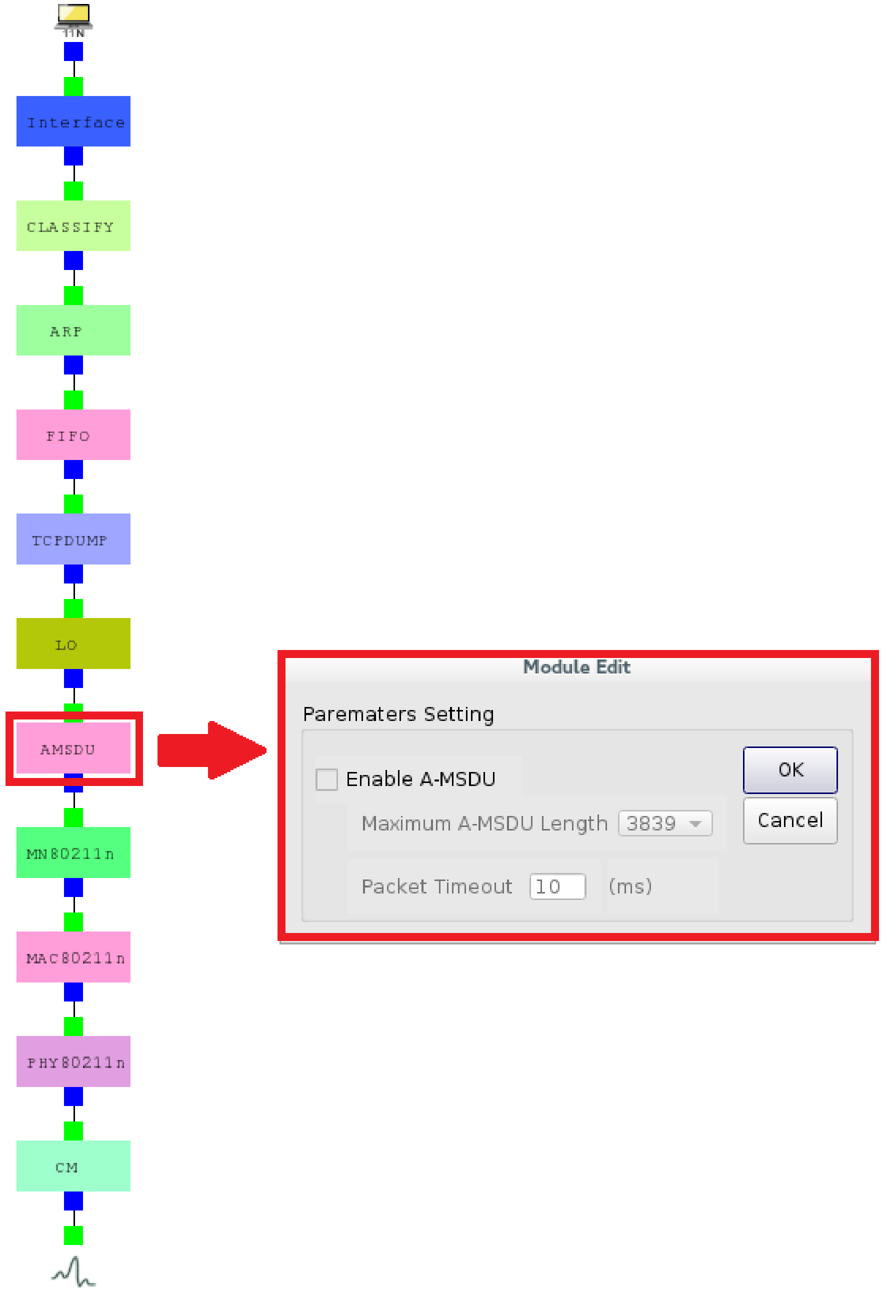


Figure 5.1.21 *The AMSDU module of IEEE 802.11(n) infrastructure mode node10*

In addition, there are "BLOCK ACK" and "A-MPDU" mechanisms setting parameters in MAC module of IEEE 802.11(n). "A-MPDU" is the other type of frame aggregation for MPDU. The MPDU aggregation also collects Ethernet frames to be transmitted to a single destination, but it wraps each frame in an IEEE 802.11(n) MAC header. Normally this is less efficient than MSDU aggregation, but it may be more efficient in environments with high error rates due to a mechanism called block acknowledgement ("BLOCK ACK"). This mechanism will aggregate several acknowledgements to a block acknowledgement. The default disables “Block Ack and A-MPDU” mechanism in MAC80211n module as the following figure 5.1.22.

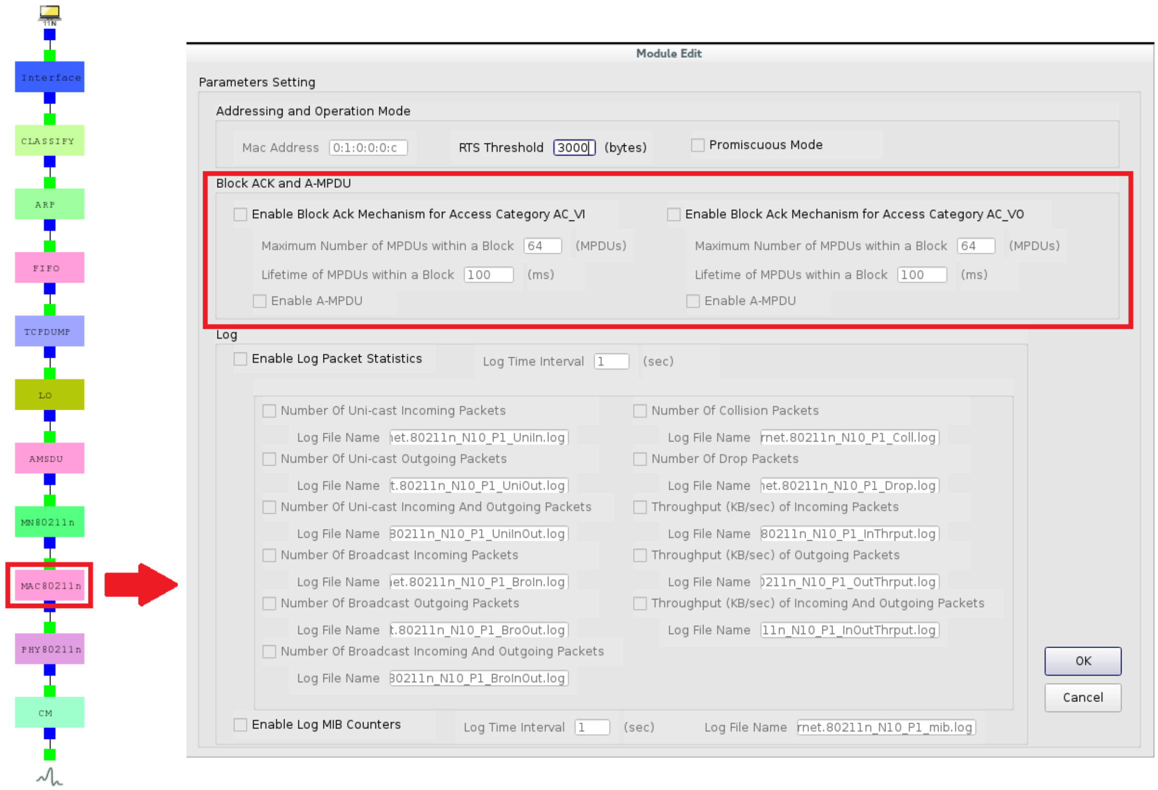


Figure 5.1.22 *The MAC module of IEEE 802.11(n) infrastructure mode node10*

v. Double click multi-interface mobile node11 and choose tab "Interface". To select the interface ID 12: IEEE 802.11(n) mobile node (infrastructure mode) with click “Edit” button then add the command: “ttcp -t -s –L 1.0.4.3 -p 8002 1.0.4.2” to send TCP packet to node10 as figure 5.1.23. The command "-L IP address" means binding the source IP address. Then, select the interface ID 13: IEEE 802.11(n) MNode (ad hoc mode) with click “Edit” button and add the command: “rtcp -p 8003” to receive TCP packet from node14 as figure 5.1.24.

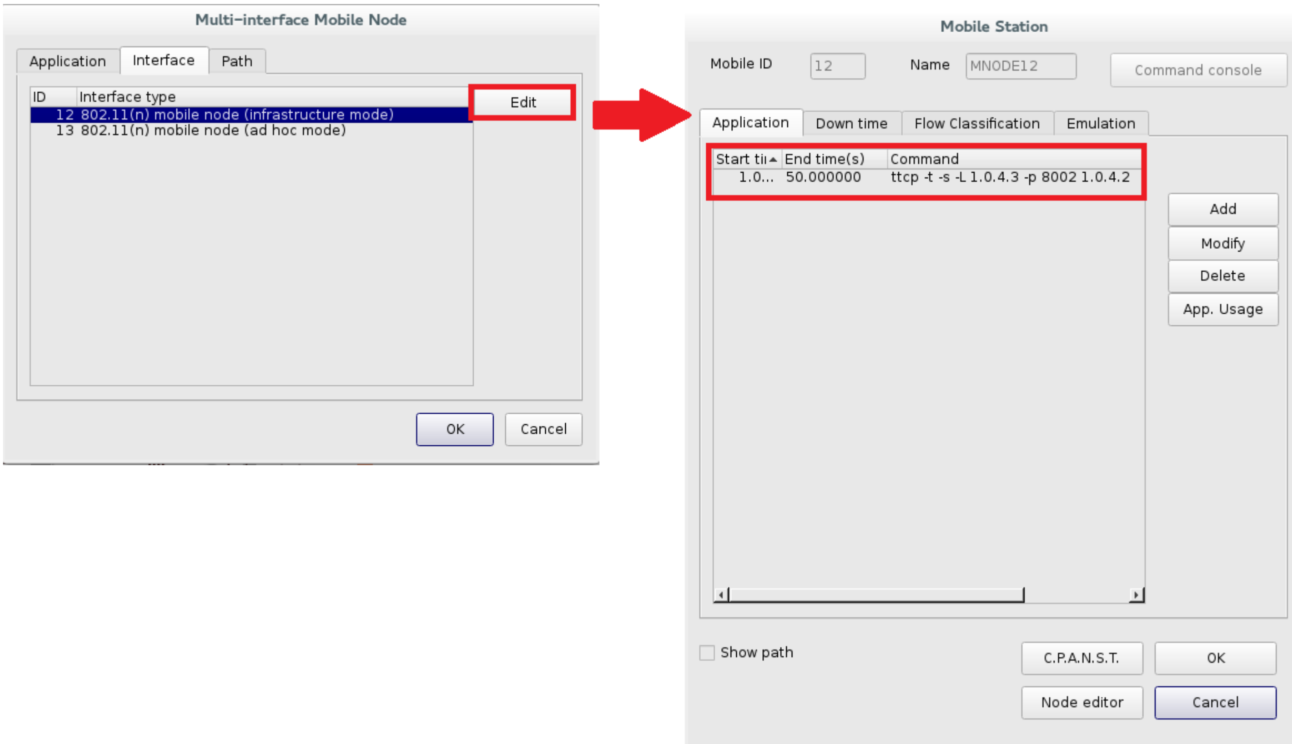


Figure 5.1.23 *Add application to interface id 12 of node11*

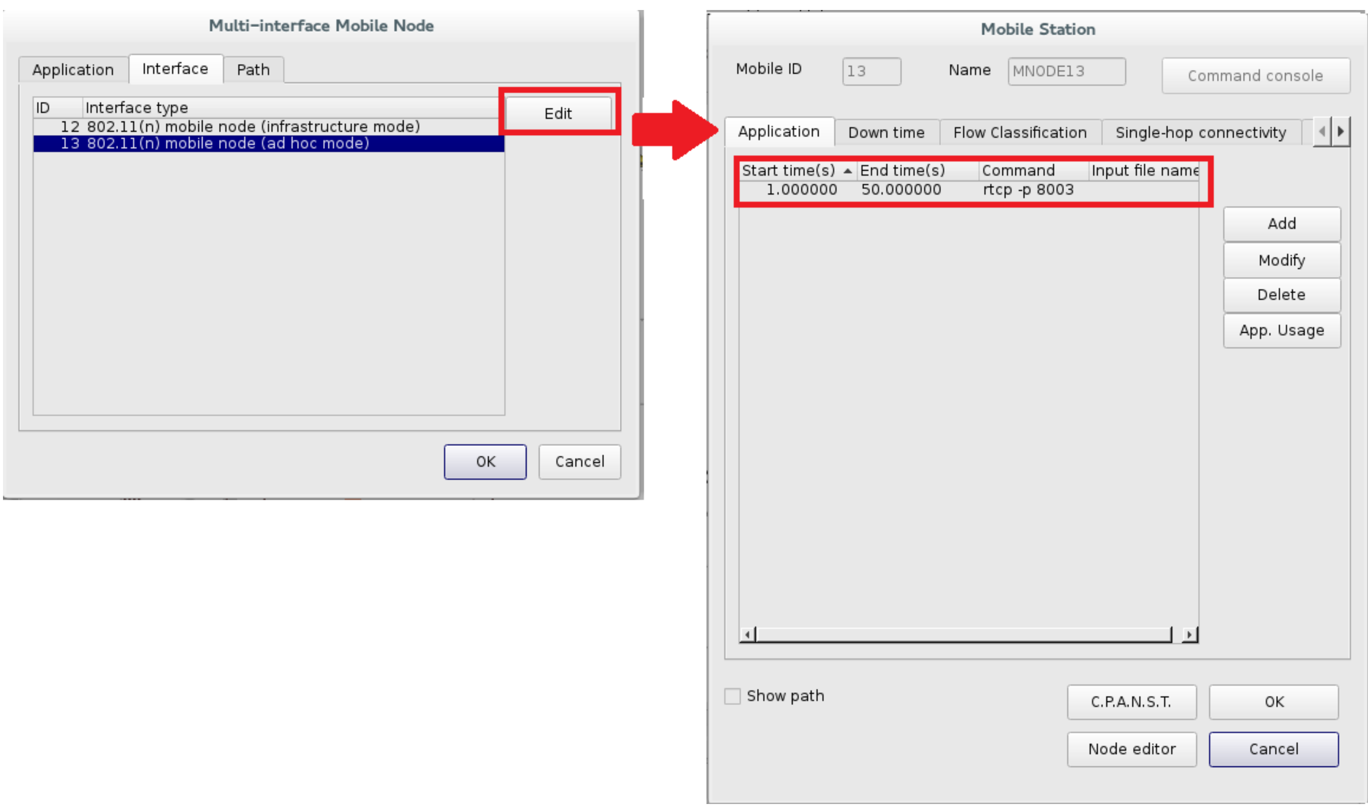


Figure 5.1.24 *Add application to interface id 13 of node12*

vi. Node14: Double click IEEE 802.11(n) ad hoc mobile node14 and add the command “stcp -p 8003 1.0.2.1” to send TCP packet to node11.

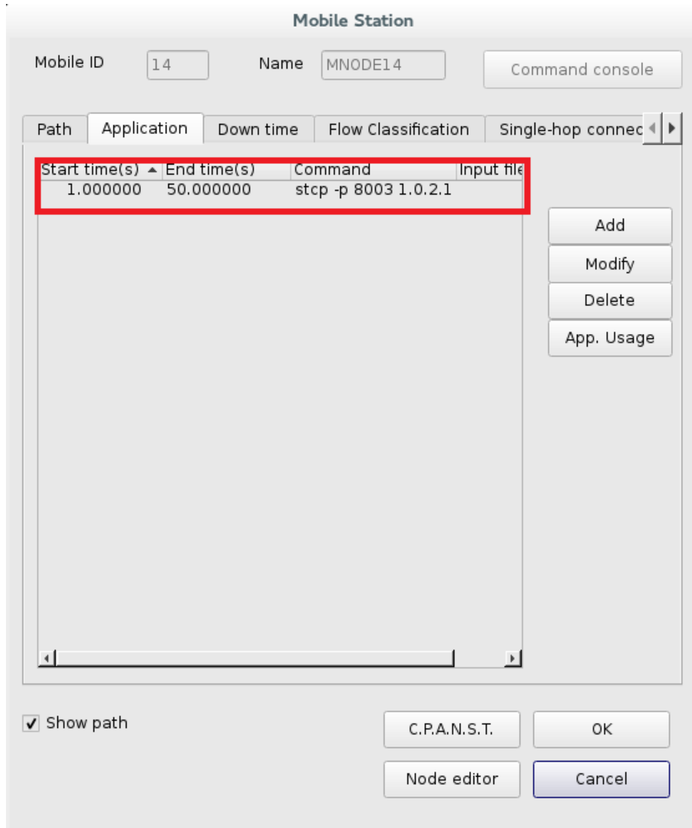


Figure 5.1.25 *Add application to node14*

(3) Run Simulation

After complete step by step settings, user can switch mode from E mode E to R mode R . Then from "**Menu** → **Simulation** → **Run**" to execute the simulation.

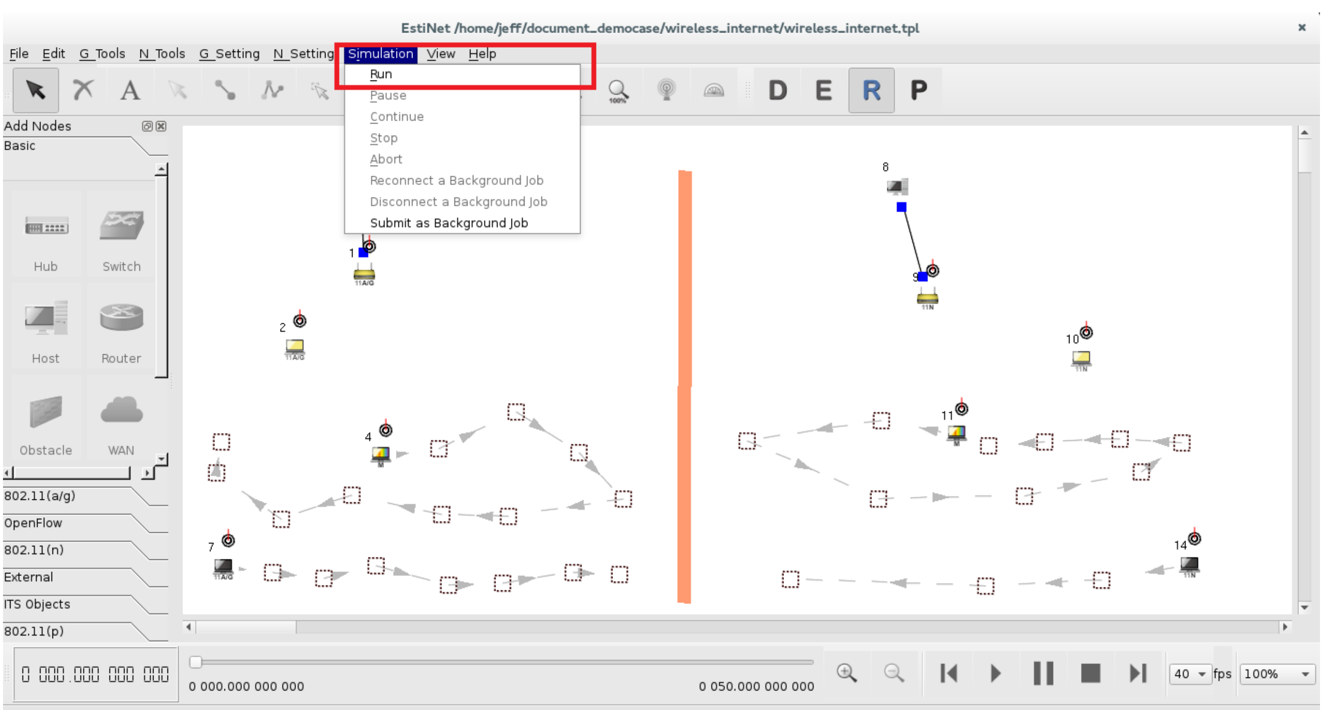


Figure 5.1.26 *Ready to run simulation*

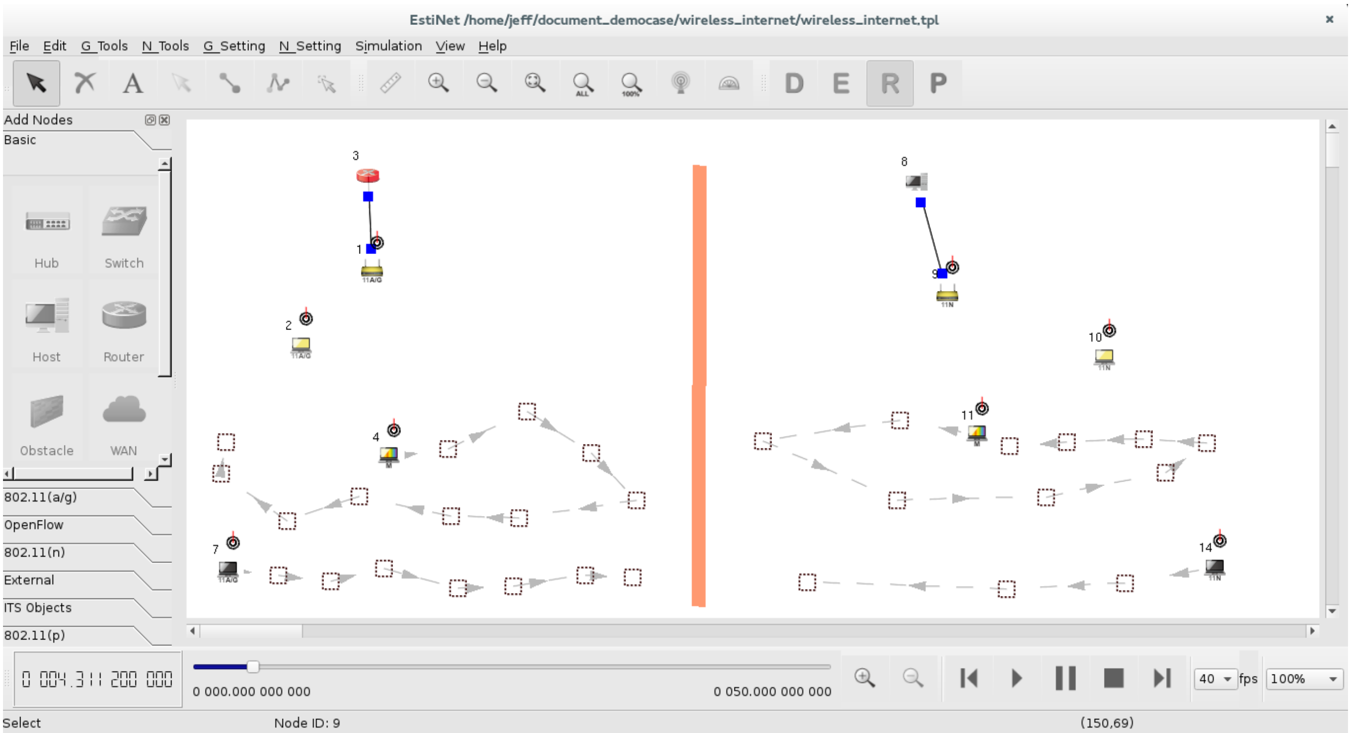


Figure 5.1.27 *Simulation is running*

(4) Playback

When simulation is completed, GUI will get result back from simulation engine automatically.

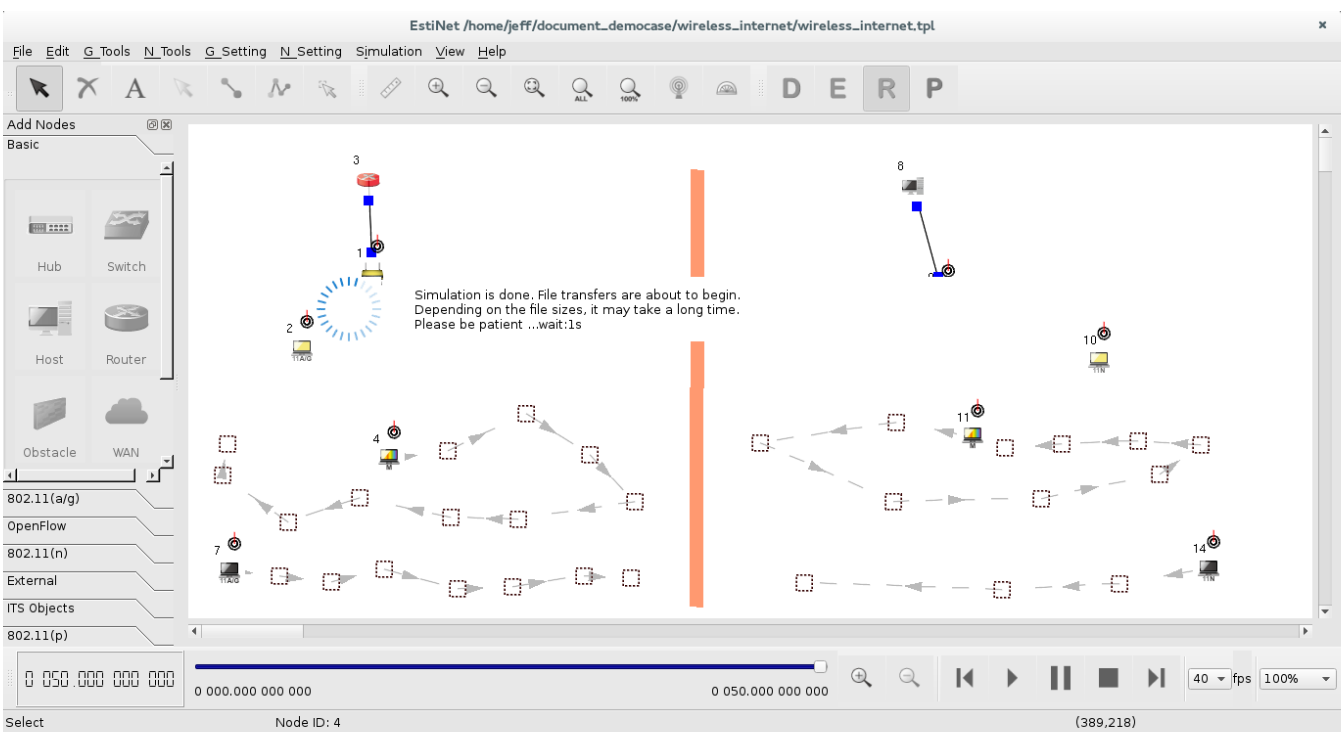


Figure 5.1.28 *Wait for getting simulation result*

Then GUI will switch to P mode P. User could click button to play the simulation result.

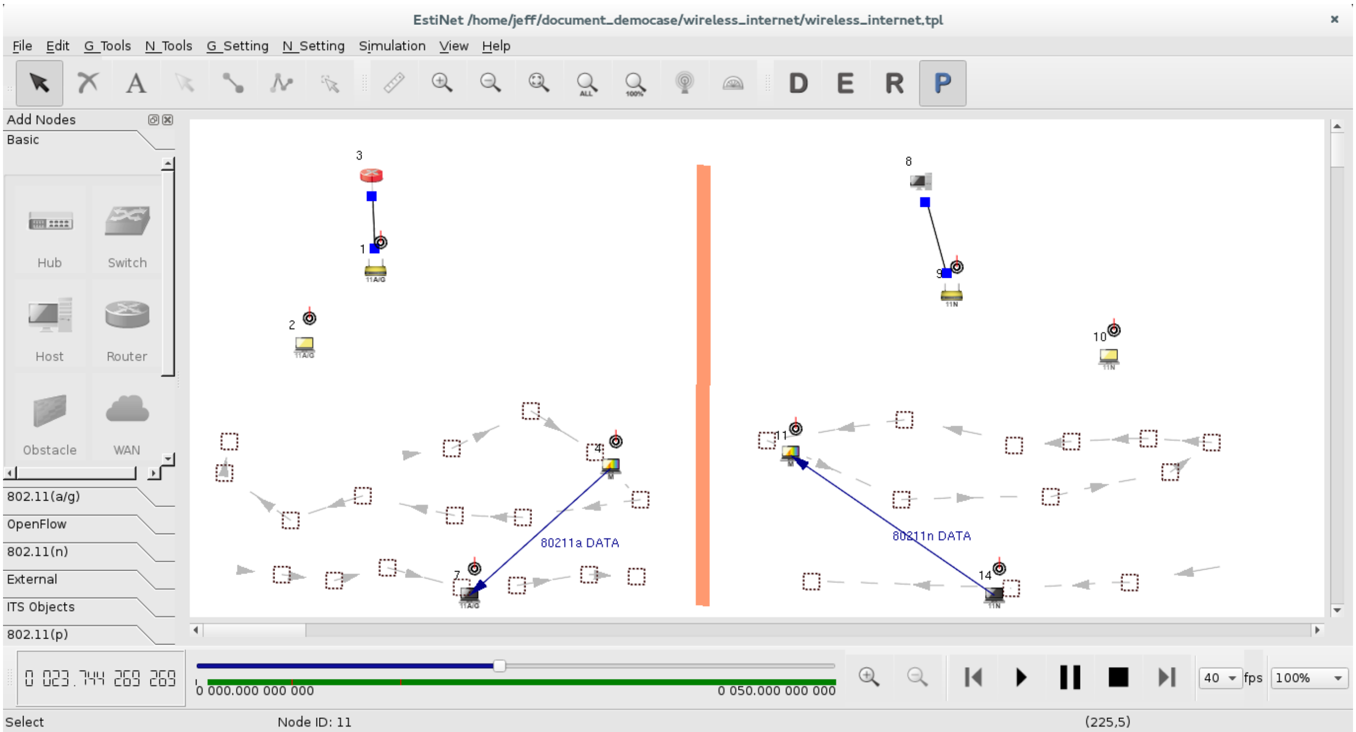


Figure 5.1.29 *Playback*

GUI provides a tool "**Specify physical-layer and channel model parameters** ( )" to specify the attributes of wireless physical-layer modules and the parameters of wireless channel models in D mode **D** and E mode **E** . This tool integrates all of the functions for setting the physical-layer and channel model parameters into a unified user interface, which includes the following components: 1) antenna attribute specification, 2) channel model specification, 3) connectivity calculation and display, 4) antenna gain pattern and antenna directivity specification.

To start this tool, user should click the "Specify physical-layer and channel model parameters" tool button which at "User Control" area. Then click the mobile node which user intend to configure. It will pop up a setting dialog as figure 5.1.30.

When "**Use the transmitting node's perspective**" mode is selected, GUI will show which nodes will be interfered by the chosen node (the clicked node) in case it is transmitting a packet. This is accomplished by drawing a red-colored dotted arrow ( ) from the chosen node to a potentially-interfered node as figure 5.1.31.

When "**Use the receiving node's perspective**" mode is selected, GUI will show the maximum range that a node **X** can interfere with the chosen node, where node **X** denotes the set of nodes in the network other than the chosen node itself. In addition, if a node X can interfere with the chosen node using current setting (which includes the current antenna settings of these two nodes, the distance between them, and the existence of obstacles on their line-of-sight path, etc.), GUI will draw a red-colored dotted arrow ( ) from node **X** to the chosen node, indicating that if the current setting is adopted, the transmission activity of node **X** can be sensed by the chosen node and may block the receiving activity of the chosen node as figure 5.1.32.

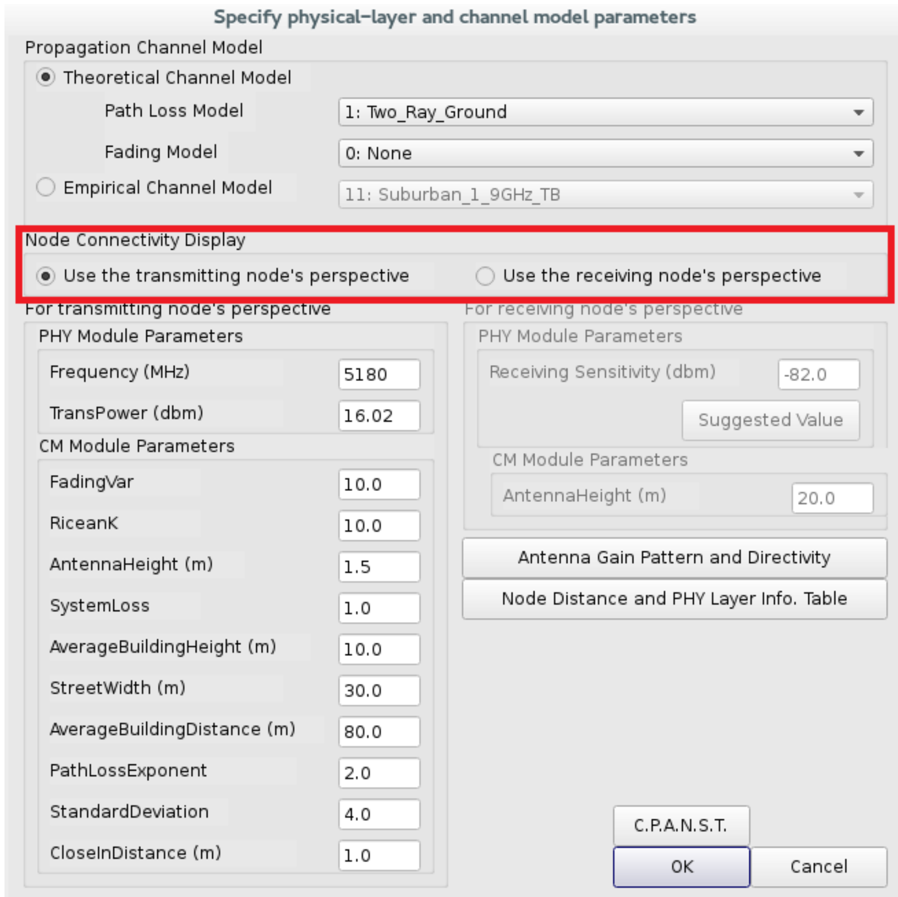


Figure 5.1.30 *The setting dialog of "**Specify physical-layer and channel model parameters** ( ) "*

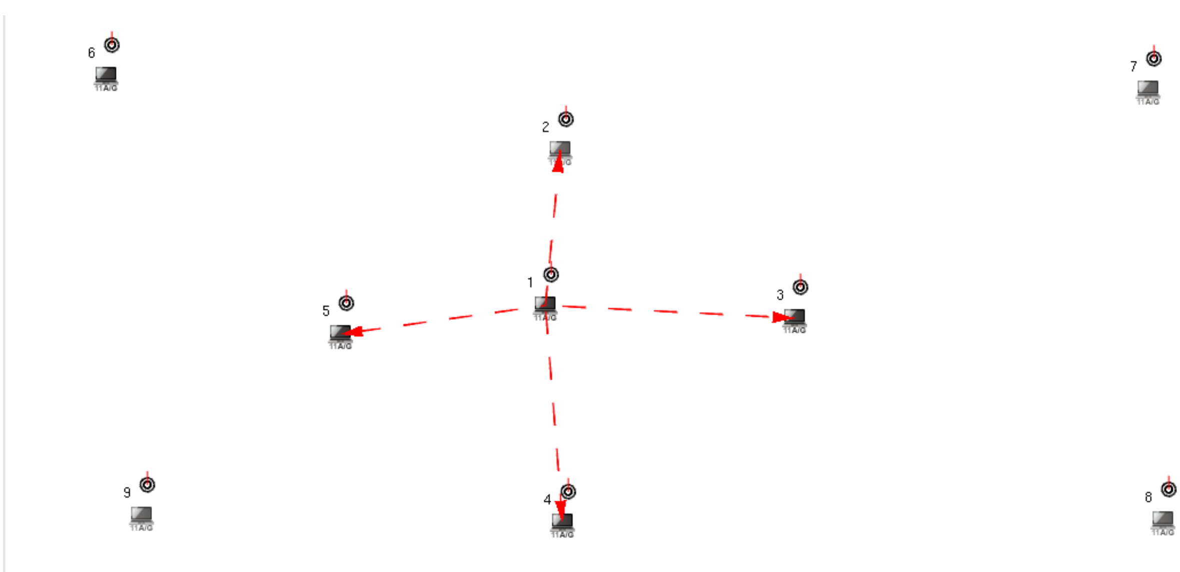


Figure 5.1.31 *"Use the transmitting node's perspective"*

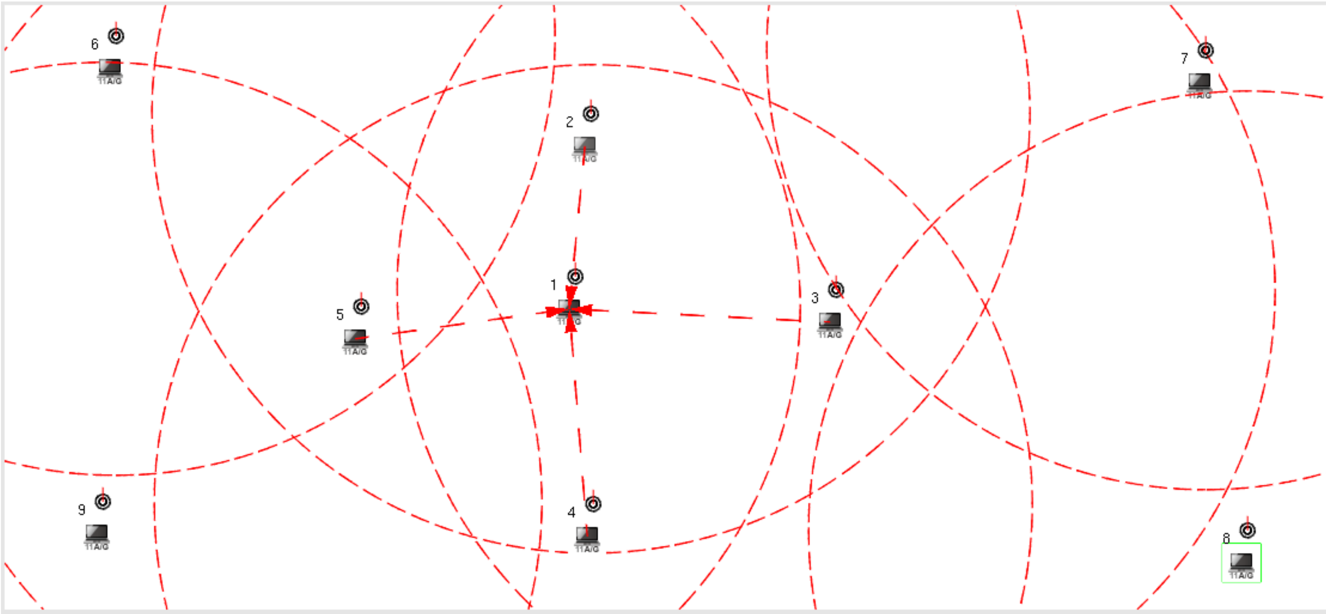


Figure 5.1.32 *Node connectivity display when select "Use the receiving node's perspective"*

Note that, when using the "**Use the receiving node's perspective**" mode, one is required to properly set the parameters of the "**AntennaHeight (m)**" and "**Receiving Sensitivity (dbm)**", so that GUI could automatically and correctly determine the connection of nodes. If user could not make sure what values are suitable for the network type, EstiNet lists "Suggested Value" by Data Rate (Mbps) and CSThreshold (dBm) for IEEE 802.11(a/g) and IEEE 802.11(n). User could click the "**Suggested Value**" button to show the suggested power threshold value table as figure 5.1.34.

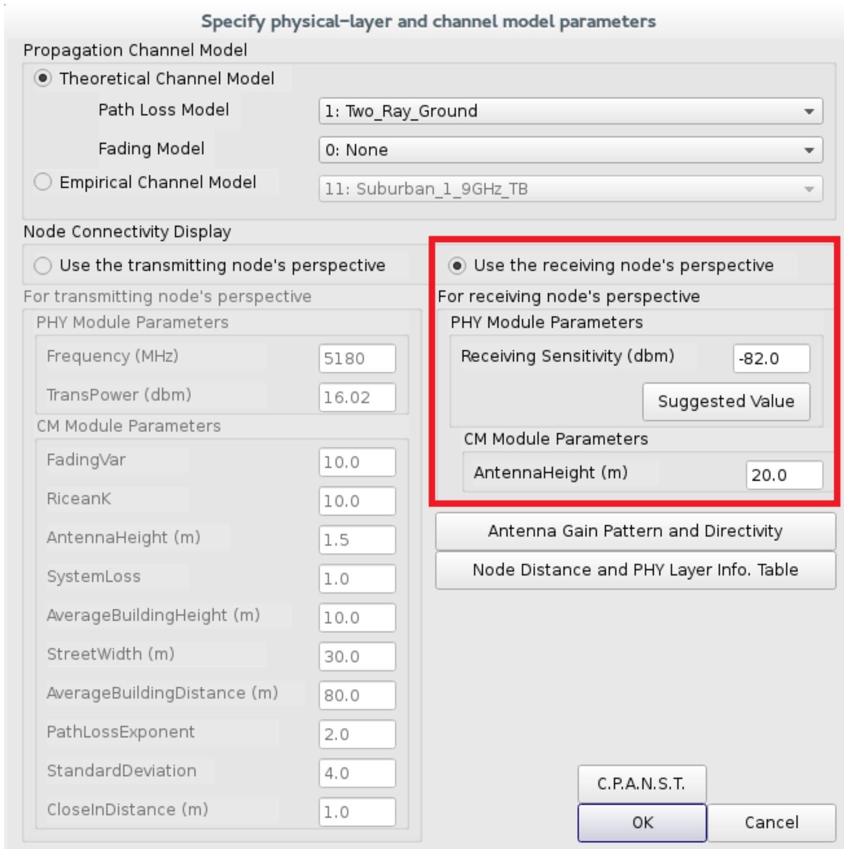


Figure 5.1.33 *Set parameters for receiving node's perspective*

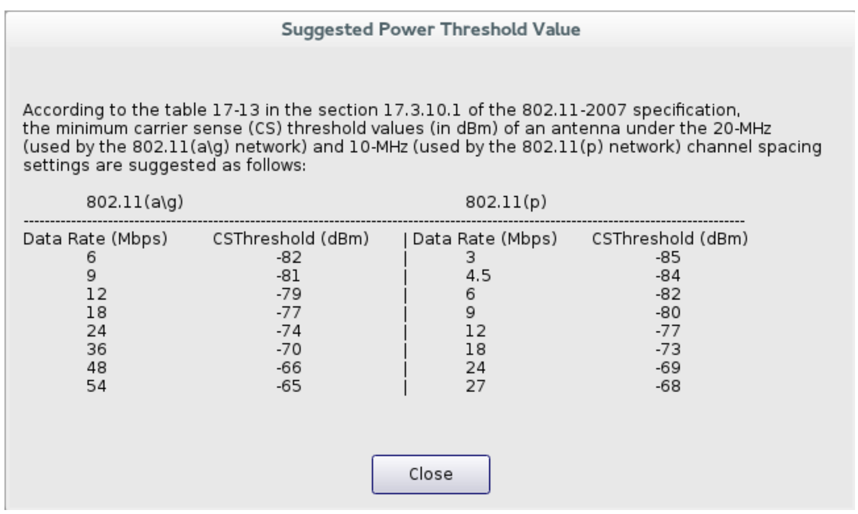


Figure 5.1.34 *Suggested power threshold value*

On the top of the dialog box is the column of channel model. User could choose the signal propagation channel model that will be used in the simulation. EstiNet categorizes the supported channel models into two classes. One is the "**Theoretical Channel Model**" class which collects the channel models that are developed using theoretical formulas. In this sample, just select the "**Path Loss Model**" that is intended to be used in the simulation. If user selects "**Fading Model**", it will simulate more realistic fading effect in simulation.

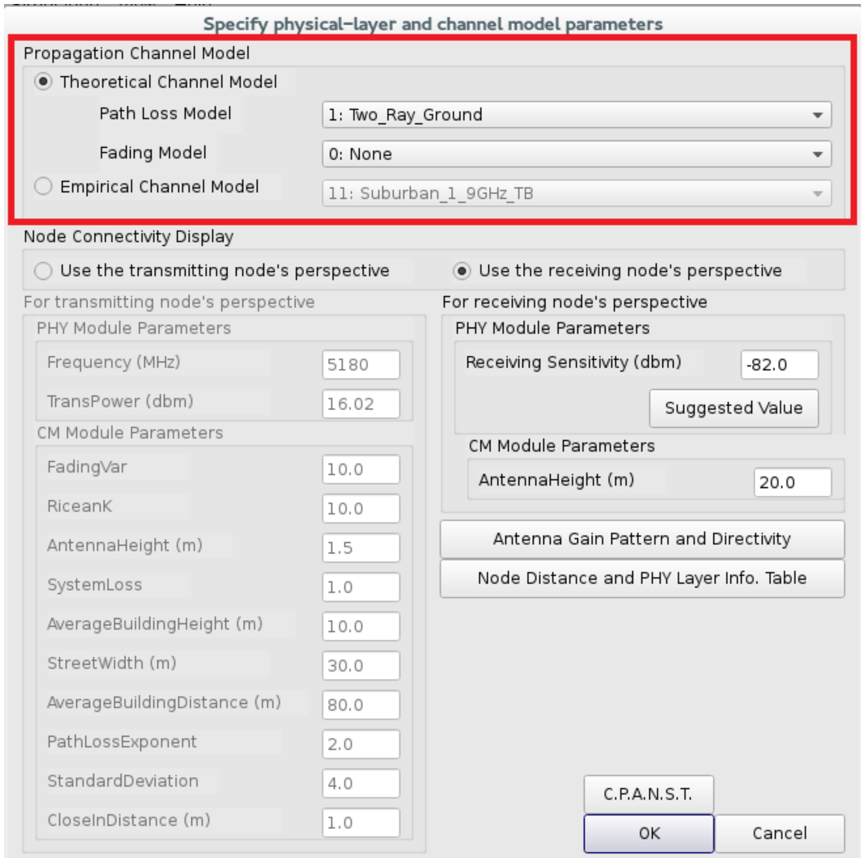


Figure 5.1.35 *Two kinds of Channel models*

Currently, there are three theoretical path-loss models in EstiNet simulator “Free_Space,” “Two_Ray_Ground,” and “Free_Space_and_Shadowing”. And there are three different fading models: “no-fading (None),” “Rayleigh Fading,” and “Ricean Fading.”

The other model class is the “**Empirical Channel Model**” which collects channel models that are developed based on real-life measurement results. So far, EstiNet supports 35 empirical channel models, e.g., “LEE_Microcell,” “Okumura,” “COST_231_Hata,” and so forth. User could choose one of them to simulate wireless channels.

In addition to setting the channel model used in simulation here, alternatively user could choose and configure the used channel model (CM) via **Node Editor** as figure 5.1.36.

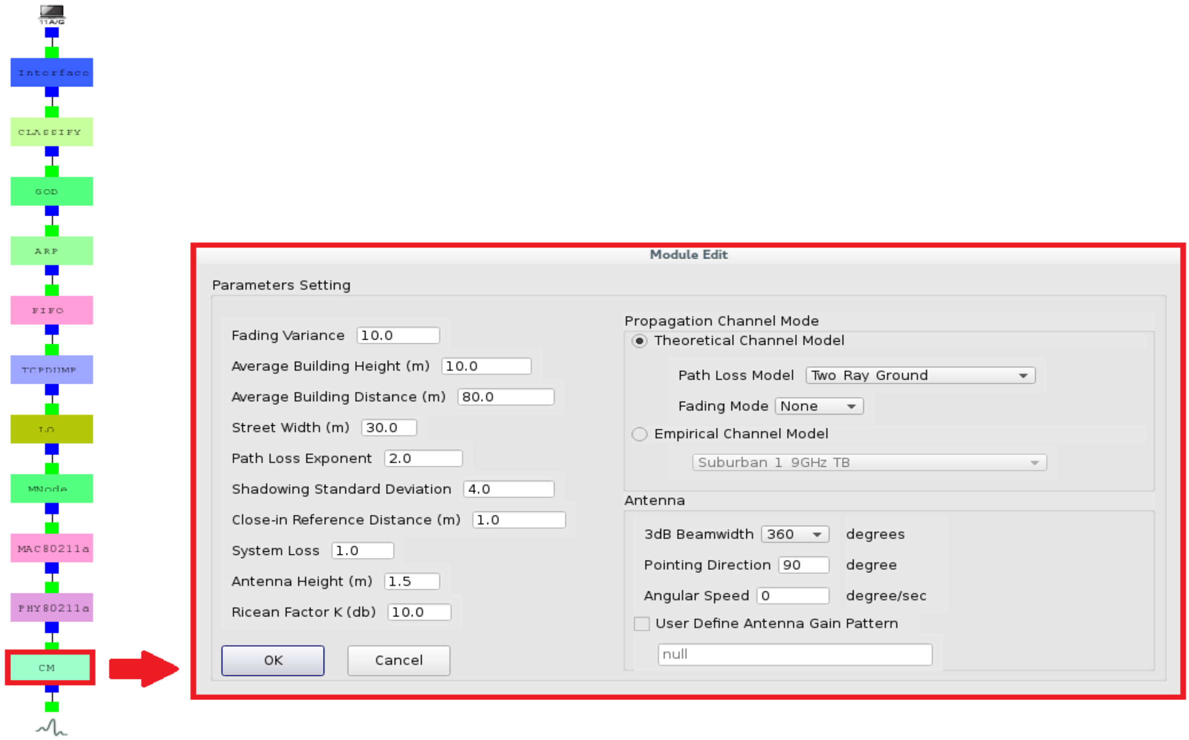


Figure 5.1.36 *Configure the used channel model (CM) via Node Editor*

Recalculation and Display for Node Connectivity

Click "**Node Distance and Physical-layer Information Table**" (NDPIT) button to show the following information (as figure 5.1.37): 1) Distance: it lists the distance between the chosen node and specify NID node with the same PHY protocol; 2) TxGain, RxGain: the antenna gain values for each pair of nodes; 3) I.R. (m): the interference range (I.R.) from the perspective chosen node to specify NID node; 4) RxSensitivity (dbm): the RxSensitivity value from the perspective chosen node to specify NID node.

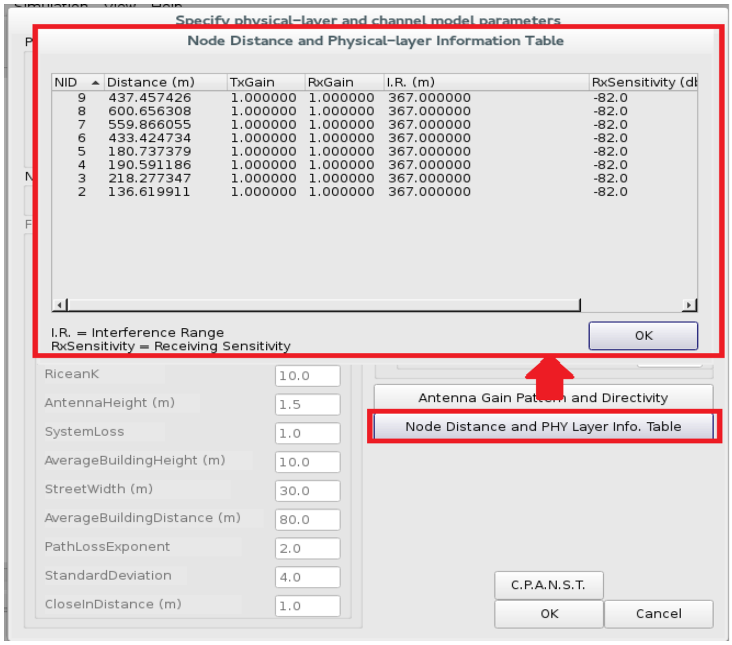


Figure 5.1.37 *Node Distance and Physical-layer Information Table*

Note that the meanings of the 2nd, 3rd, and 4th items depend on whether the "**Use the transmitting node's perspective**" mode is used or the "**Use the receiving node perspective**" mode is used. For example, suppose that the chosen node is node1 and the first entry of NDPIT is for node9. When the "**Use the transmitting node perspective**" mode is used, the *TxGain* value denotes the antenna gain value of the chosen node1 and the *RxGain* value denotes the antenna gain value of node9. On the other hand, when the "**Use the receiving node perspective**" mode is used, the *TxGain* value denotes the antenna gain value of node9 and the *RxGain* value denotes the antenna gain value of the chosen node1.

Similarly, when the "**Use the transmitting node perspective**" mode is used, the interference range denotes the maximum range that the chosen node1 can interfere with node9 and the receiving sensitivity value denotes the minimum power threshold value that node9 should set. If it want to be capable of sensing the transmission activity of node1. In contrast, when the "**Use the receiving node perspective**" mode is used, the interference range denotes the maximum range that node9 can interfere with node1 and the receiving sensitivity value denotes the minimum power threshold value that node1 should set, if it wants to be capable of sensing the transmission activity of node9.

Antenna Gain Pattern and Directivity

Click the "**Antenna Gain Pattern and Directivity**" button to specify the gain pattern of a node's antenna and antenna directivity setting as figure 5.1.38.

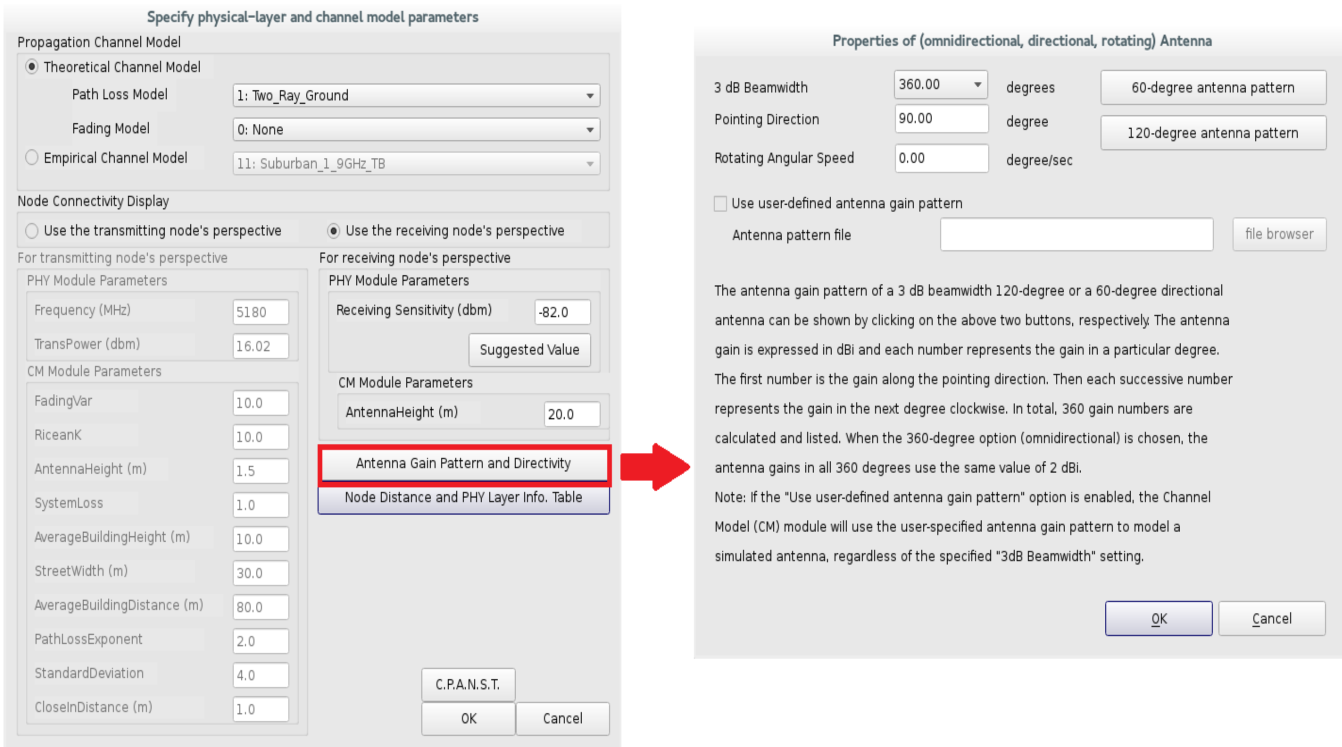


Figure 5.1.38 *The setting dialog of antenna gain pattern and directivity*

In this dialog box, one can specify 1) 3 dB Beamwidth, 2) Pointing Direction (orientation), 3) Rotating Angular Speed 4) Use user-defined antenna gain pattern. The meanings of these attributes are explained below.

User could use the "**Antenna**" tool to set up the default values before adding new mobile nodes. The command path of the "Antenna" is in "**Menu→G_Setting→Antenna**" as figure 5.1.39.

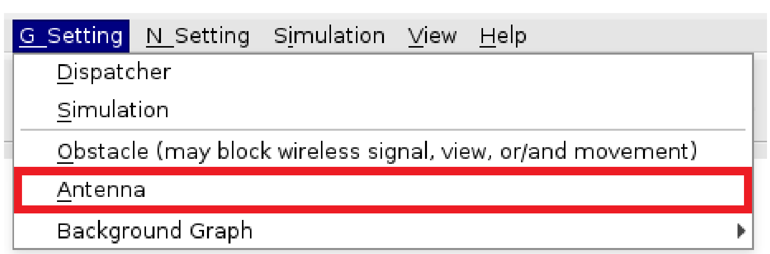


Figure 5.1.39 Menu→G_Setting→Antenna

EstiNet supports three default types of antennas: 1) omni-directional (360-degree) antenna; 2)120-degree 3-dB beamwidth directional antenna; and 3) 60-degree 3-dB beamwidth directional antenna.

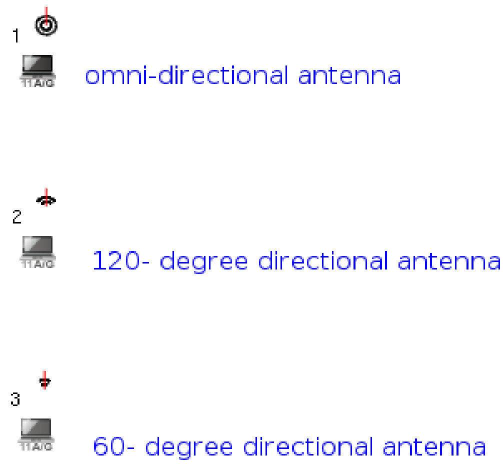


Figure 5.1.40 *Three types of directional antenna*

To show the detail antenna gain value at each degree, user could click the button "**60-degree antenna pattern**" and "**120-degree antenna pattern**" in the dialog box as figure 5.1.41 and 5.1.42.

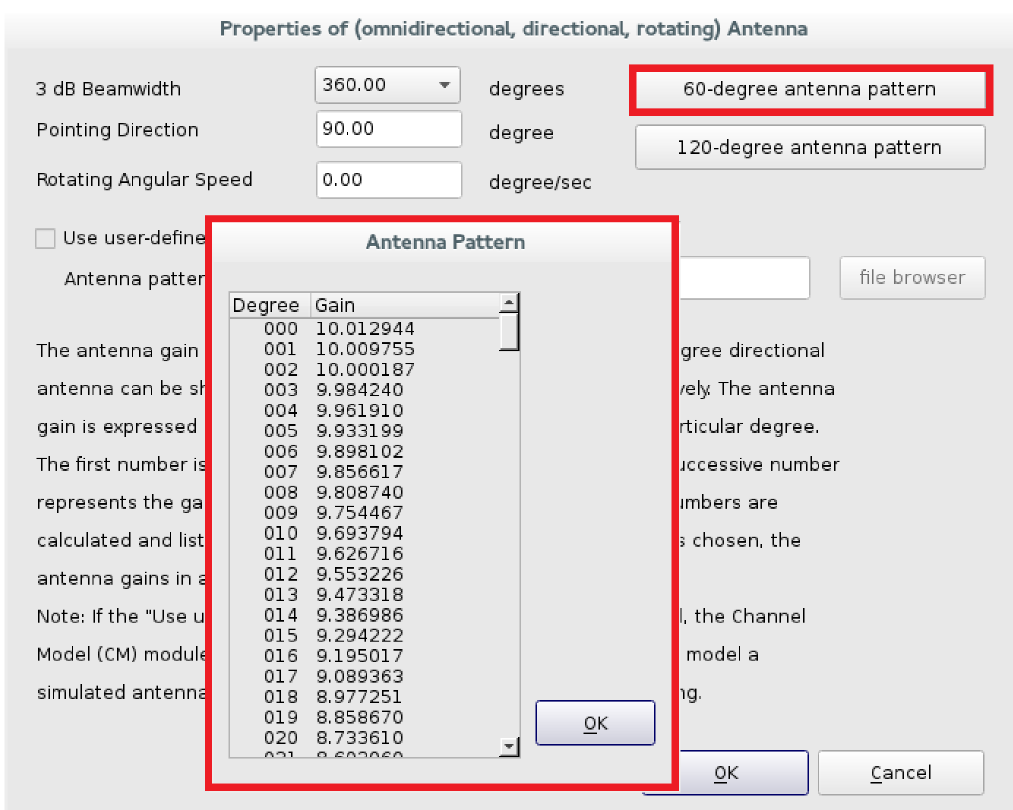


Figure 5.1.41 *Antenna pattern of 60-degree*

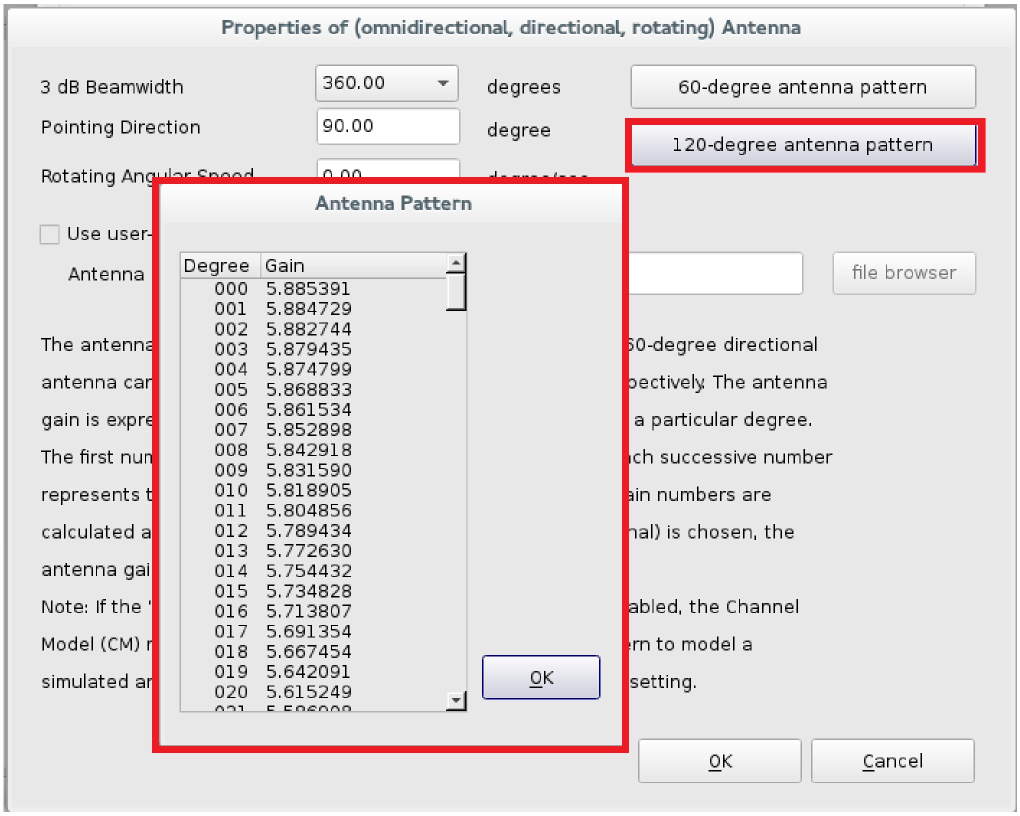


Figure 5.1.42 *Antenna pattern of 120-degree*

Besides using example antenna gain patterns, EstiNet allows user to conduct simulation using an arbitrary antenna gain pattern. To accomplish this, firstly user should turn on the "**Use user-defined antenna gain pattern**" option and then specify the path of antenna gain pattern file in "**antenna pattern file"** field.

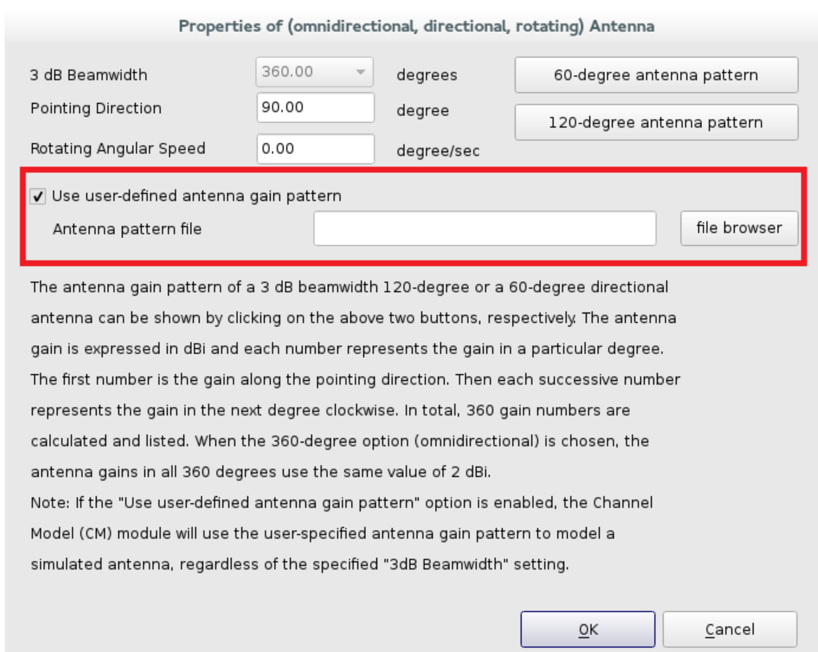


Figure 5.1.43 *Using user-defined antenna gain pattern*

The format of an antenna gain pattern file (.agp file) is simple. Each line represents a gain value entry, which is composed of two fields separated by a comma. The former field denotes the degree of the antenna relative to its pointing direction, while the latter is the gain value of the antenna in dBi. The following figure shows an example of the .agp file.

```
File  Edit  View  Scrollback  Boo
0, 15.000000
1, 15.000000
2, 15.000000
3, 15.000000
4, 15.000000
5, 15.000000
6, 15.000000
7, 15.000000
8, 15.000000
9, 15.000000
10, 15.000000
11, 15.000000
12, 15.000000
13, 15.000000
14, 15.000000
15, 15.000000
16, 15.000000
17, 15.000000
18, 15.000000
19, 15.000000
```

Figure 5.1.44 *The format of the user-defined antenna gain pattern file*

6. Performance Monitor

Performance monitor is a generous-purpose and useful tool that can graphically display plots of performance metrics. For example, it can help user monitors the utilization of a link or the throughput of a TCP connection. This chapter gives an overview of the functions provided by the performance monitor.

(1) Draw topology of performance monitor

In D mode D , user could use these nodes: host and router in “Basic” group to draw a simple topology as figure 6.1.1 and save the topology file as “performance_monitor.tpl.”

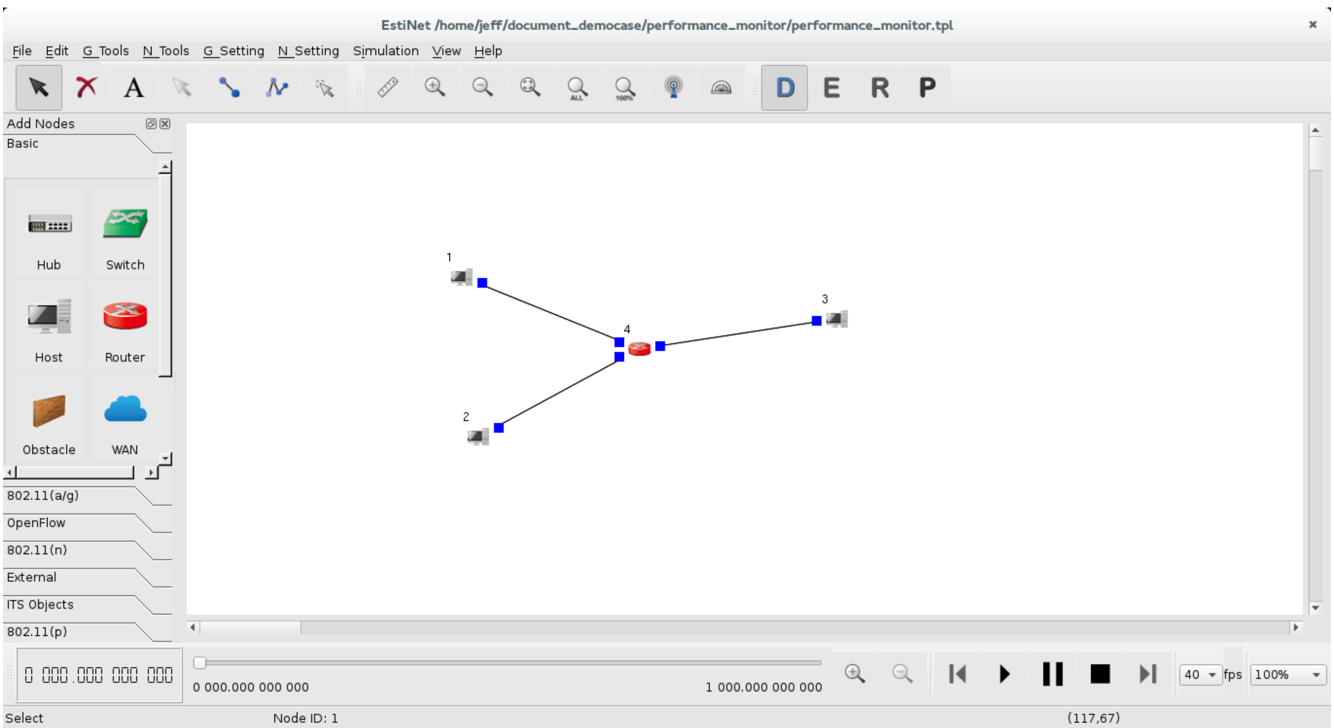


Figure 6.1.1 *The topology of performance monitor*

(2) Setup Properties

Switch to E mode E , the GUI program automatically subdivides various subnets and assigns IP, MAC address as figure 6.1.2.

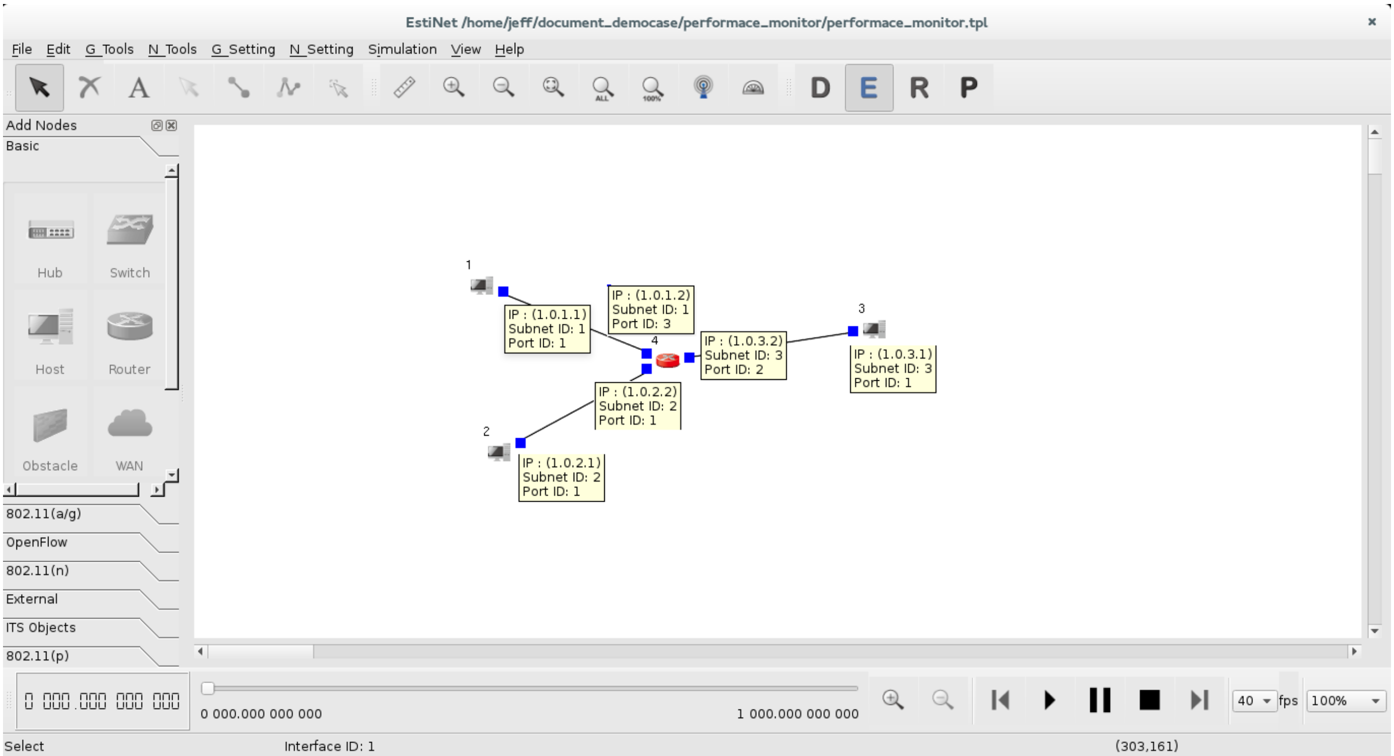


Figure 6.1.2 *The node information of IP address and subnet*

Set Network Traffic:

In this sample, use command “ttcp” to generate TCP packets. Host node1 and node2 send TCP packets to host node3.

i. Double click host node1 and add the command “ttcp -t -s -p 8001 1.0.3.1” under the “Application” tab.

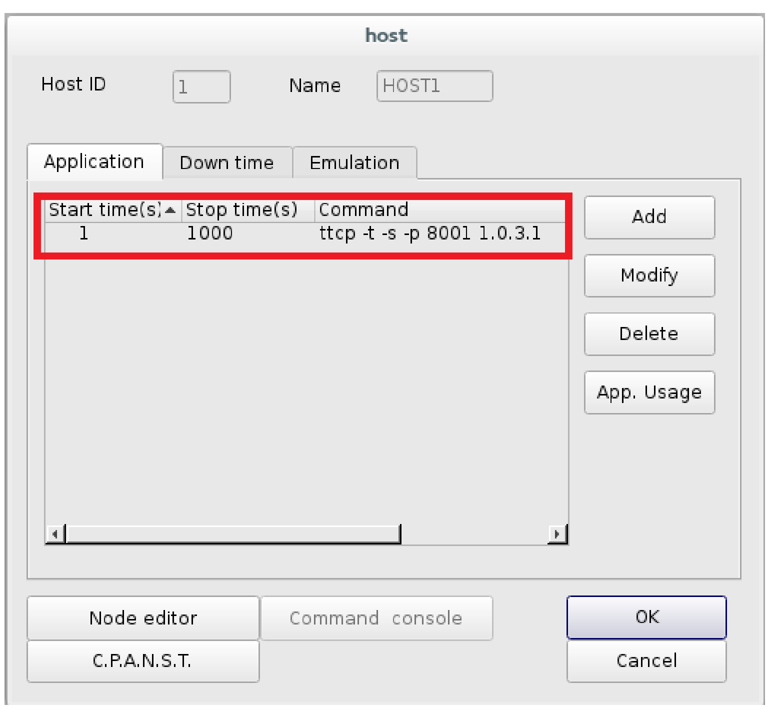
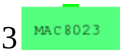


Figure 6.1.3 *Add application ttcp to node1*

Click “Node editor” button and double click module MAC8023  . It will pop up a “Module Edit” dialog. Check the parameter: “Log Packet Statistics” and select “Throughput (KB/sec) of Outgoing Packets”.

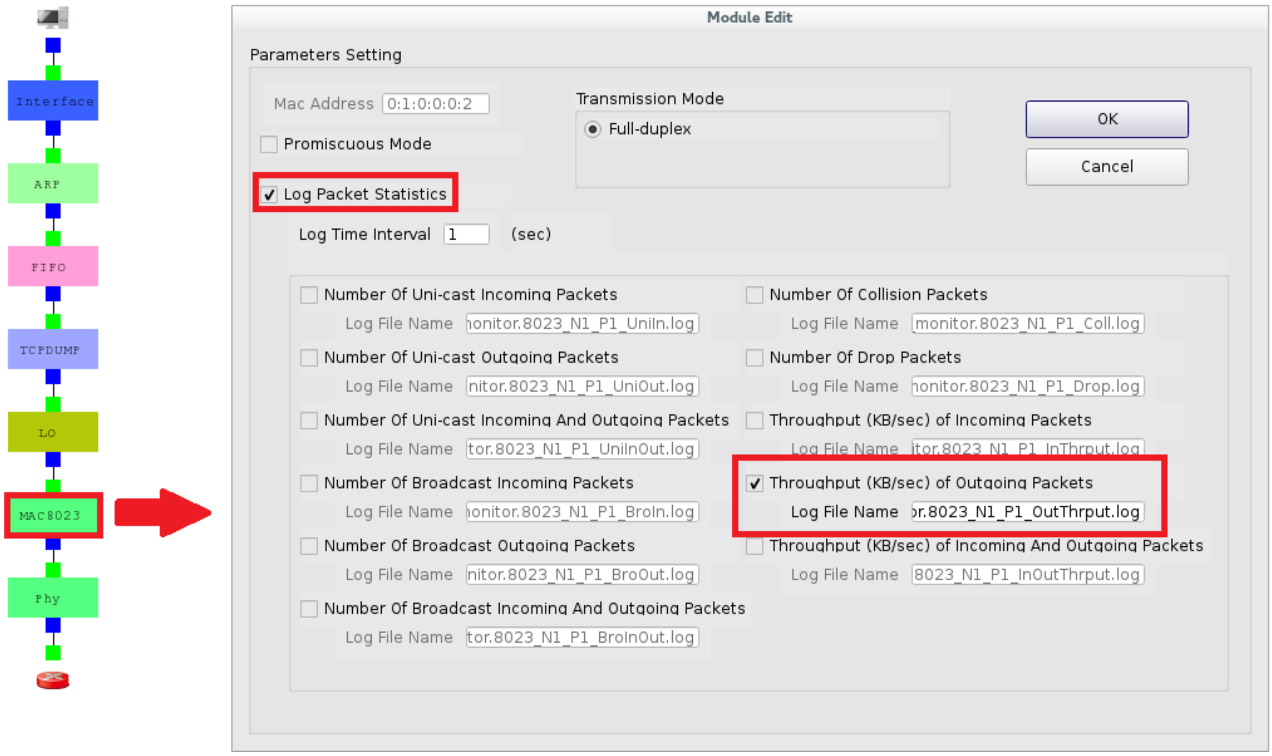


Figure 6.1.4 *Output MAC module log file of node1*

ii. Double click host node2 and add the command “ttcp -t -s -p 9001 1.0.3.1” with start time 500s under the “Application” tab.

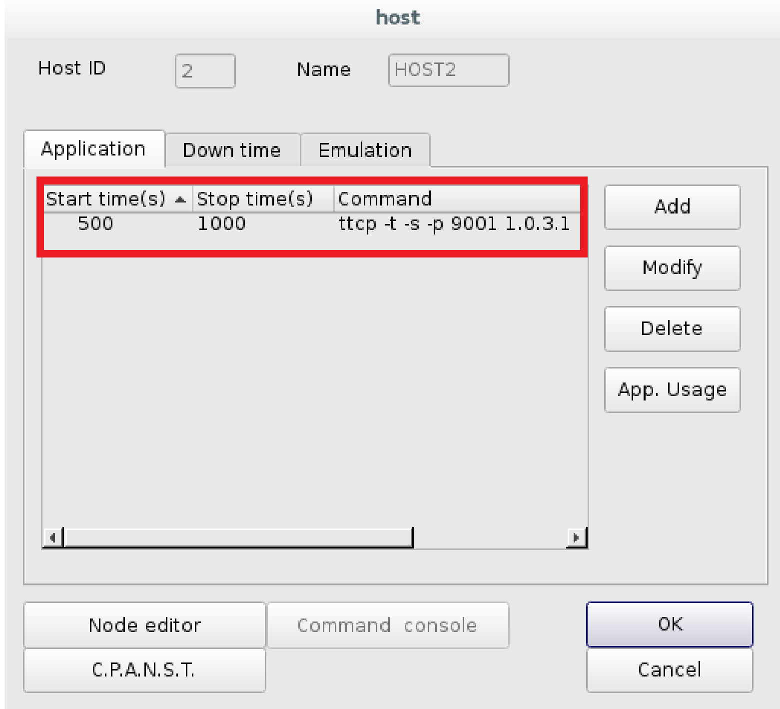
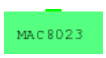


Figure 6.1.5 *Add application ttcp to node2*

The setting steps are the same as node1. Click “Node editor” button and double click module MAC8023  . It will pop up a “Module Edit” dialog. Check the parameter: “Log Packet Statistics” and select “Throughput (KB/sec) of Outgoing Packets”.

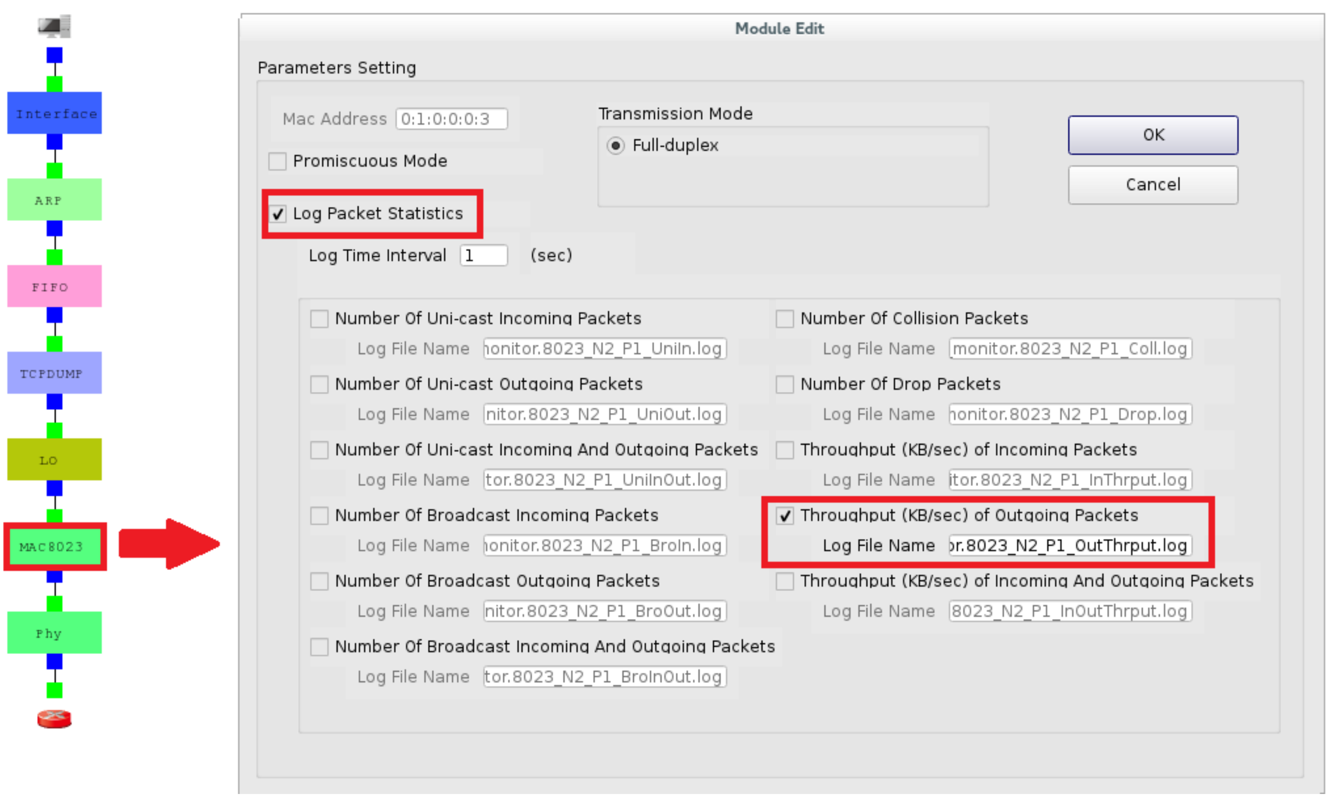


Figure 6.1.6 *Output MAC module log file of node2*

iii. Double click host node3 and add these commands: “ttcp -r -s -p 8001” and “ttcp -r -s -p 9001” under the “Application” tab.

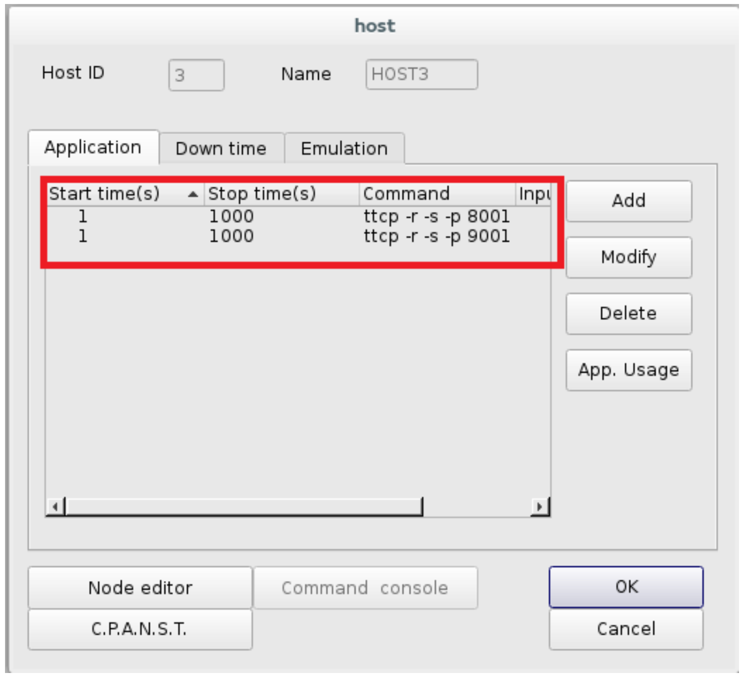


Figure 6.1.7 *Add application ttcp to node3*

(3) Run Simulation

After above settings, switch to R mode **R** . Click “ **Menu→Simulation → Run”** to start a simulation.

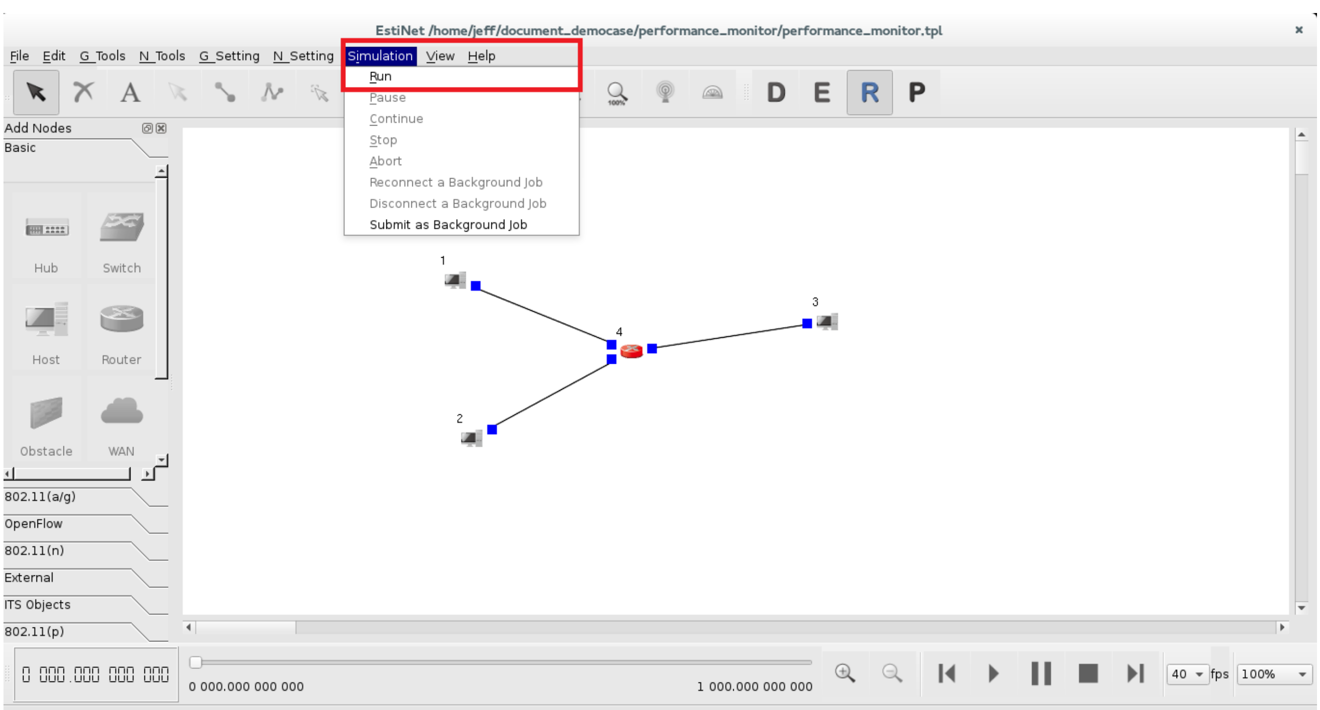


Figure 6.1.8 *Ready to run simulation*

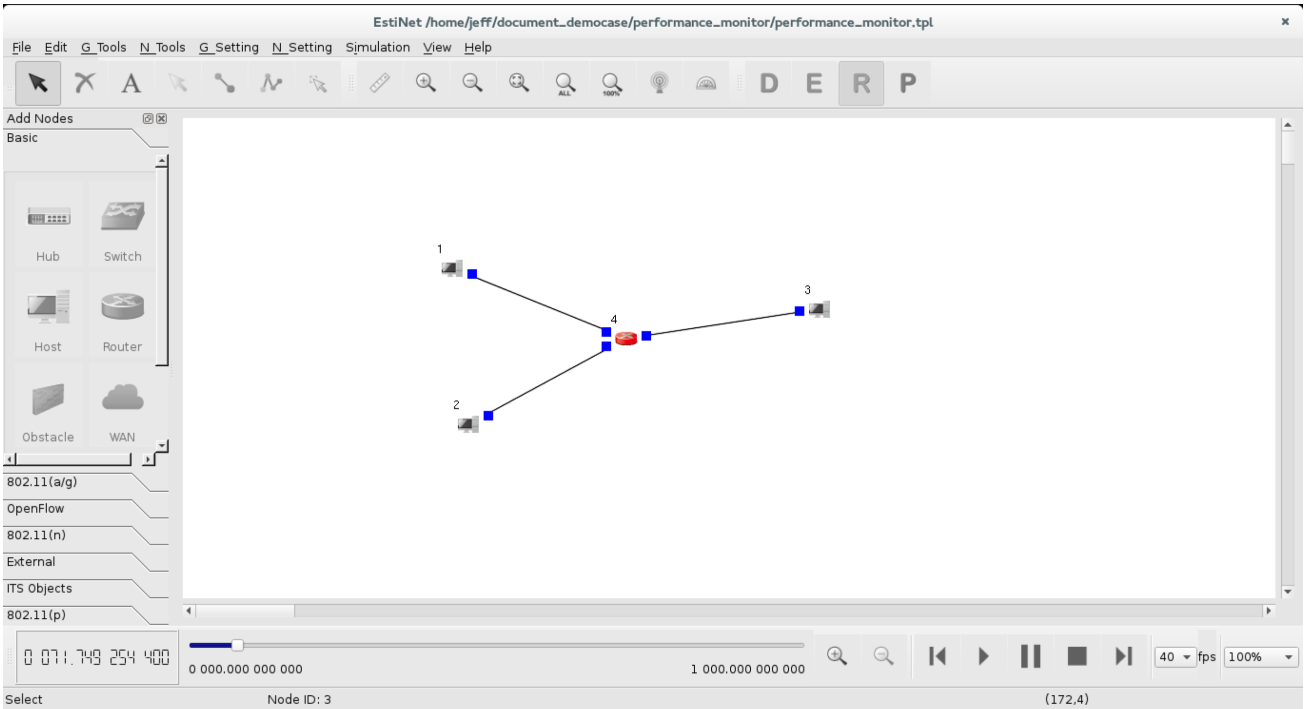


Figure 6.1.9 *Simulation is running*

(4) Playback

When simulation is done, the GUI program will get the result back automatically.

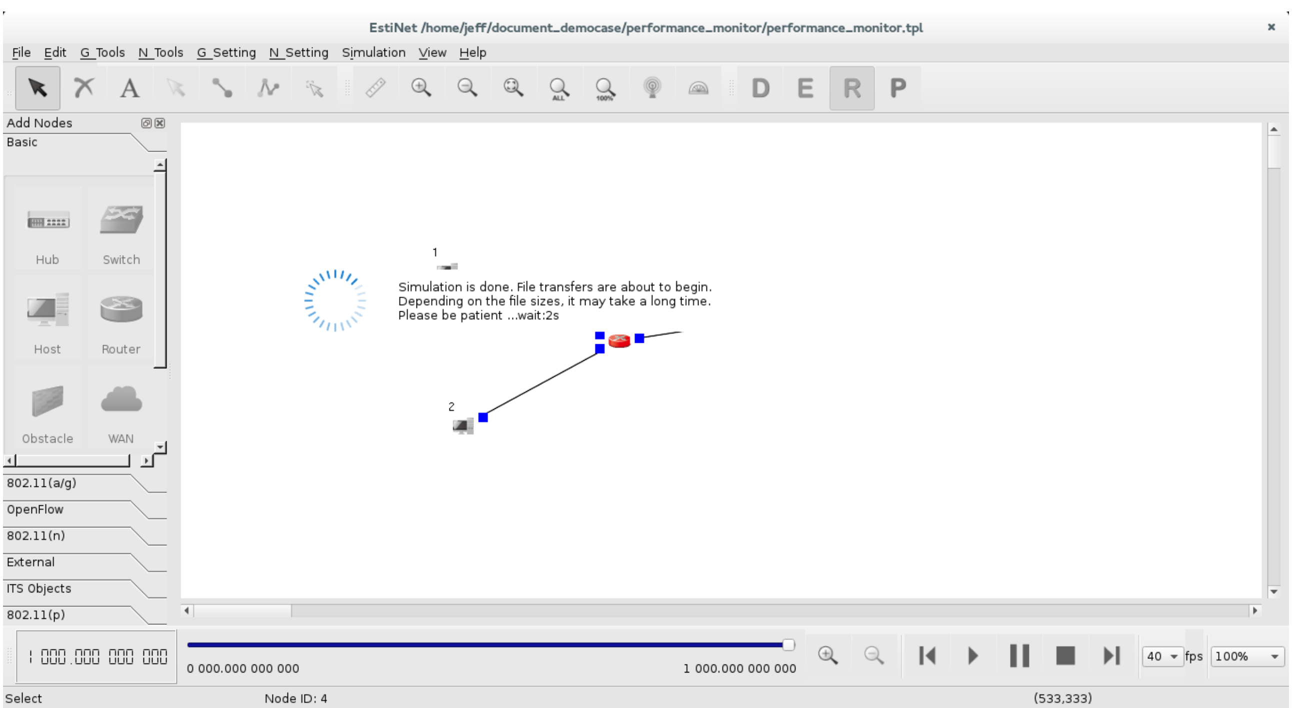


Figure 6.1.10 *Wait for getting result*

The GUI program will switch to P mode **P** after getting the result back. Then, execute "**Menu→G_Tools → Plot Graph**" to launch the performance monitor before click play button ▶ to play the result.

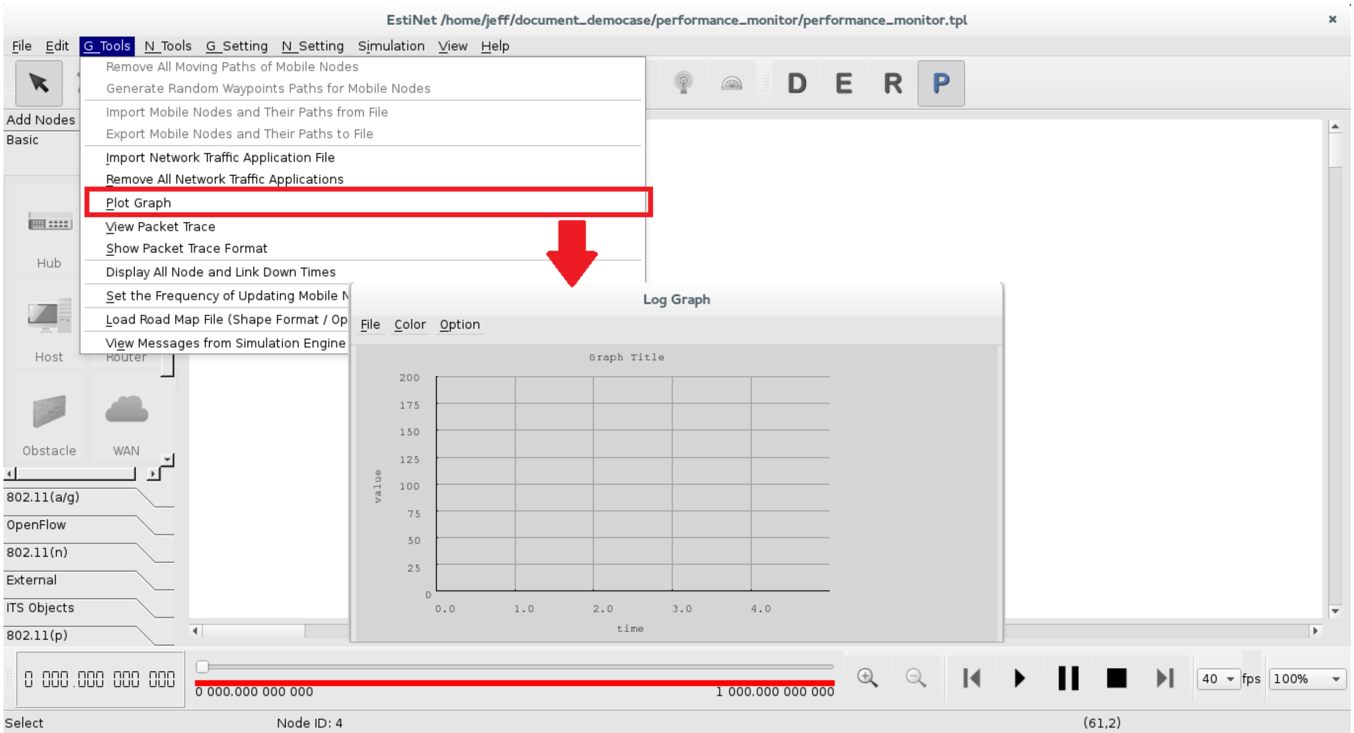


Figure 6.1.11 *Plot Graph*

In dialog “Log Graph”, execute **File→ Open** to open the throughput log files that are set before. (The user can select multiple files, but no more than six files) as figure 6.1.12. After opening files, menu will display “Graph” option, and the legend will be displayed in the right-top of the window as figure 6.1.13.

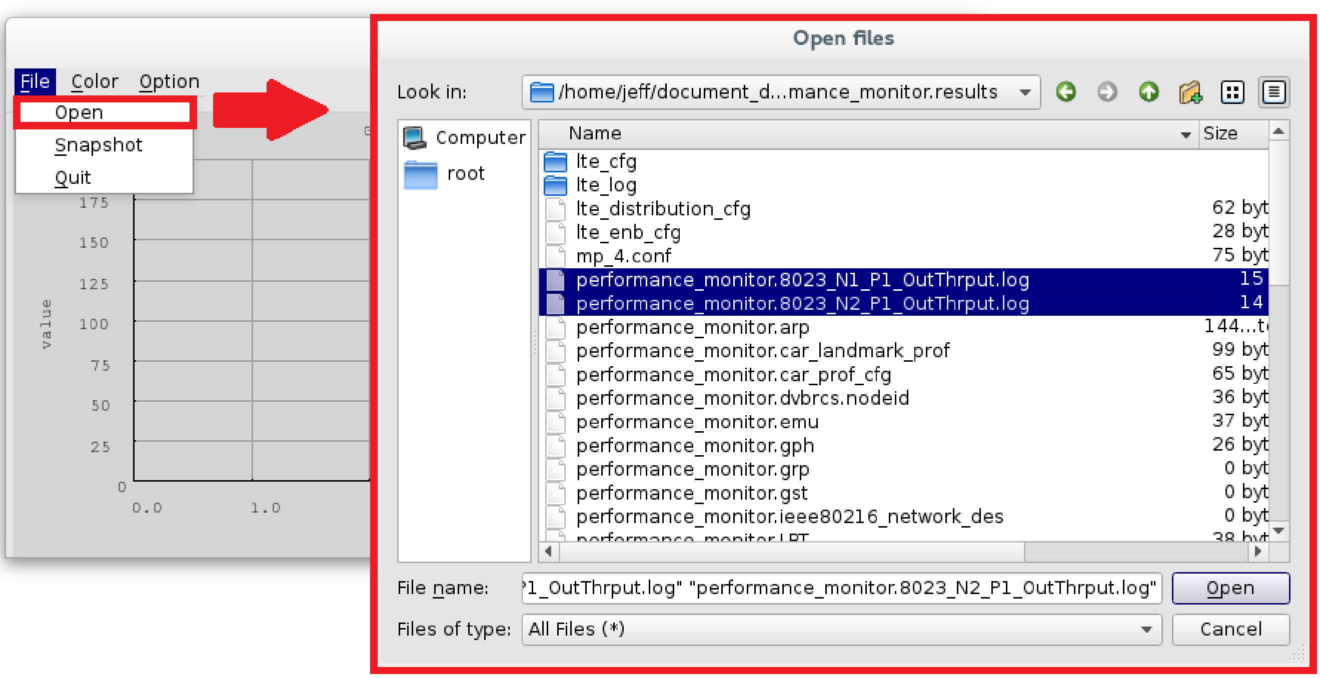


Figure 6.1.12 *Open log files*

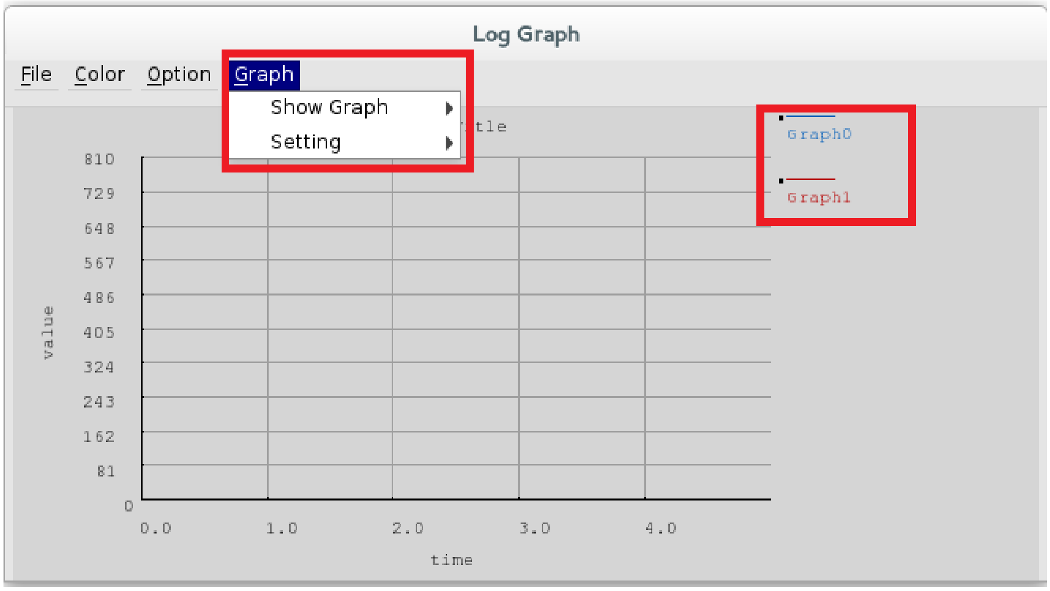


Figure 6.1.13 *Show Graph menu and hints after open log files*

Click play button ▶ to play the result in performance monitor.

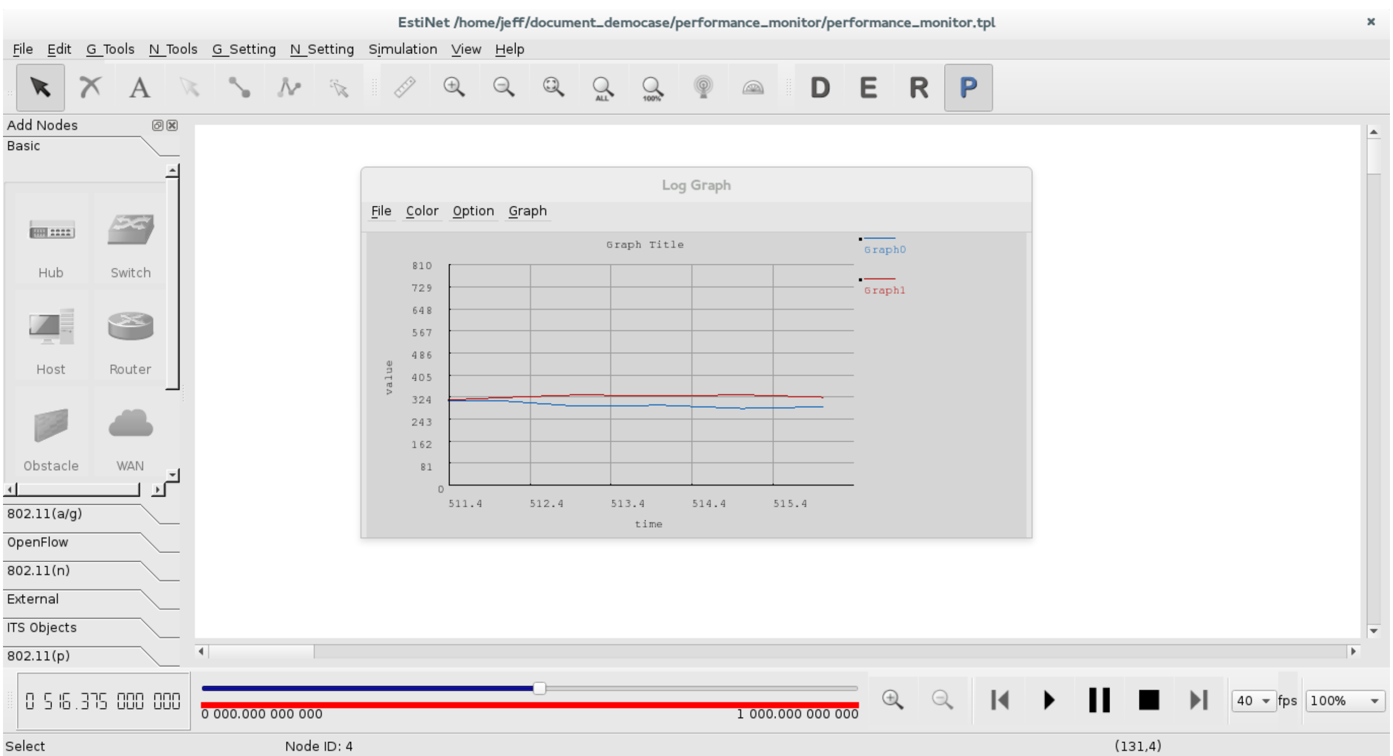


Figure 6.1.14 *Playback the result with performance monitor*

The usage of performance monitor (PM) are described as below.

File: File Menu as figure 6.1.16

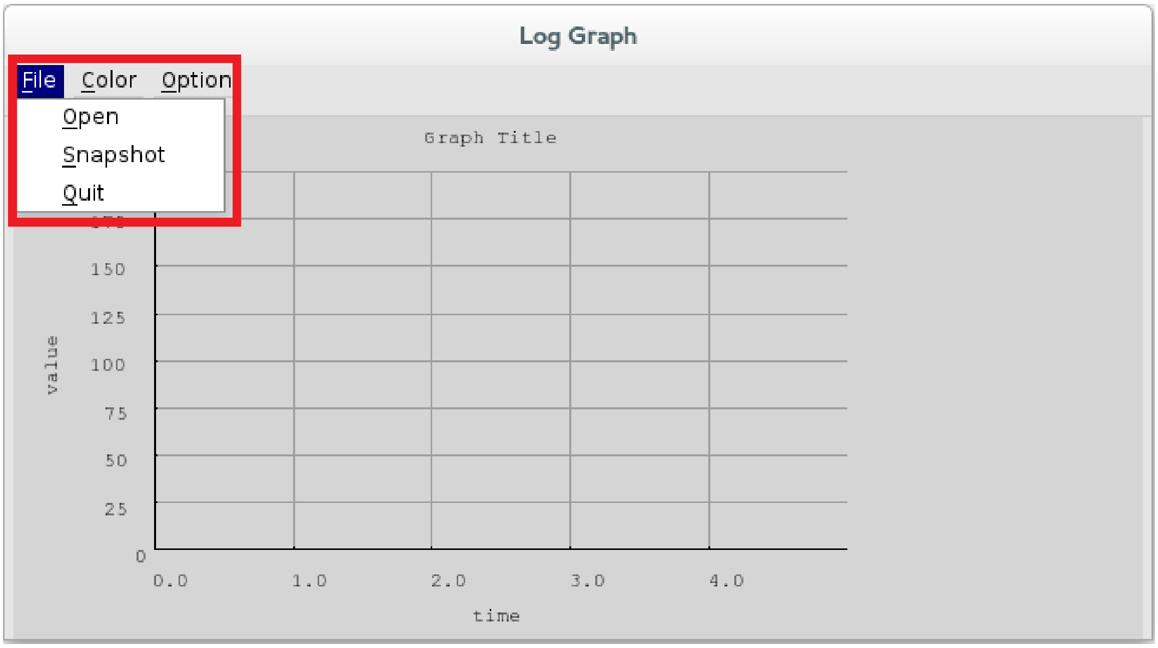


Figure 6.1.15 *File Menu*

File →Open

User could execute this command to open one to six desired log files as a source in “Log Graph”. The default log files will be generated in folder “.results”.

File →Snapshot

User could execute this command to save current performance curve. It will pop up a snapshot window as figure 6.1.16.

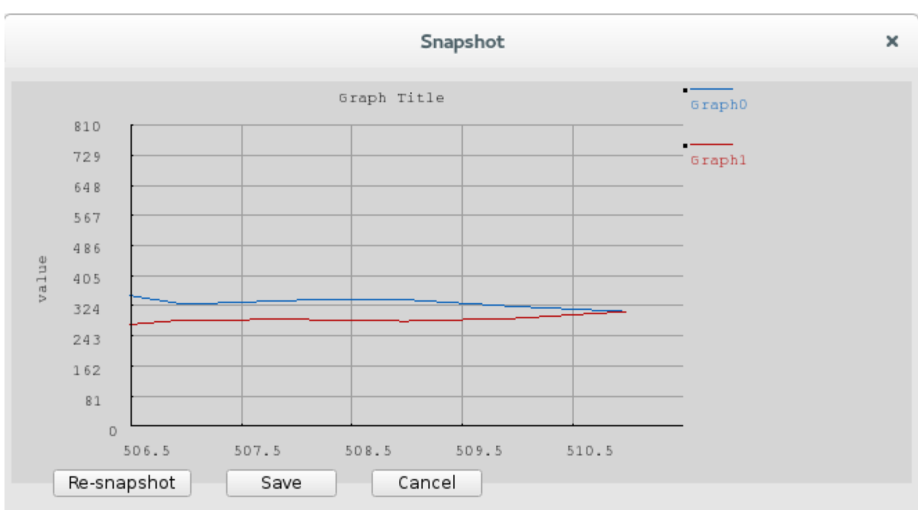


Figure 6.1.16 *Snapshot the performance monitor*

In this window, clicking button “Re-snapshot”, it will re-capture the performance curve. Clicking button “Save”, it will save the snapshot as a “png” type image file. Clicking button “Cancel”, it will quit this window.

File →Quit

User could execute this command to quit the current graph window.

Color: Color Menu as figure 6.1.17.

Several commands are provided under the sub-menu to set colors in different parts of the graph window.

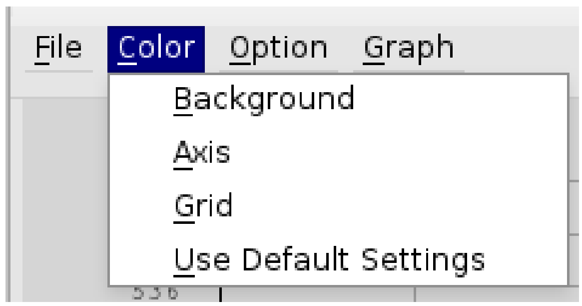


Figure 6.1.17 *Color Menu*

Color→Background

Executing this command can select a color from the palette and set it as the background color of current graph window.

Color→Axis

Executing this command can select a color from the palette and set it as the color of the X and Y axes of the current graph window.

Color→Grid

Executing this command can select a color from the palette and set it as the color of grids of the current graph window.

Color→Use Default Settings

Executing this command will use the default colors for the background, axes, and grids of the current graph window.

Option: Option Menu as figure 6.1.18.

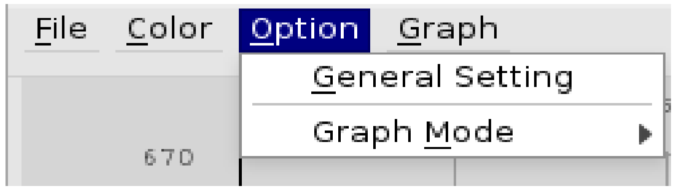


Figure 6.1.18 *Option Menu*

Option→General Setting

Executing this command can set the various presentation parameters of a graph window.

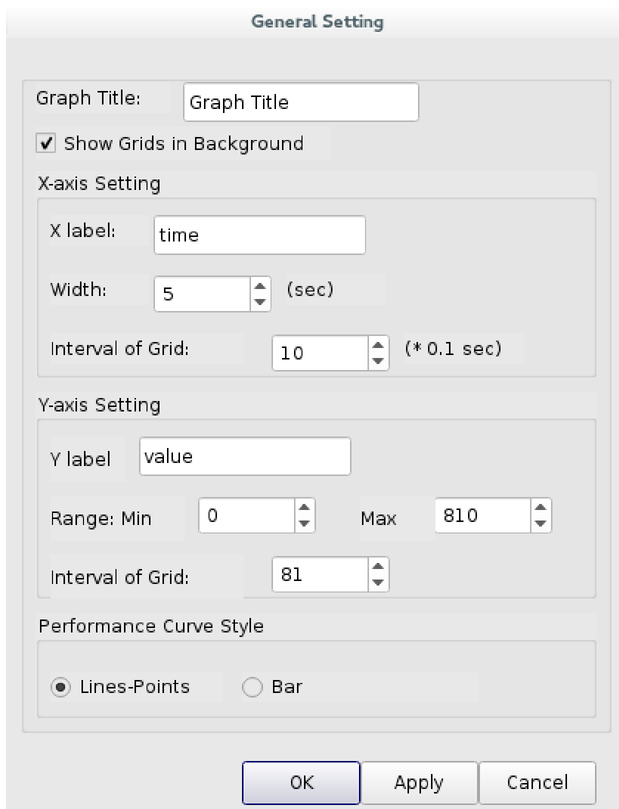


Figure 6.1.19 *General setting dialog*

Graph Title:

This field sets the title of the graph.

Show Grids in Background:

This field sets whether the grids should be visible.

X label:

This field sets the label of the X axis.

Width (X-axis):

This field specifies how many seconds' worth of data should be displayed in the graph window.

Interval of Grid (X axis):

This field sets the grid interval of the X axis. This value will be multiplied by 0.1 to derive the interval between two consecutive grids on X axis.

Y label:

This field sets the label of the Y axis.

Range - Y Min:

This field sets the minimum value of the Y axis.

Range - Y Max:

This field sets the maximum value of the Y axis.

Interval of Grid (Y axis):

This field sets the grid interval of the Y axis.

Performance Curve Style - Line-Points:

Selecting this option specifies that the performance curve should be drawn using straight lines to connect adjacent points.

Performance Curve Style - Bar:

Selecting this option specifies that the performance curve should be drawn using bar style to connect adjacent points.

In addition to the above settings, a performance curve's legend and color can be easily set by double-clicking the curve's legend located at the top-right corner. For example, a user can double-click the default "Graph 0" curve legend to change it. It as shown as the figure 6.1.23.

Option→Graph Mode

Executing this command can control how to display a performance curve. Currently there are two modes. The first mode is "Line Points" while the second mode is "Bar."

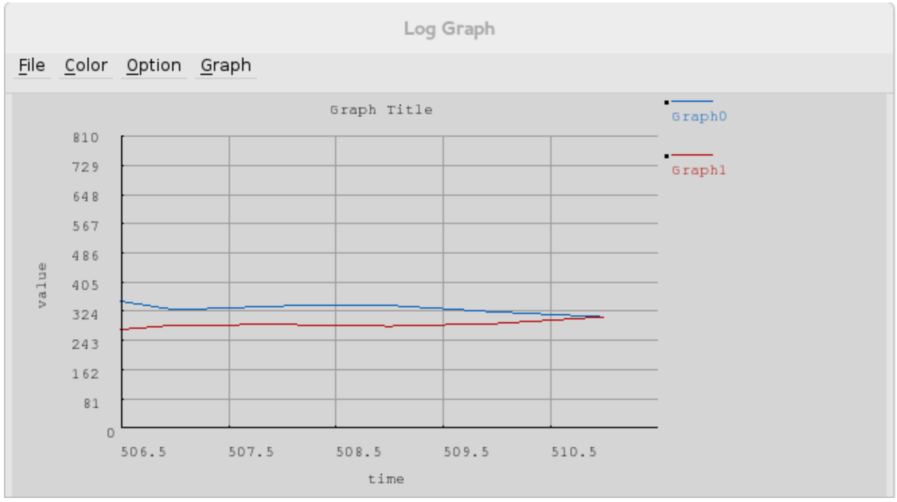


Figure 6.1.20 *The performance curve is plotted by line-point format*

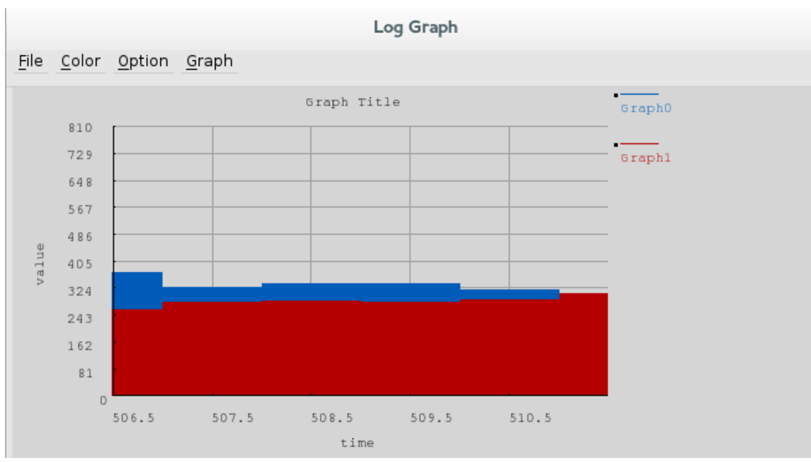


Figure 6.1.21 *The performance curve is plotted by bar format.*

Graph: Graph Menu as figure 6.1.22.

This command just shows up when you opened log files.

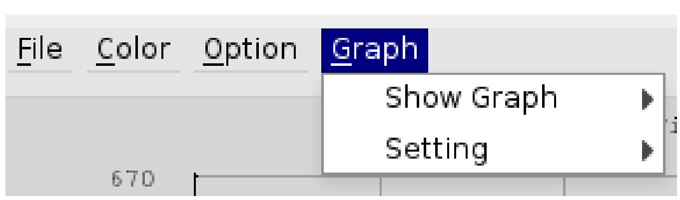


Figure 6.1.22 *Graph Menu*

Graph→Show Graph

This command provides a sub-menu for user to choose which one can be displayed its performance curve.

Graph→Setting

This provides a sub-menu for user to choose which one can be displayed its performance curve. In addition, user could press button "Choose" to select a color from the palette and set it as performance curve in current graph window.

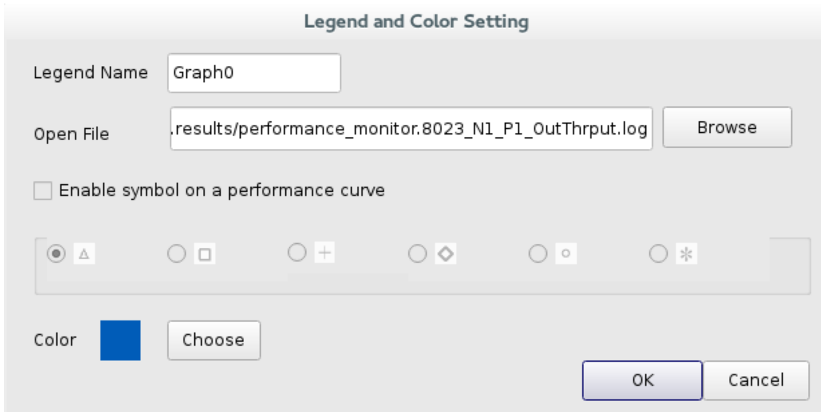


Figure 6.1.23 *The dialog of Graph setting*

Legend Name

This field sets the label of the graph.

Open File

User could change the source file from here.

Enable symbol on a performance curve

Enable this option to select one symbol and the performance curve will be drawn by selected symbol.

Color

User could select a color from the palette and set it as the performance curve in current graph window.

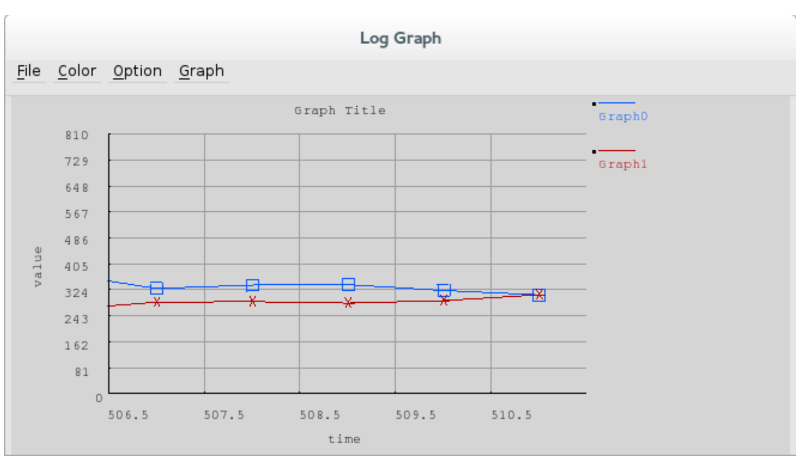


Figure 6.1.24 *Draw symbol on performance curves*

Appendix A

Installation and Configuration

In the package provided by EstiNet Technologies Inc., the user can see the installation script, named install.sh, and a RPMS directory. To install the package, the user should execute the installation script and answer some optional prompts. More details about installation are in “EstiNet_9.0_InstallationGuide”, user can read it to get more information. After installation, a directory named “estinet” will be created under the /usr/local/ directory. In addition, there are some subdirectories under /usr/local/estinet, such as “bin”, “etc”, “tools”, “BMP”, and “lib”, etc.. Each subdirectory stores different type of files for different purpose. Each one is explained as below.

```
root@localhost:/usr/local/estinet
File  Edit  View  Search  Terminal  Help
[root@localhost estinet]# pwd
/usr/local/estinet
[root@localhost estinet]# ls
bin  BMP  etc  insts.sh  lib  license_key.txt  result.log  tools
[root@localhost estinet]# 
```

The subdirectories under /usr/local/estinet

1. /usr/local/estinet/bin

This directory stores executable programs such as GUI program, dispatcher, coordinator, and the simulation engine. Their file names are “estinetgui”, “dispatcher”, “coordinator”, and “estinetse”, respectively.

```
root@localhost:/usr/local/estinet/bin
File  Edit  View  Search  Terminal  Help
[root@localhost estinet]# pwd
/usr/local/estinet
[root@localhost estinet]# ls
bin  BMP  etc  insts.sh  lib  license_key.txt  result.log  tools
[root@localhost estinet]# cd bin/
[root@localhost bin]# ls
cancel                                estinetgui      inputkey        qt.conf
coordinator                           estinetgui.bin  iptables-check  renew_usbkey
dispatcher                            estinetse       kill_tun        reserve
dispatcher_coordinator_estinetgui     estinetxterm    printPtr
[root@localhost bin]# 
```

The programs in /usr/local/estinet/bin

2. /usr/local/estinet/tools

This directory stores executable programs of various applications and tools pre-installed by EstiNet. For example, currently “stcp,” “rtcp,” “ttcp,” “tcpdump,” “ospfd,” “estinettcsh,” “script,” “stg,” “rtg,” “ifconfig,” and “ping” are supported. Some daemon programs used by EstiNet are also stored in this directory. For example, the daemon program used for emulation is stored here. These car agent programs can be run on mobile nodes to control the moving behavior of mobile nodes.

Due to the use of a novel kernel-reentering simulation methodology, EstiNet has two advantages as follows: (1) Any real-life application program can be run on a simulated network to generate traffic and (2) Their performance can be evaluated under different simulated network conditions.

Thus the real-life application programs pre-installed in this directory represent only a very small subset of real-life application programs that can be used with EstiNet.

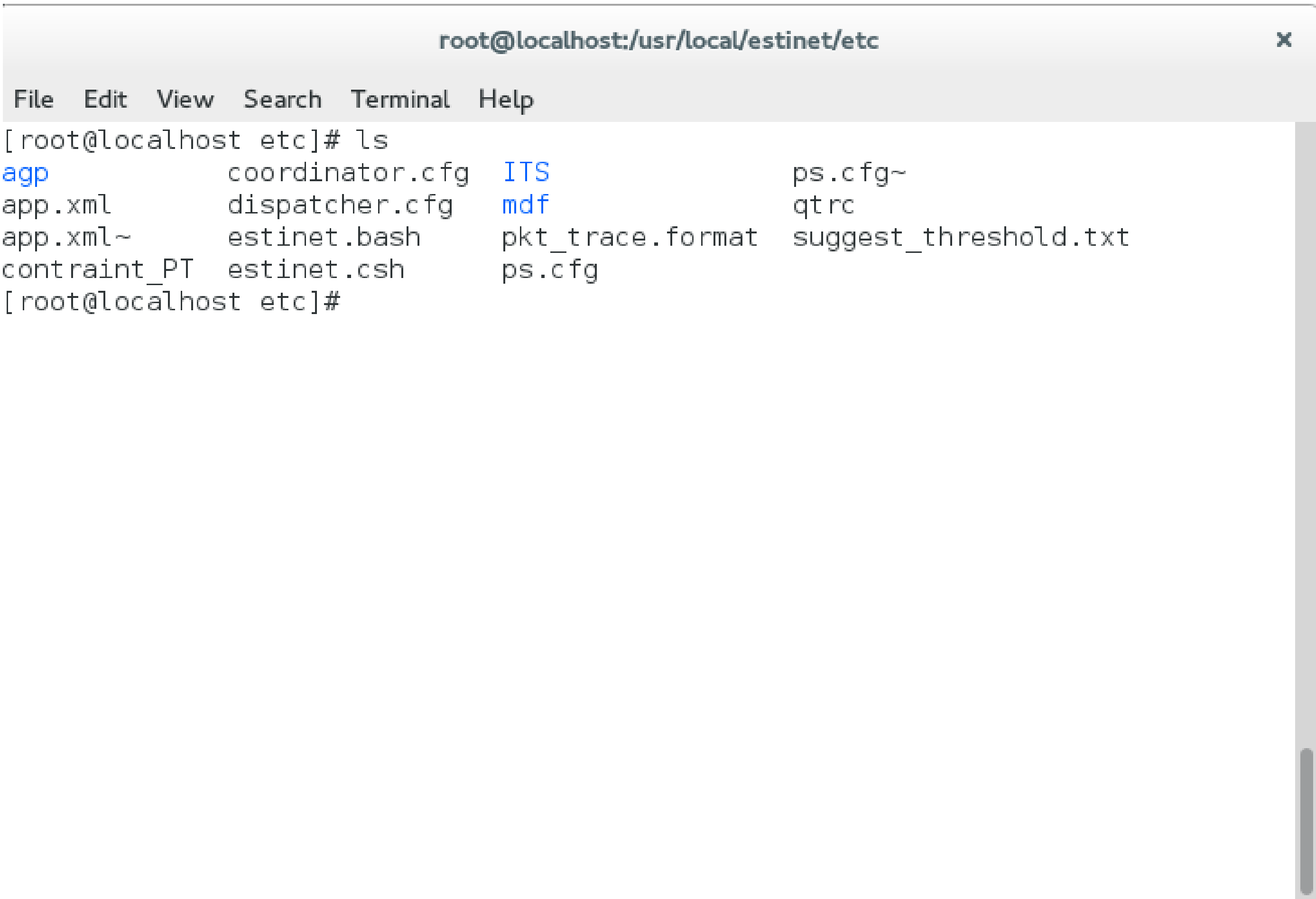
During simulation, if a user wants the simulation engine to run up an application program that is not pre-installed in this subdirectory (e.g., the P2P BitTorrent program), the user must first copy that program into this subdirectory (i.e., /usr/local/estinet/tools), so that the simulation engine can find it during simulation.

```
root@localhost:/usr/local/estinet/tools
File  Edit  View  Search  Terminal  Help
[root@localhost tools]# ls
adapt_bw                          Magent2    README          stg
adapt_sessionbw                   Magent3    reset           STYLE
bt_client                         Magent4    rip             sync
bt_tracker                        Magent5-c  route           tcpdump
CarAgent                          Magent5-t  RSUAgent        tests
Detailed_Testing_Methodology.txt  nox        rtcp            tools
doc                               nox_core   rtg             ttcp
estinettcsh                       nox.info   rtprecvonly     udpc
etc                               oft        rtpsendrecv     udps
id                                oft.log    run_switch.py   WSM
ifconfig                          ospfd      script          WSM_Forwarding
java                              ping       SignalAgent1
LICENSE                           platforms  src
Magent1                           profiles   stcp
[root@localhost tools]#
```

The programs in /usr/local/estinet/tools

3. /usr/local/estinet/etc

This directory stores the configuration files needed by the dispatcher and coordinator programs. Their names are “dispatcher.cfg,” and “coordinator.cfg,” respectively. Some other configuration files used by EstiNet are also stored here. For example, the “app.xml,” which is read by the GUI program to explain the usages of pre-installed application programs, is also stored here. An “mdf” subdirectory (which stands for “module description file”) is created here. Inside this directory, the parameter definitions and dialog box layout design of supported protocol modules are stored in separate sub-directories. The GUI program will read the files inside the “mdf” directory to know the definition of supported protocol modules. The “ps.cfg” file describes the default internal protocol stack used by each supported network node.



The configuration files in /usr/local/estinet/etc

4. /usr/local/estinet/BMP

This directory stores the icon picture files used by the GUI program. These icon files are used for displaying various devices’ icons and control buttons.

```
root@localhost:/usr/local/estinet/BMP
File  Edit  View  Search  Terminal  Help
d-click.png                  multiport_adhoc.bmp          switch.bmp
d.png                        NCC.bmp                      switch.png
e.bmp                        next.png                     traffic_light.png
e_click.bmp                  next.xpm                     triangle.bmp
e-click.png                  obs_opt_switch.bmp           undo.png
e.png                        opflow_controller.bmp        undo.xpm
estinetlogo80.png            opflow_controller.png        wall.bmp
estinetlogo8.1.png           opflow_gw.bmp                wall.png
EstiNet_logo_9.png           opflow_gw.png                w.bmp
estinetlogo.png              opflow_switch131.bmp         zero.bmp
estinetpeople.bmp            opflow_switch131.png         zoomall.png
ext_car_80211p.png           opflow_switch13_inband.png   zoomblock.png
ext_opflow_controller.bmp    opflow_switch.bmp            zoomblock.xpm
ext_opflow_controller.png    opflow_switch.png            zoomin2.bmp
Feeder.bmp                   opt_path.bmp                 zoomin.bmp
flag.png                     opt_ring.bmp                 zoomin.png
form_group.png               opt_switch.bmp               zoomone.bmp
form_subnet.png              pause.png                    zoomone.png
form_subnet.xpm              pause.xpm                    zoomout2.bmp
Gateway.bmp                  p.bmp                        zoomout.bmp
ggsn.bmp                     p_click.bmp                  zoomout.png
gprs_switch.bmp              p-click.png                  zoomout.xpm
GS.bmp                       play.png
[root@localhost BMP]#
```

The icon picture files in /usr/local/estinet/BMP

5. /usr/local/estinet/lib

This directory stores the libraries used by the simulation engine and GUI.

```
root@localhost:/usr/local/estinet/lib
File  Edit  View  Search  Terminal  Help
libQt5Multimedia.so                 libQt5X11Extras.la
libQt5Multimedia.so.5               libQt5X11Extras.prl
libQt5Multimedia.so.5.3             libQt5X11Extras.so
libQt5Multimedia.so.5.3.0           libQt5X11Extras.so.5
libQt5MultimediaWidgets.la          libQt5X11Extras.so.5.3
libQt5MultimediaWidgets.prl         libQt5X11Extras.so.5.3.0
libQt5MultimediaWidgets.so          libQt5Xml.la
libQt5MultimediaWidgets.so.5        libQt5XmlPatterns.la
libQt5MultimediaWidgets.so.5.3      libQt5XmlPatterns.prl
libQt5MultimediaWidgets.so.5.3.0    libQt5XmlPatterns.so
libQt5Network.la                    libQt5XmlPatterns.so.5
libQt5Network.prl                   libQt5XmlPatterns.so.5.3
libQt5Network.so                    libQt5XmlPatterns.so.5.3.0
libQt5Network.so.5                  libQt5Xml.prl
libQt5Network.so.5.3                libQt5Xml.so
libQt5Network.so.5.3.0              libQt5Xml.so.5
libQt5Nfc.la                        libQt5Xml.so.5.3
libQt5Nfc.prl                       libQt5Xml.so.5.3.0
libQt5Nfc.so                        libxcb-static.a
libQt5Nfc.so.5                      libxcb-static.prl
libQt5Nfc.so.5.3                    pkgconfig
libQt5Nfc.so.5.3.0                  plugins
libQt5OpenGLExtensions.a
[root@localhost lib]#
```

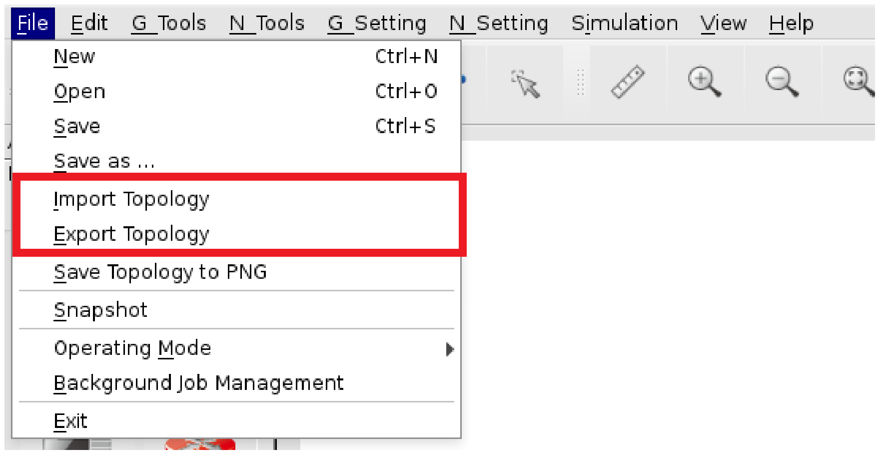
The libraries in /usr/local/estinet/lib

After the installation is finished, the machine must be rebooted and then the user must choose the EstiNet kernel to boot. After the machine boots up with the EstiNet kernel, the whole installation can be considered successful.

Appendix B

Import Topology and Export Topology

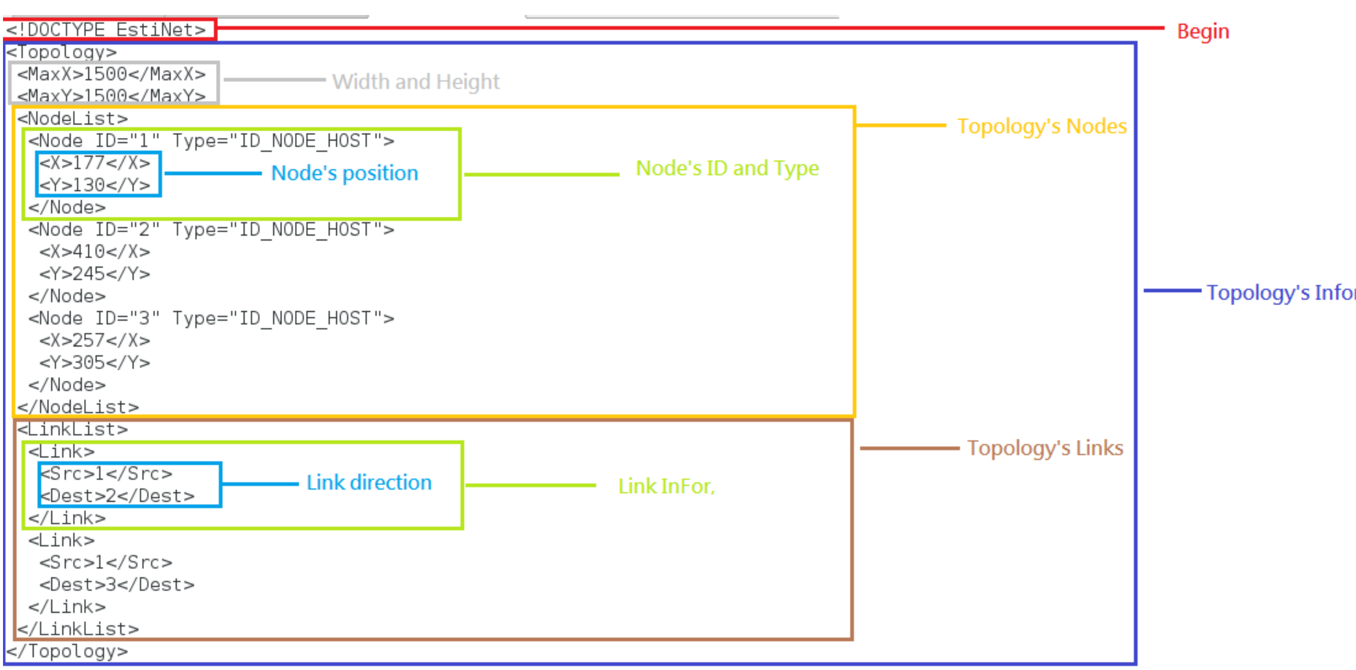
EstiNet GUI provide a method to import and export topology files for user to create a larger topology more quickly. User only need to follow the format of STPL (Simple Topology) to create a topology file by any programming language. Using **"Menu→File→Import Topology"** to load user's topology such as "xxx.stpl" and doing related setting for simulation. Please notice that it don't support "VANET" and "Multi-interface Mobile Nodes" in this function.



Import and Export Topology functions

File Format of STPL

The file format of STPL is similar to XML. A sample file of STPL is as below:



The format of STPL file

The description of STPL:

<Topology>: It includes all information in topology, such as Nodes, Links, width and height. And it must be ended by </Topology>

<MaxX>: The width of a topology. And it must be ended by </MaxX>

<MaxY>: The height of a topology. And it must be ended by </MaxY>

<NodeList>: It records all Nodes in a topology. And it must be ended by</NodeList>

<Node>: It records the ID and Type of Node. And it must be ended by</Node>

<X>: The x-coordinate of Node. And it must be ended by</X>

<Y>: The y-coordinate of Node. And it must be ended by</Y>

<LinkList>: It records all link information in a topology. And it must be ended by</ LinkList >

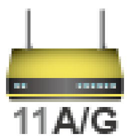
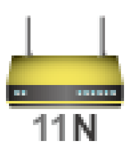
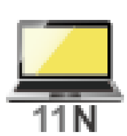
<Link>: It records the link information between two Nodes. And it must be ended by</ Link >

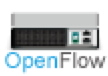
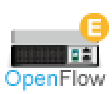
<Src>: The source NodeID of a linkage. And it must be ended by</ Src >

<Dest>: The destination NodeID of a linkage. And it must be ended by</ Dest >

A List of Node Type which supported in STPL.

Node	**Node Type**
 Host	ID_NODE_HOST
 Hub	ID_NODE_HUB
 Switch	ID_NODE_SWITCH

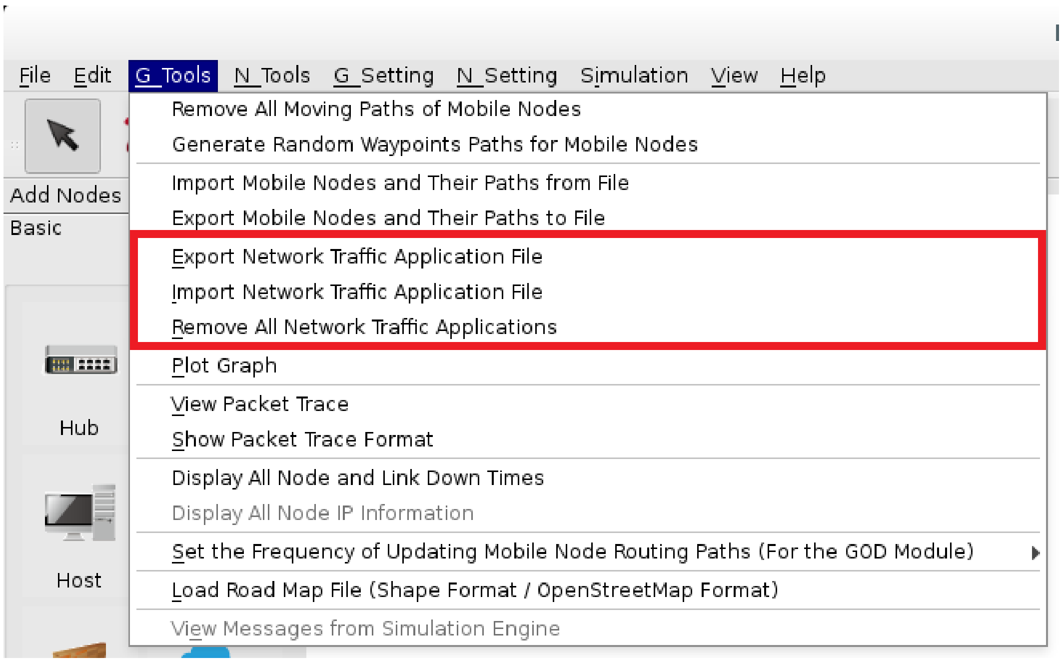
 Router	ID_NODE_ROUTER
 Wan	ID_NODE_WAN
 802.11(a/g) AP	ID_NODE_80211A_AP
 802.11(a/g) Adhoc	ID_NODE_80211A_ADHOC
 802.11(a/g) Infra	ID_NODE_80211A_INFRA
 802.11(n) AP	ID_NODE_80211N_AP
 802.11(n) Adhoc	ID_NODE_80211N_ADHOC
 802.11(n) Infra	ID_NODE_80211N_INFRA
 80211(p) RSU	ID_NODE_80211P_RSU

 External Host	ID_NODE_EXTHOST
 External 802.11(a/g) Adhoc	ID_NODE_MOBILE_EXT
 External 802.11(a/g) Infra	ID_NODE_MOBILE_INFRA_EXT
 OpenFlow Switch V1.0	ID_NODE_OPFLOW_SWITCH
 OpenFlow Switch V1.3	ID_NODE_OPFLOW_SWITCH_V13
 OpenFlow Controller Switch	ID_NODE_OPFLOW_GW
 OpenFlow Controller	ID_NODE_OPFLOW_CONTROLLER
 OpenFlow External Controller	ID_NODE_EXT_OPFLOW_CONTROLLER

Appendix C

Import/Export Network Traffic Application File

EstiNet GUI provide a method to import and export traffic application file for user to set all network traffic application more quickly. Using "**Menu→G_Tools→Exmport Network Traffic Application File**", "**Menu→G_Tools→Import Network Traffic Application File**" and "**Menu→G_Tools→Remove All Network Traffic Applications**" to execute this function.

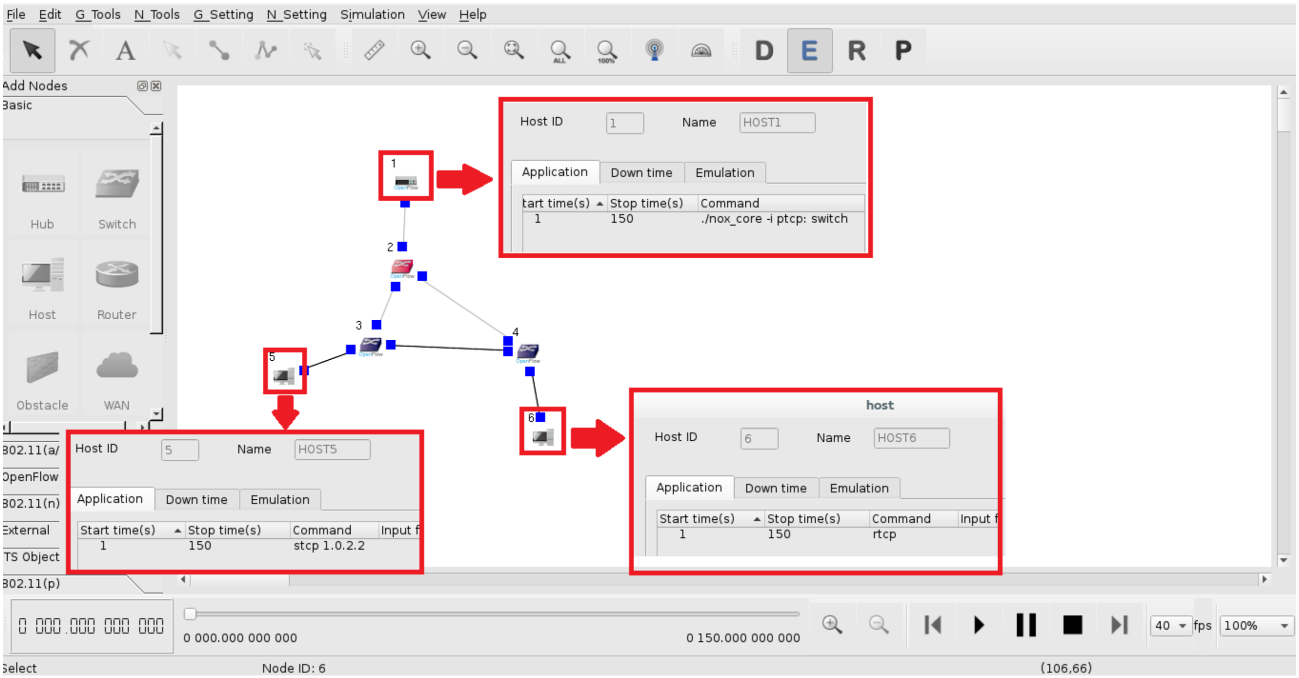


Export and Import Network Traffic function

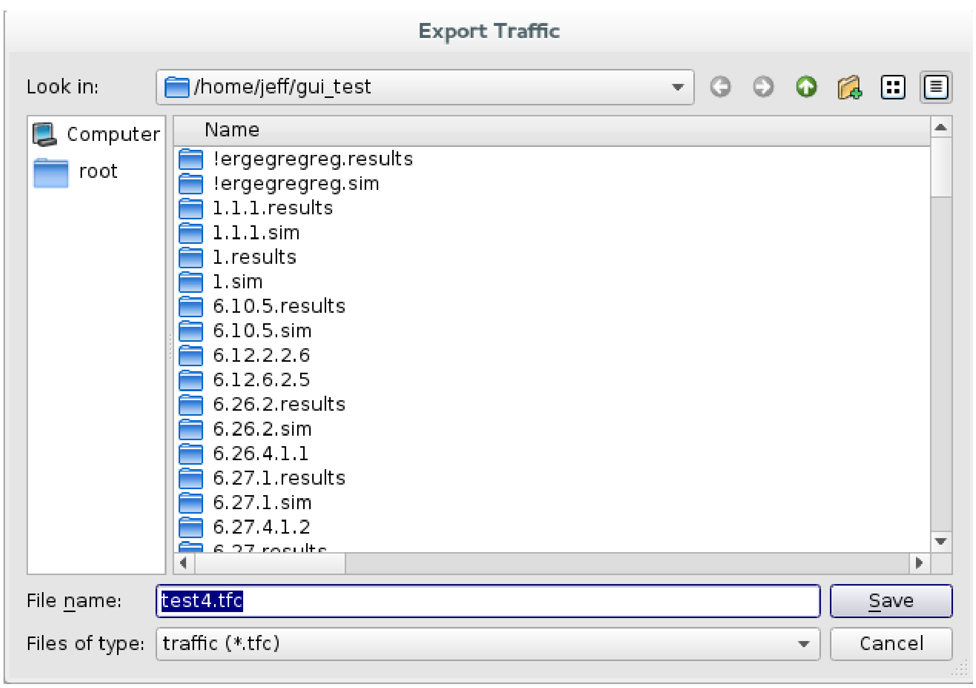
The usage of these three network traffic application functions is as below:

1. **Export Network Traffic Application File**

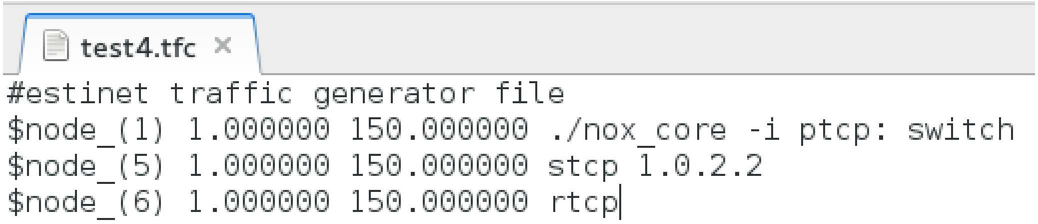
After complete all settings of network traffic application as below figure, please click "**Menu→G_Tools→Exmport Network Traffic Application File**".



It will pop up a dialog for user to export a TFC (Traffic) file in the directory which user selected. Then click button “Save” to execute it. The output file such as “xxx.tfc”.

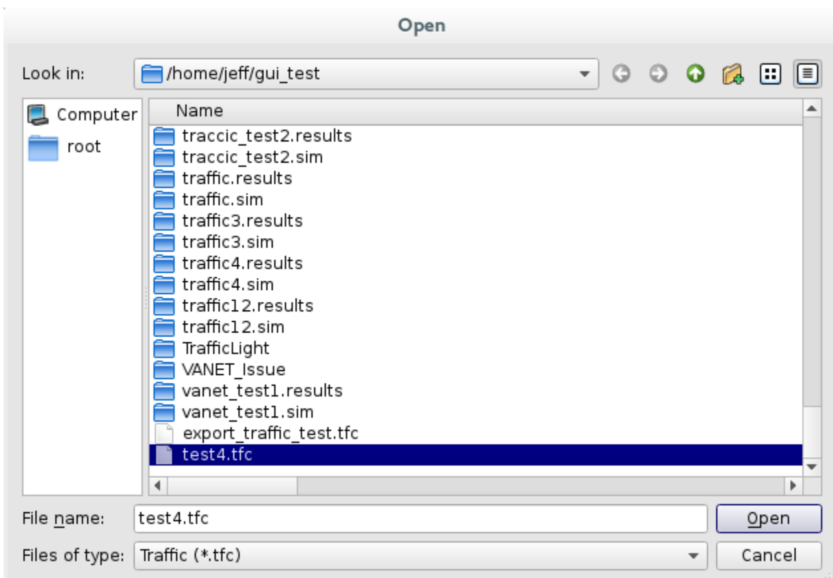


The format of file TFC is “$node_(Node ID) Start time(s) Stop time(s) Command” as below figure.

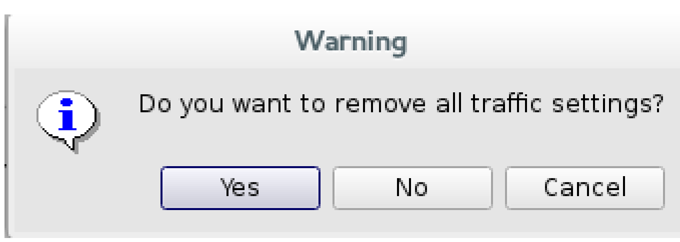


2. Import Network Traffic Application File

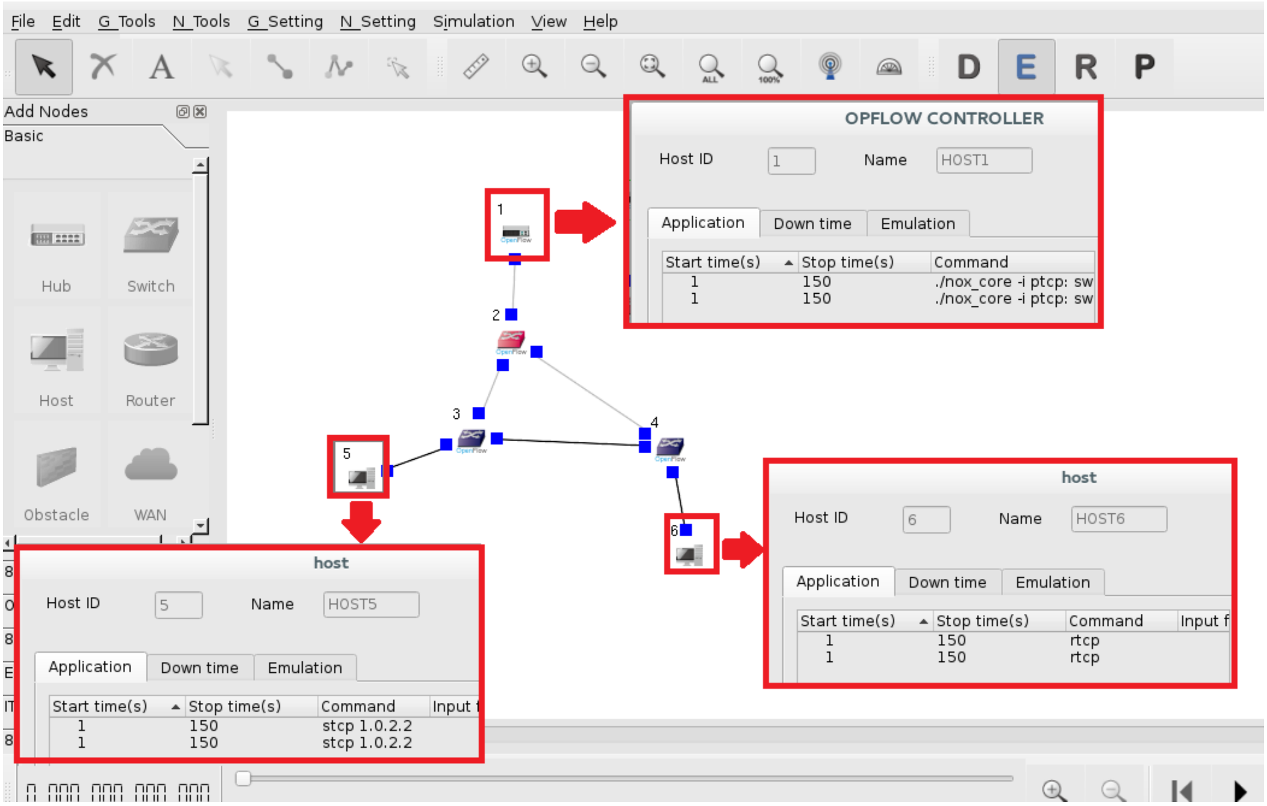
Please click **"Menu→G_Tools->Import Network Traffic Application File"** to import a network traffic application file. It will pop up a dialog for user to select a TFC file. After file is selected, click button "Open" to import file.



The GUI will pop up another dialog to ask user "Do you want to remove all traffic settings?" It means, if user clicks button "Yes", it will remove all original traffic settings in a topology.

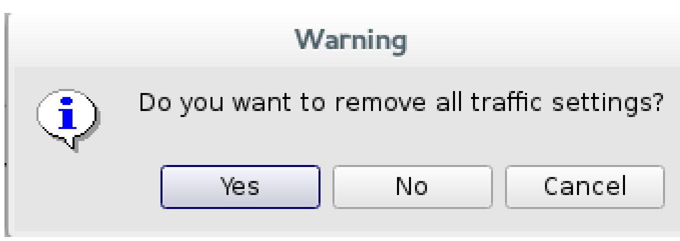


If user clicks button "No", it will keep all original traffic settings and load the import tfc file traffic settings as below figure.

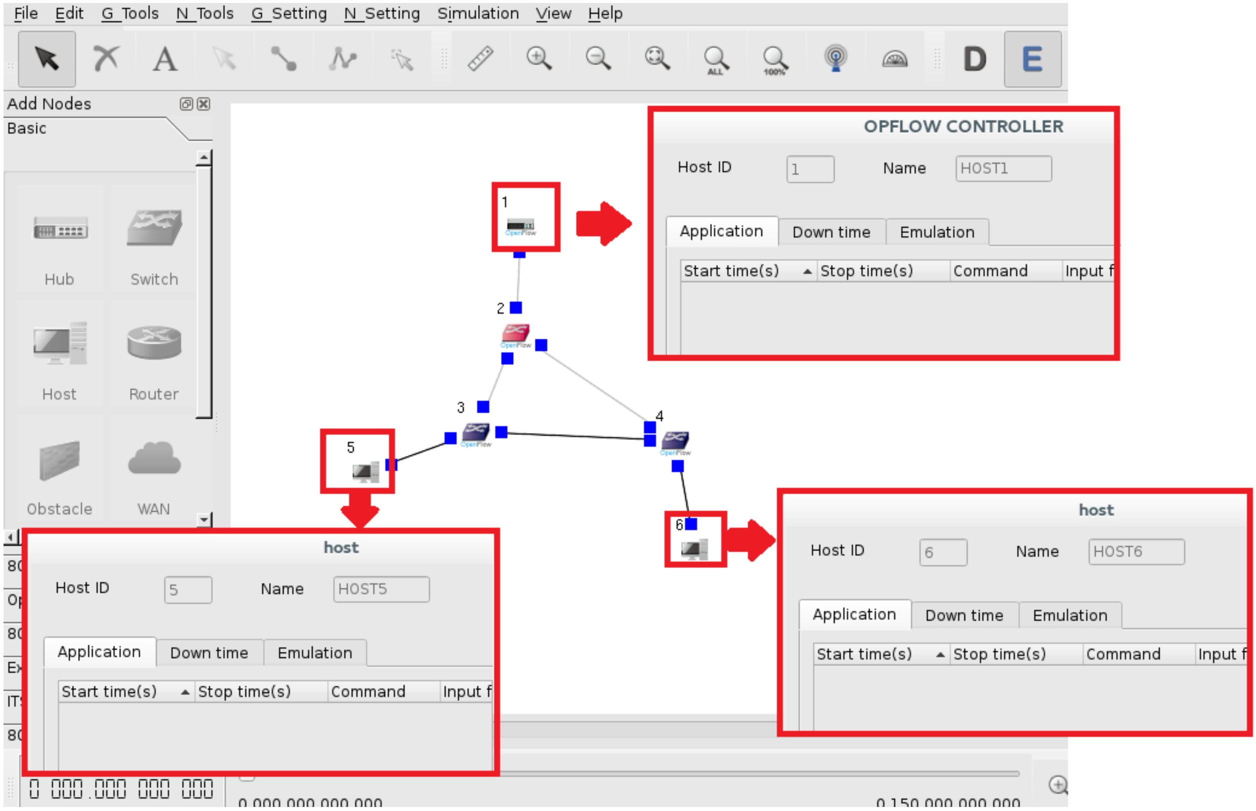


3. Remove All Network Traffic Applications

Please click "**Menu→G_Tools->Remove All Network Traffic Applications**" to remove all traffic settings in a topology. The GUI will pop up a dialog to ask user "Do you want to remove all traffic settings?"

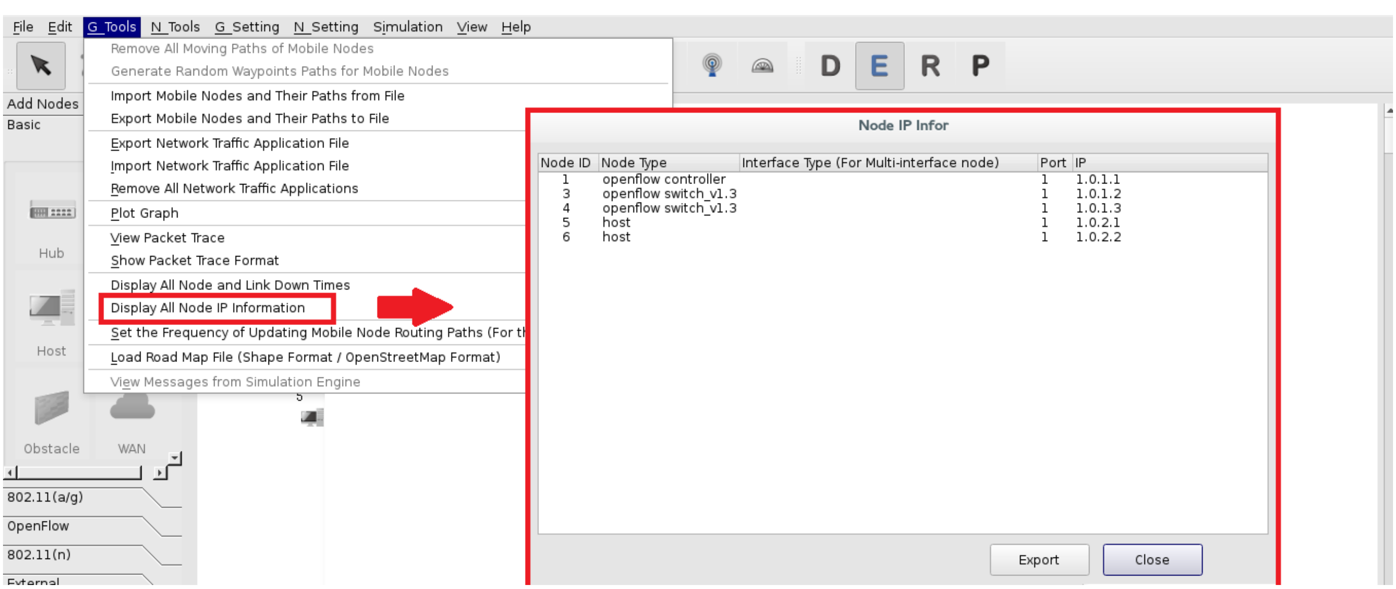


It means, if user click button "Yes", it will remove all traffic settings in a topology as below figure.



A list for all IP information per each Node ID

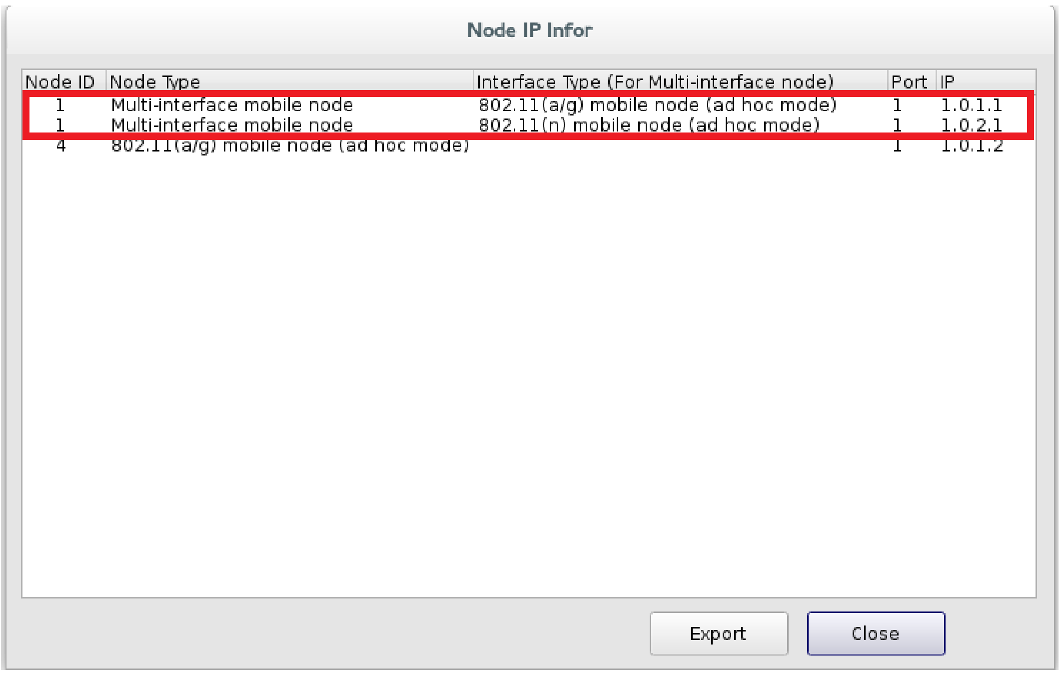
GUI provide a method for user to check all IP information in a topology. Please click "**Menu→G_Tools->Display All Node IP Information**", it will pop up a dialog to list all IP information. If user want to export these IP information into a file, just click button "Export" which export into a user defined file name such as "xxx.ip".



test4.ip

```
#estinet node ip information generator file
#format:node_id node_type interface_type port_id ip
1 openflow controller  1 1.0.1.1
3 openflow switch_v1.3  1 1.0.1.2
4 openflow switch_v1.3  1 1.0.1.3
5 host  1 1.0.2.1
6 host  1 1.0.2.2
```

If the node type belongs to multi-interface, it will list all interfaces in the field of “Interface Type”.

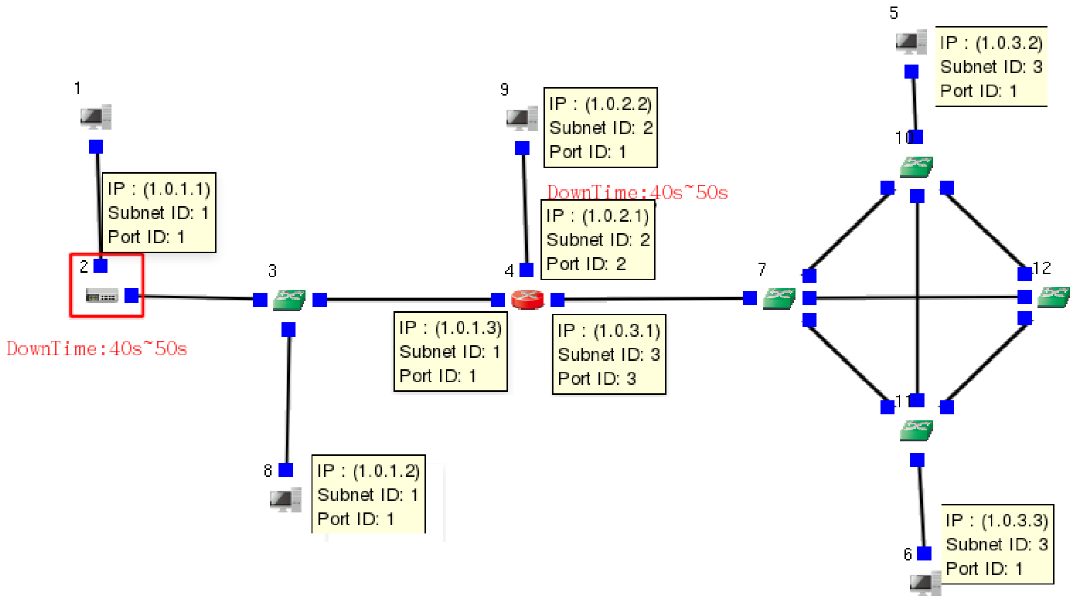


Note: All IP are assigned by GUI program automatically. When topology is changed, the IP will be reassigned. Please confirm IP information in network traffic application after import “Network Traffic Application File”.

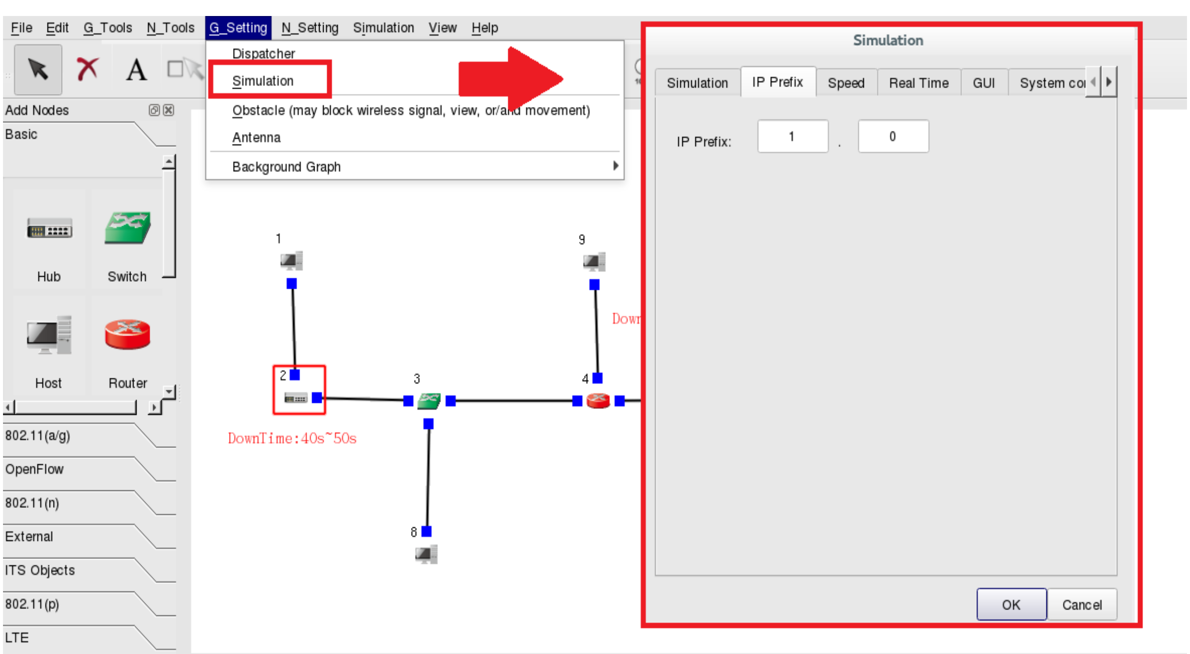
Appendix D

IP Prefix could be changed by user's settings

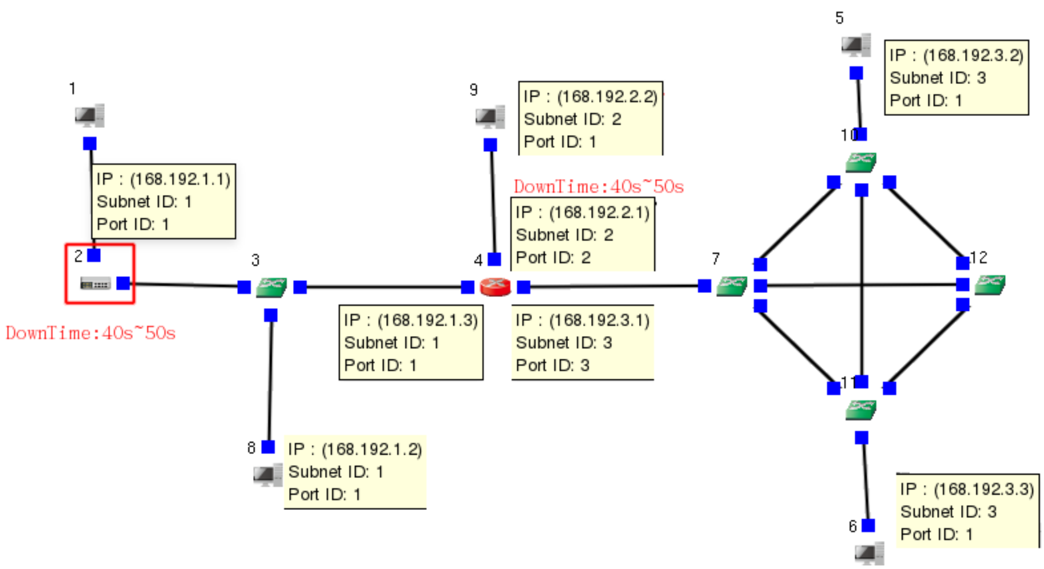
When user change mode to E Mode E Edit Property, the simulator will assign IP address automatically. The assigned IP address would be 1.0.xxx.xxx as below figure.



For IP Prefix could be changed by user's settings, GUI provides the interface in Tab "IP Prefix" from "Menu→G_Tools→Simulation". This settings must be executed in D Mode D Draw Topology to avoid the action that simulator assigns IP address automatically in E Mode. The setting path is as below figure.



After complete the IP Prefix, please change mode to E Mode **E** Edit Property. The IP Prefix would be assigned from GUI settings such as 168.192 in below figure.

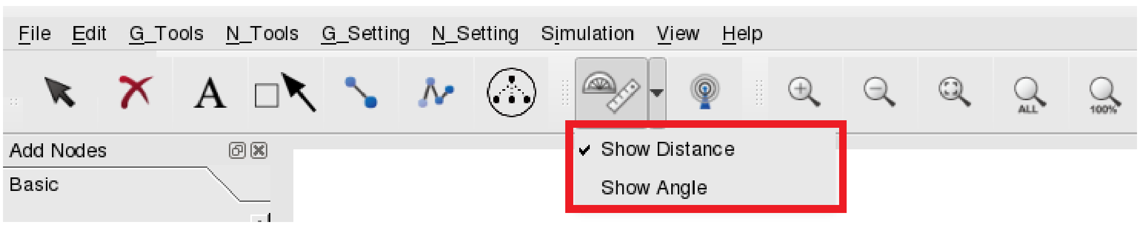


Please notice that if user reset the "IP Prefix", user should check and reset the IP address under Application command line.

Appendix E

Introduction to toolkit “Measure distance and angle” 

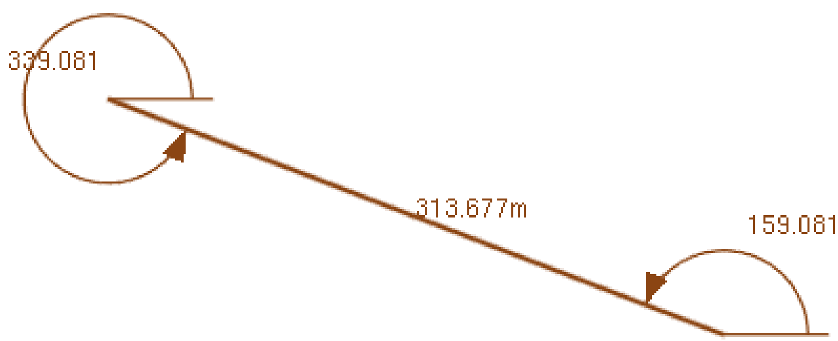
GUI provides a toolkit “Measure distance and angle”  for user to calculate the distance and angle between two Wireless Nodes. After click the button “Measure distance and angle”, the parameter setting of “Show Distance” and “Show Angle” will be displayed as below figure. User could decide which one or both should be displayed.



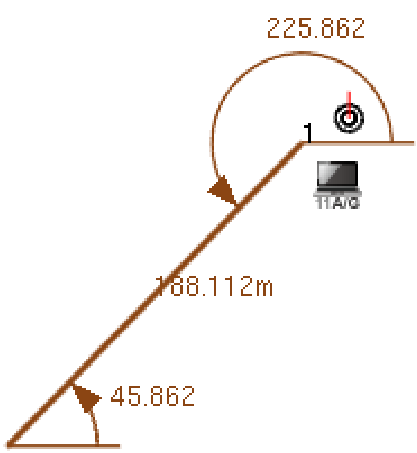
Show Distance: It could display the distance between two points. The default setting is opened.
Show Angle: It could display the angle for two points. The default setting is closed.

There are three ways to use “Measure distance and angle” as below.

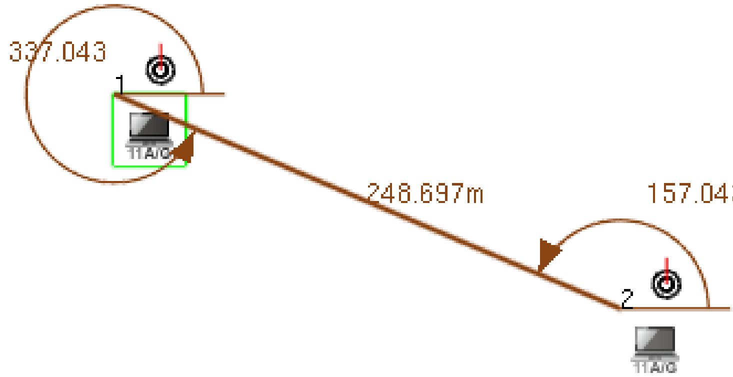
1. **Point to Point**



2. **Node to Point**

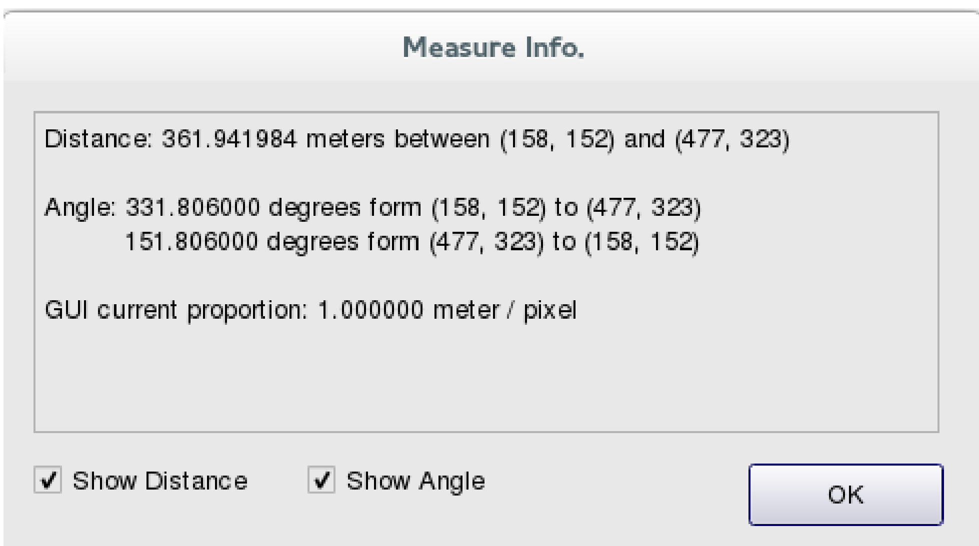


3. **Node to Node**

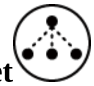


Notice: When using 2nd or 3rd methods, the upper left of Node is the point for distance and angle.

In addition to displaying the distance and angle information directly, it also could double click the line between two points. It will show more information such as coordinate, scale as below dialog.

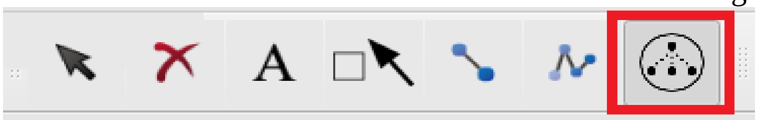


Appendix F

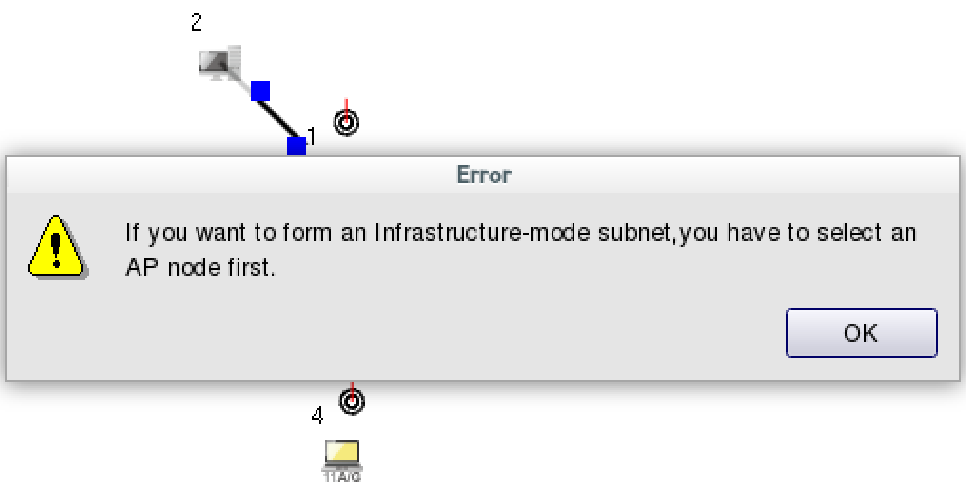
Introduction for toolkit Wireless Subnet 

1. The operation procedure

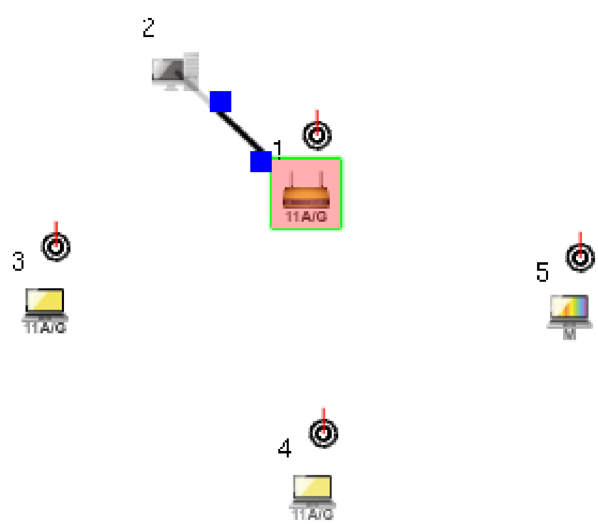
(1) Please click the tool bar icon Wireless Subnet as below figure.



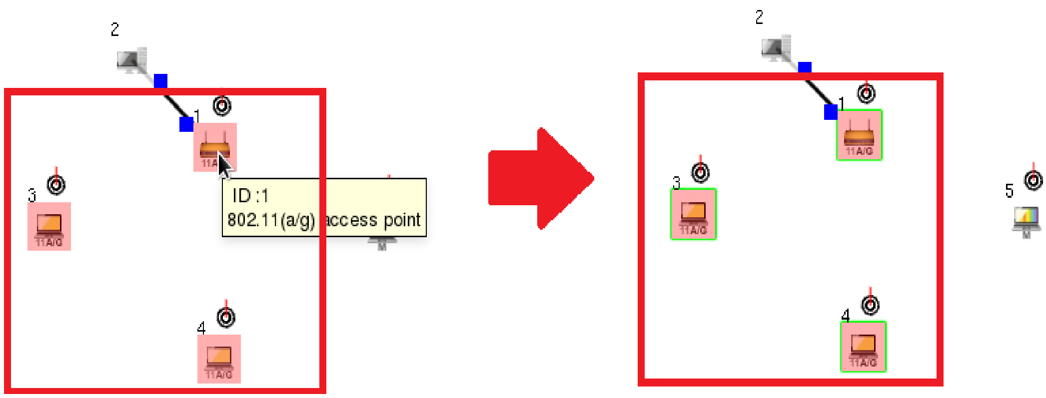
(2) Click one Mobile Node and decide this Mobile Node belongs to what kind of Network Type such as Infrastructure Mode or Ad-hoc mode. For Nodes which belong to Infrastructure Mode, firstly click the AP in the same subnet or an error message dialog will be popped as below.



After user click the first Mobile Node AP in Infrastructure Mode, a default background color would be displayed as below figure.

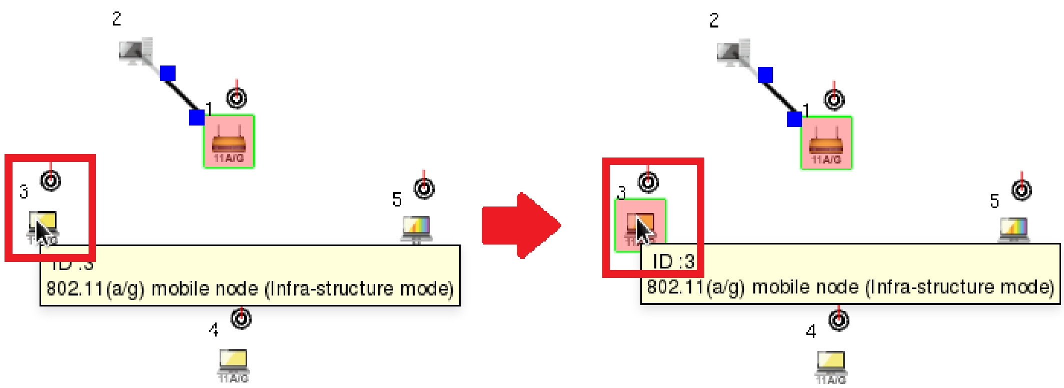


On the other side, all Nodes in the same Subnet would be selected when user click any Node in the same Subnet.

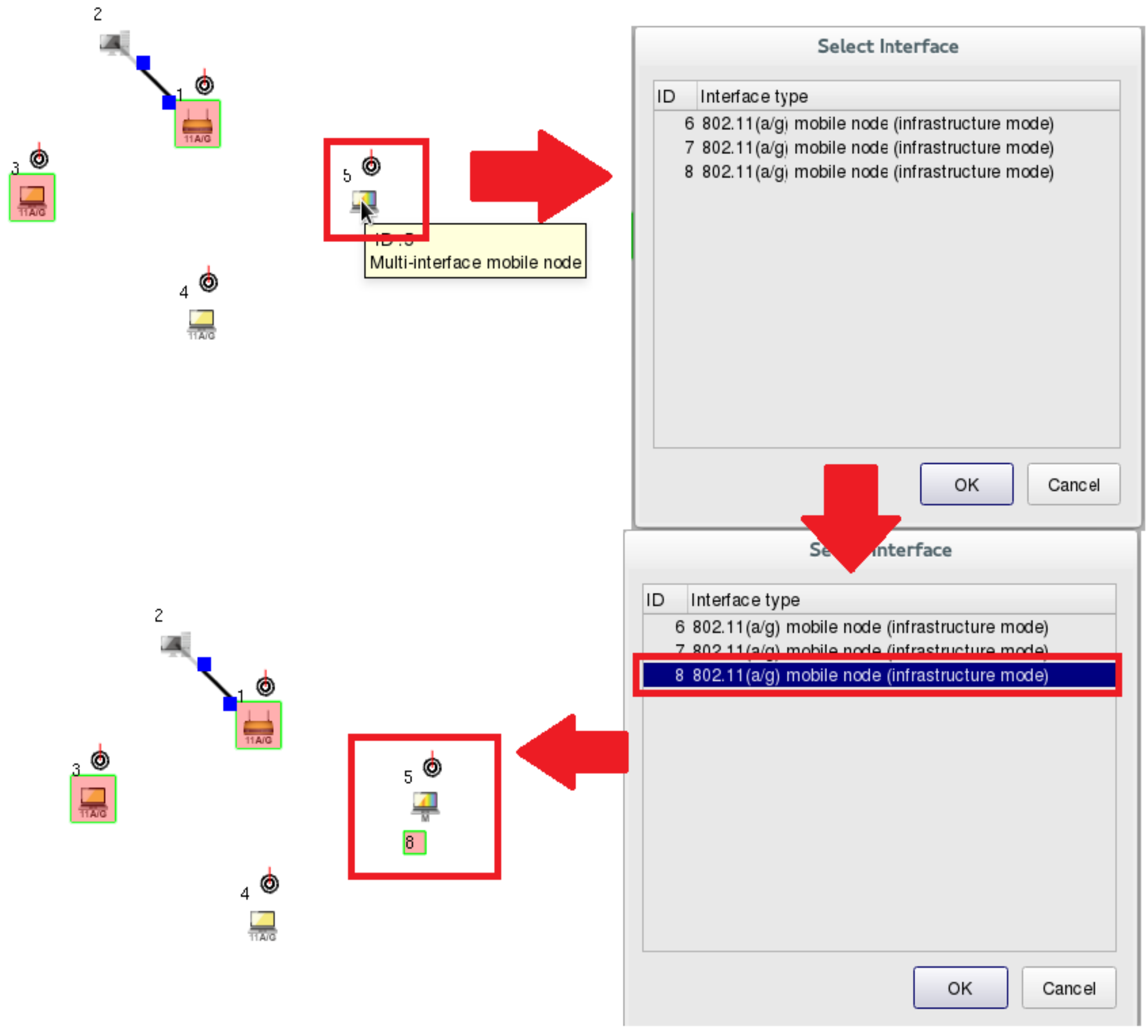


After complete this action, user could process below steps.

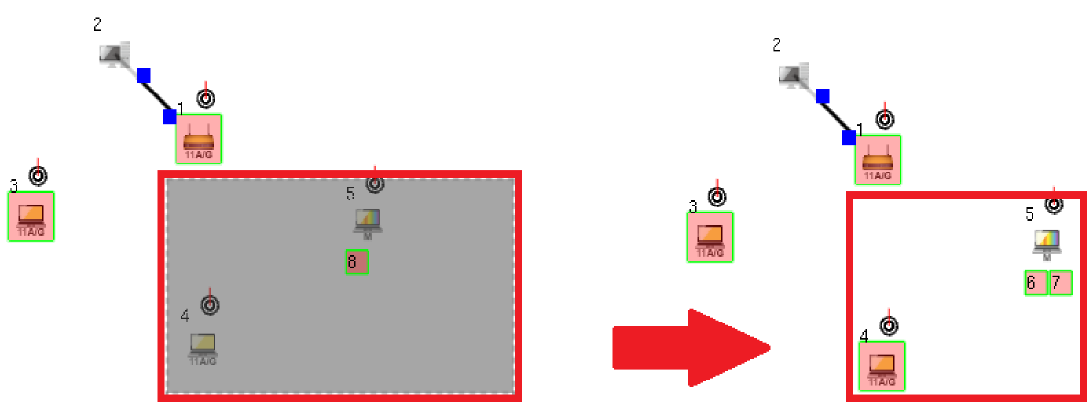
(3) If user would like to add Nodes in the same subnet, user could click the same Network Type Nodes one by one as below figure.



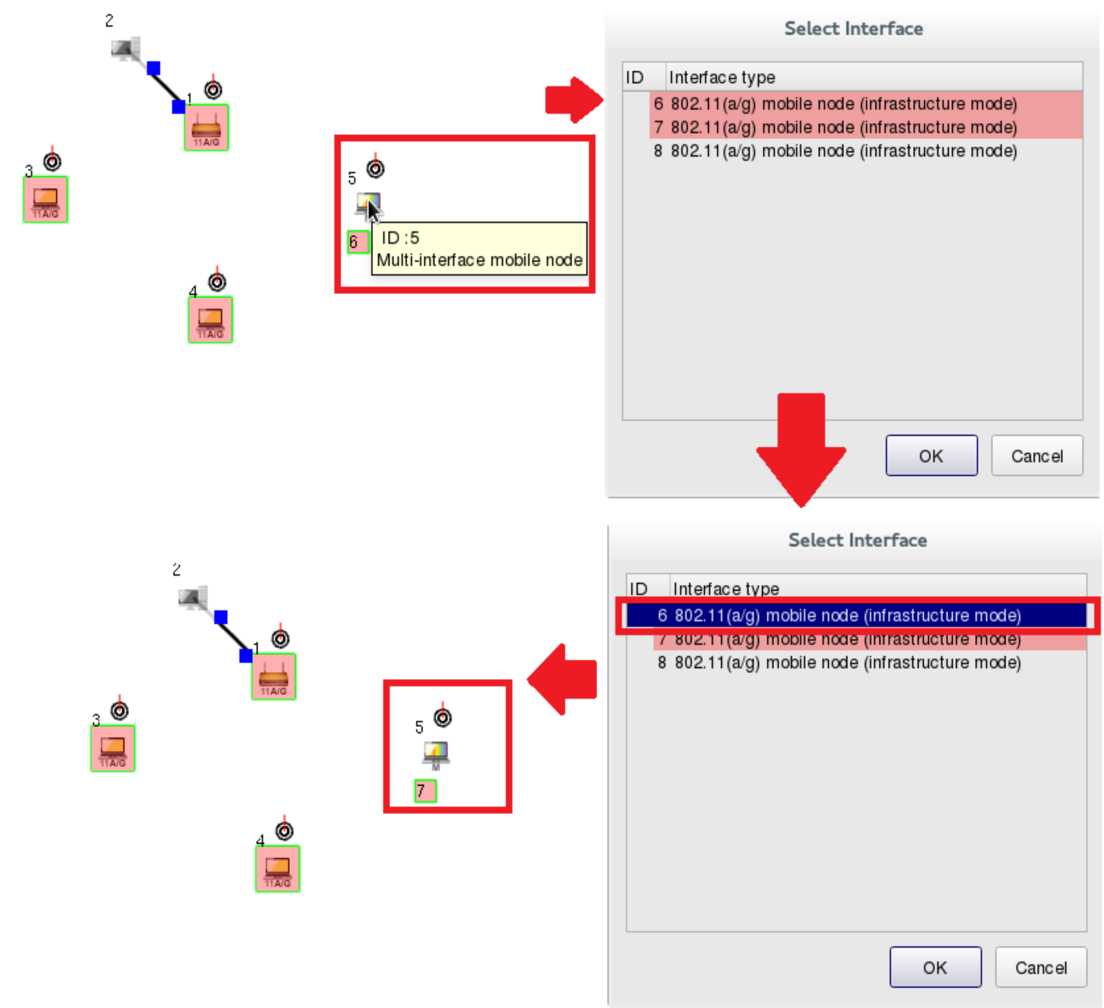
If user select a Multi-interface Node into a subnet, a dialog will be popped for user to select the Interface type as below figure. After complete this action, the selected Node ID of this Multi-interface Node will be displayed in the bottom of this Multi-interface Node.



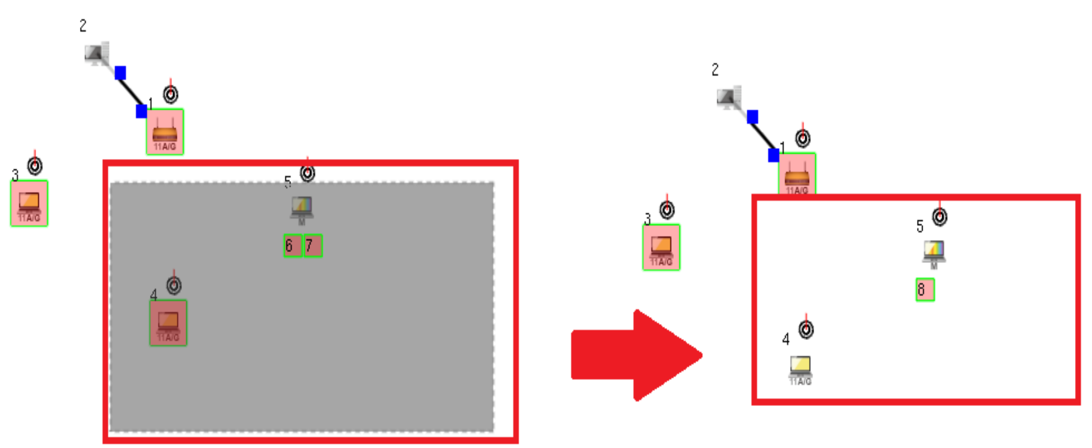
It also could draw an area to select multiple Nodes into a Subnet.



If user would like to delete Nodes into a Subnet, user could click selected Nodes in a Subnet one by one. If user would like to delete a Node which belongs to a Multi-interface Node in a Subnet, a dialog would be popped for user to select the Interface. Then click button OK to complete this action.

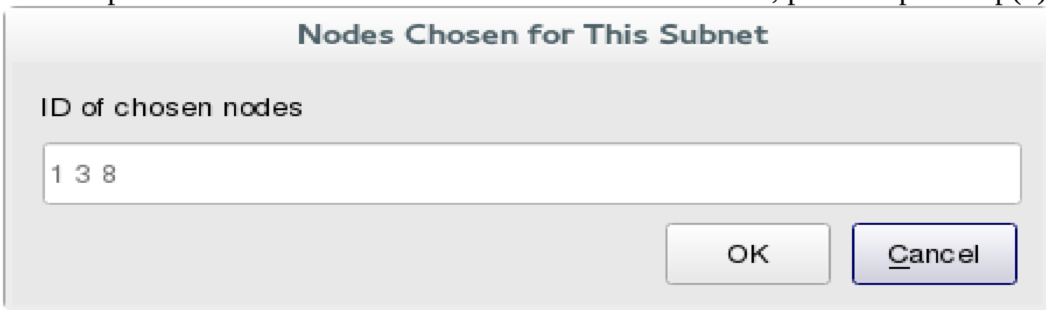


It also could draw an area to delete multiple Nodes in a Subnet.



Note: Please double check the result in a Subnet to avoid incorrect Nodes.

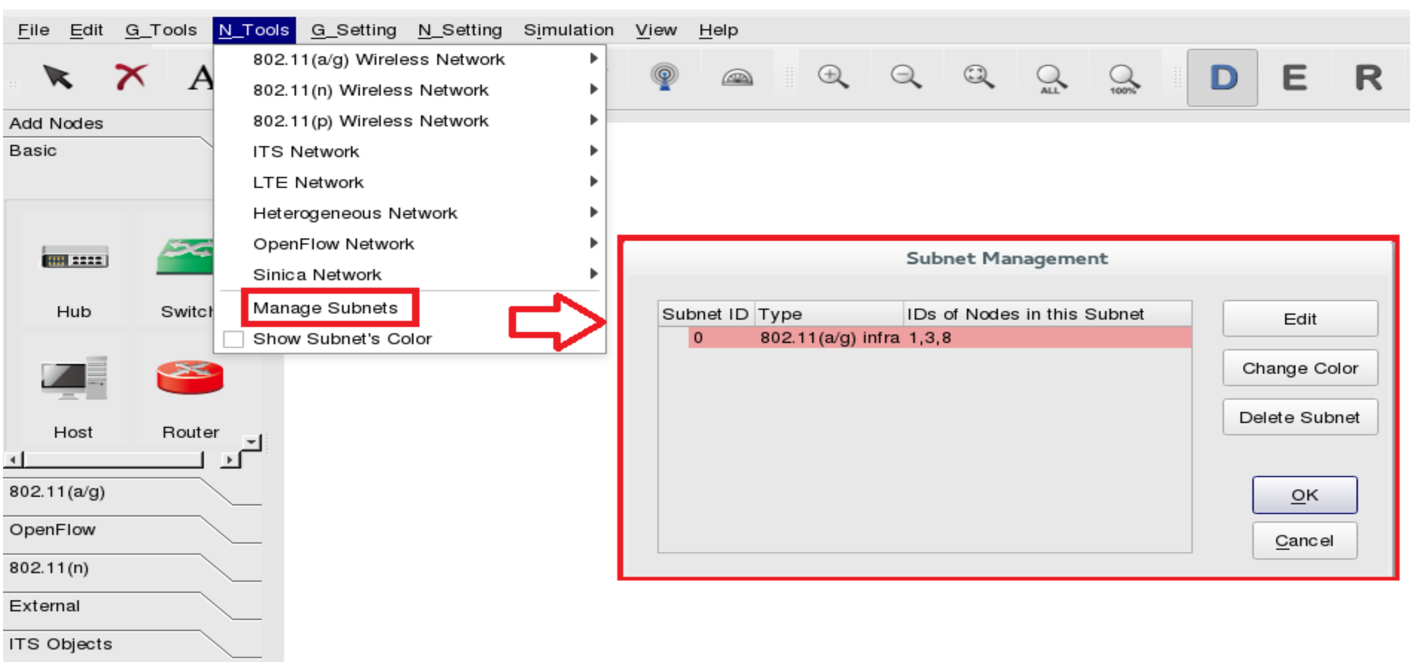
(4) After complete Nodes chosen in a Subnet, click right button of mouse. It will pop a dialog "**Nodes Chosen for This Subnet**" which displays the Node ID in this Subnet. If the result is right, click button "OK" to complete it. If there are some error Node IDs in this Subnet, please click button "Cancel" to recover into the pre-status. If use would like to select another Subnet, please repeat step(2) to step(4).



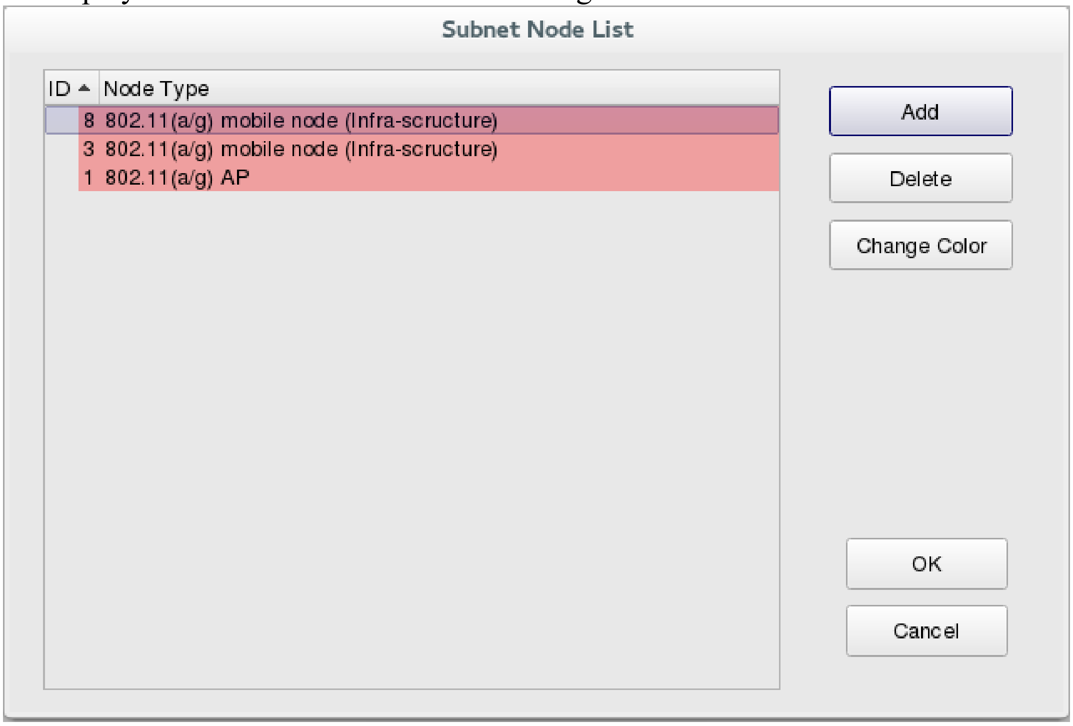
Note: GUI will automatically go to the pre-status of Subnet when user doesn't confirm the result of Subnet by click the right button of mouse and execute other actions such as "**Delete Node**", "**Add Node**"… etc.

2. Manage Subnet (From Menu->N_Tools->Manage Subnet)

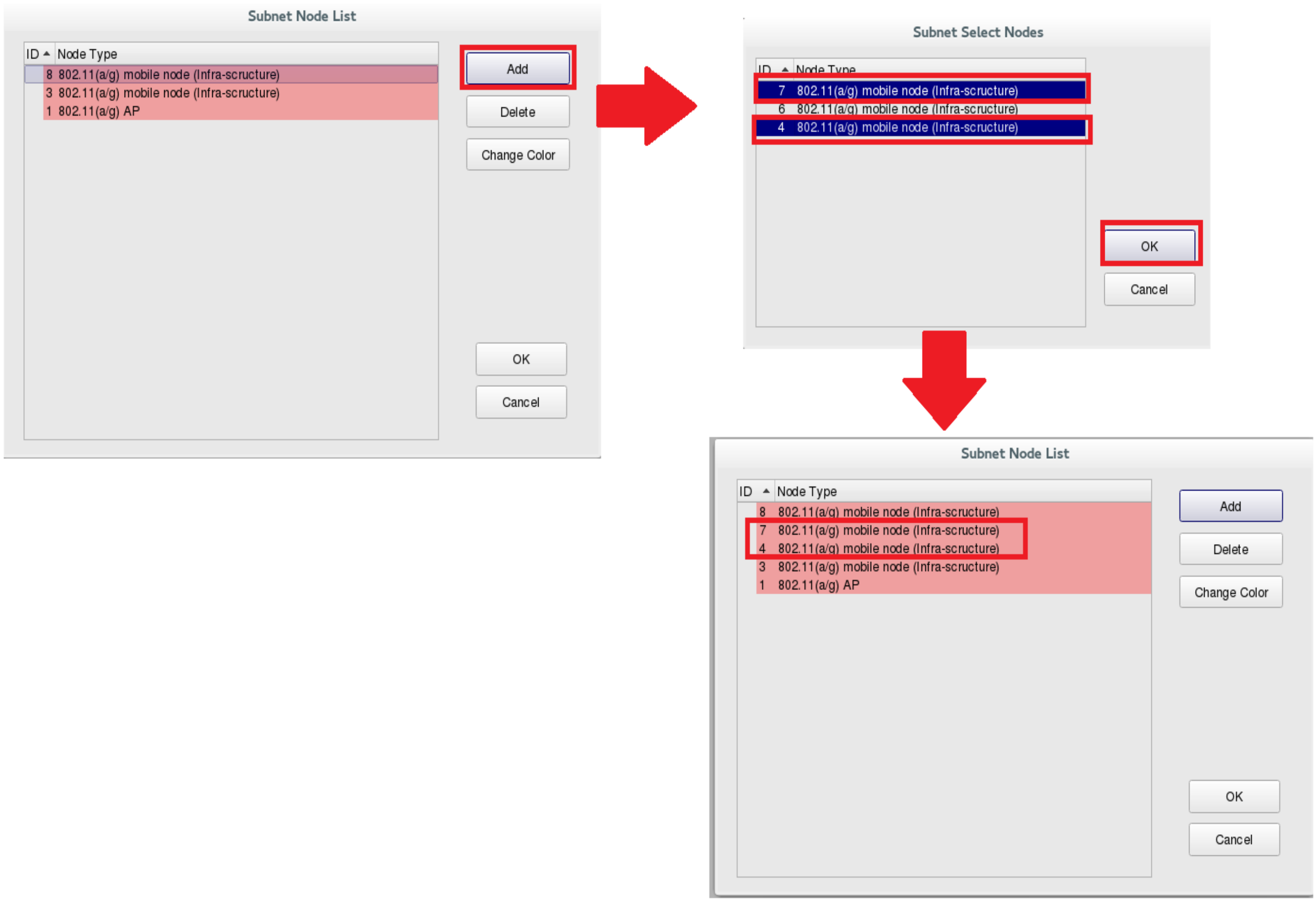
After complete the Subnet selection, if user would like to check or manage the status of Subnet, user could open a Subnet Management dialog from Menu->N_Tools->Manage Subnet as below figure.



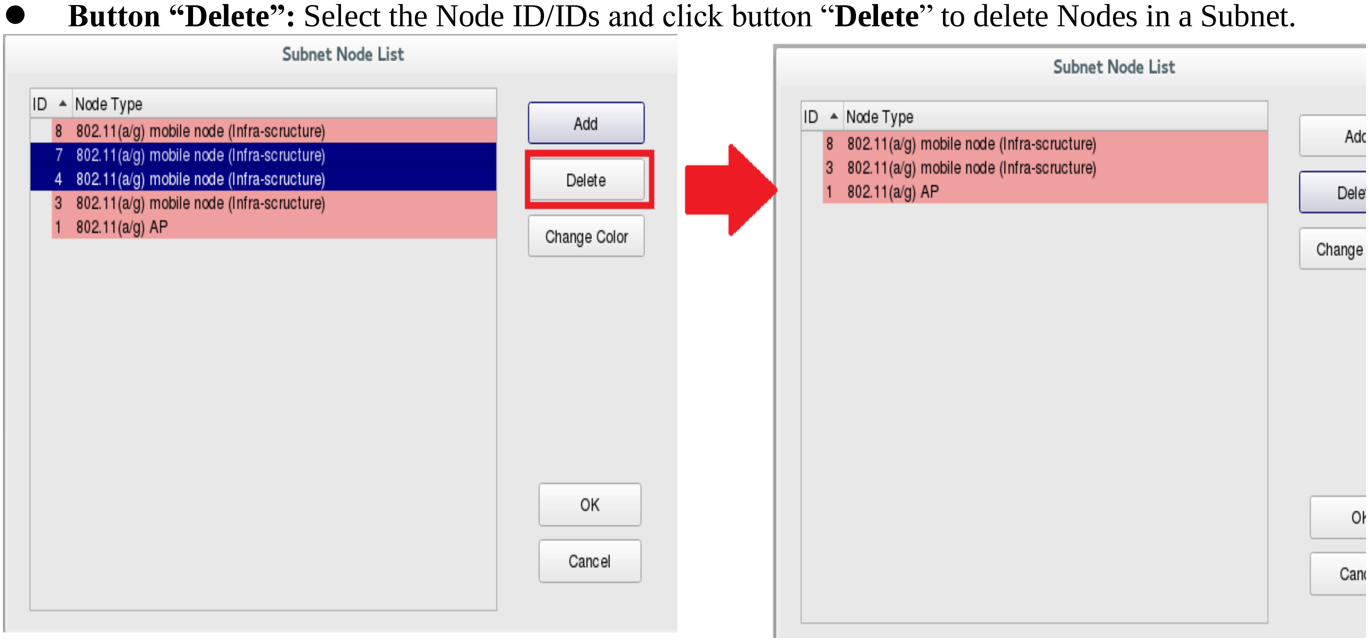
(1) Button " Edit" under Subnet Management: Click the button "Edit" to go into a window of Subnet. It will display a "**Subnet Node List**" as below figure.



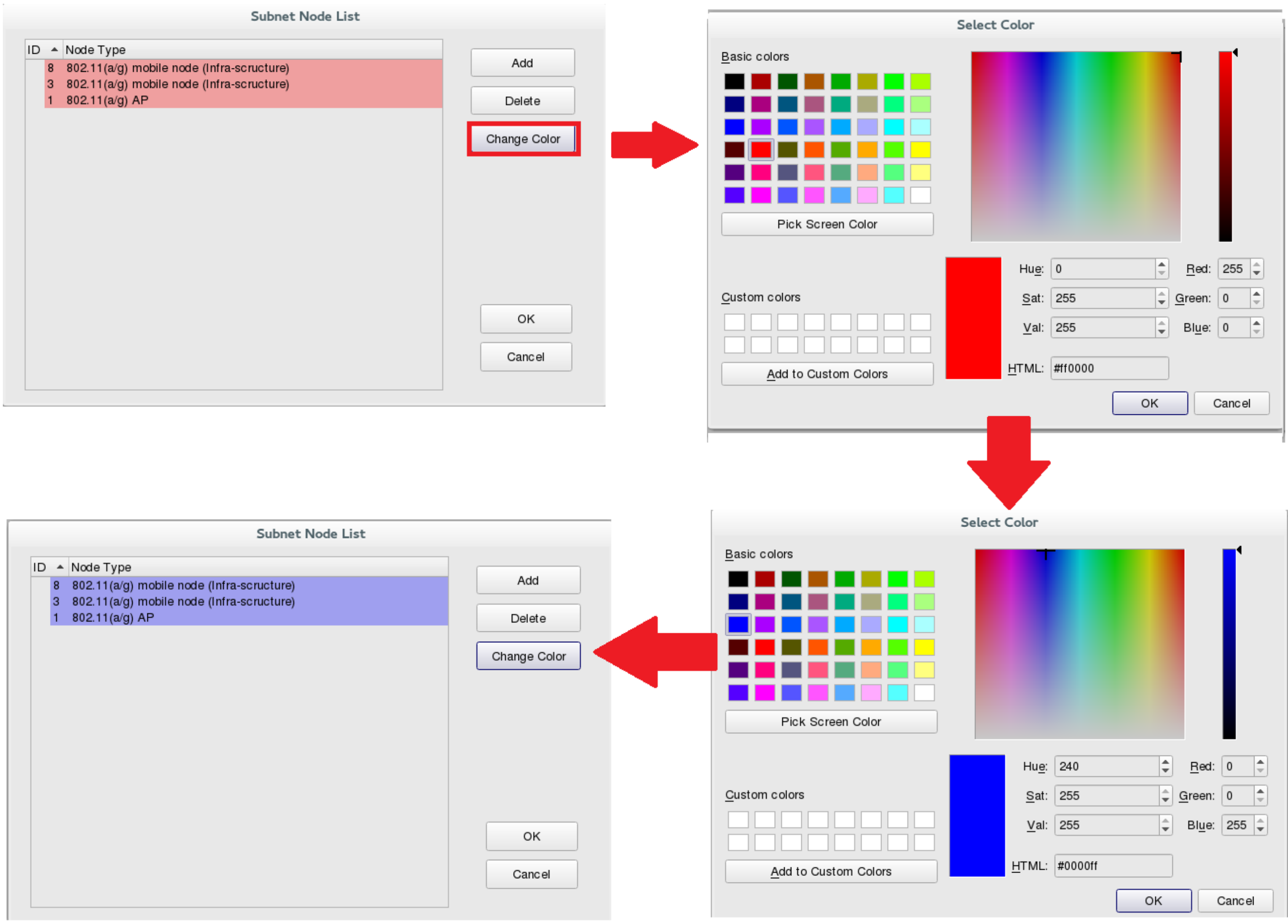
- **Button "Add":** After click button "**Add**", it will pop a dialog "**Subnet Select Nodes**". In this dialog, GUI will display a Nodes list. These Nodes with the same Network Type are still not include in any Subnet. User could add each of these Nodes into this Subnet.



- **Button "Delete":** Select the Node ID/IDs and click button "**Delete**" to delete Nodes in a Subnet.



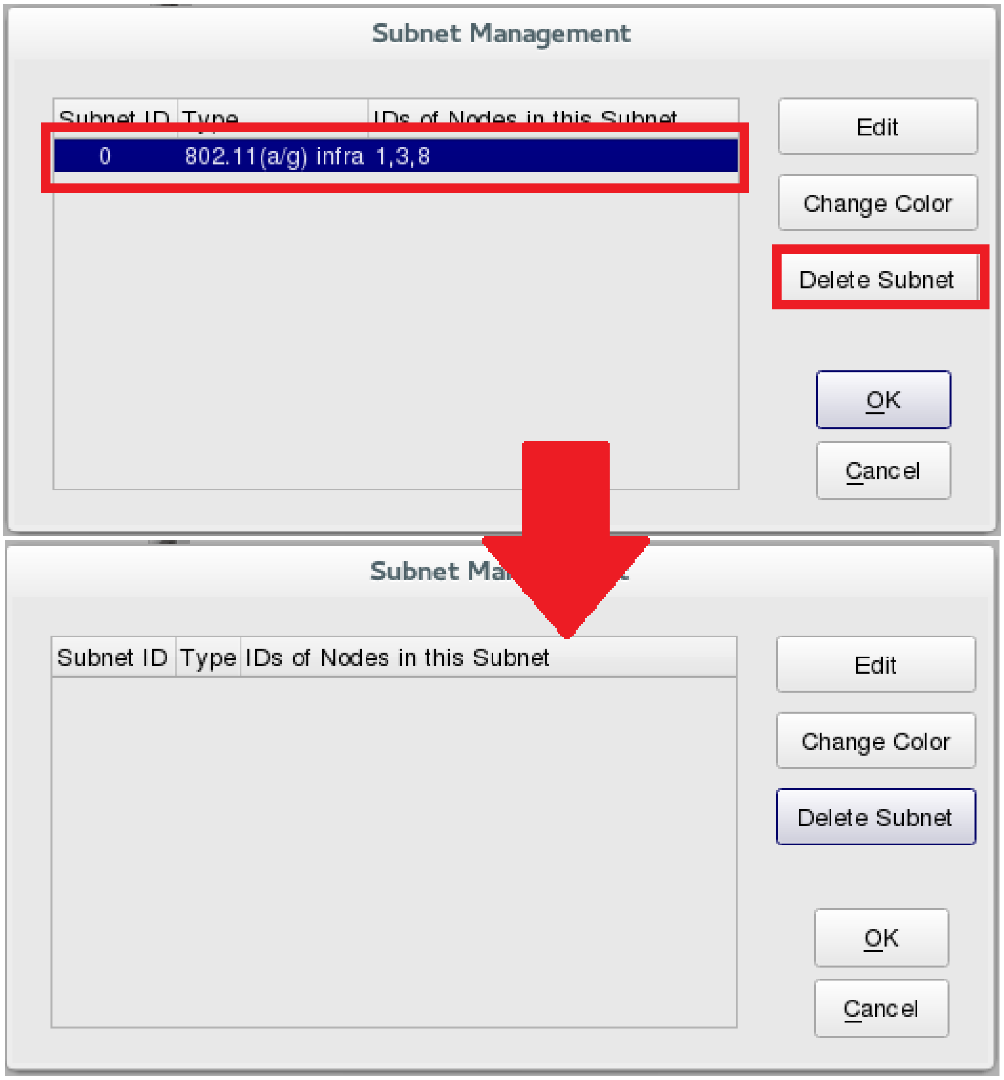
- **Button "Change Color":** It will pop a dialog of "**Select Color**" for user to change the represent color of Subnet.



(2) Button "Change Color" under Subnet Management: It will pop a dialog of "Select Color" for user to change the represent color of Subnet. The method is the same as Button Change Color under Edit.

(3)

(4) Button "Delete Subnet" under Subnet Management: To delete a Subnet in this topology.


Subnet Management
Subnet ID Type IDs of Nodes in this Subnet
0 802.11(a/g) infra 1,3,8
Edit
Change Color
Delete Subnet
OK
Cancel
Subnet Management
Subnet ID Type IDs of Nodes in this Subnet
Edit
Change Color
Delete Subnet
OK
Cancel