NS2-SIMULATION INTRODUCTION

Network simulation is an important tool in developing, testing and evaluating network protocols. Simulation can be used without the target physical hardware, making it economical and practical for almost any scale of network topology and setup. It is possible to simulate a link of any bandwidth and delay, even if such a link is currently impossible in the real world. With simulation, it is possible to set each simulated node to use any desired software. This means that meaning deploying software is not an issue. Results are also easier to obtain and analyze, because extracting information from important points in the simulated network is as done by simply parsing the generated trace files.

Simulation is only of use if the results are accurate, an inaccurate simulator is not useful at all. Most network simulators use abstractions of network protocols, rather than the real thing, making their results less convincing. S.Y. Wang reports that the simulator OPNET uses a simplified finite state machine to model complex TCP protocol processing. [19] NS-2 uses a model based on BSD TCP, it is implemented as a set of classes using inheritance. Neither uses protocol code that is used in real world networking.

GETTING STARTED

Setting up the environment

A user using the NCTUns in single machine mode, needs to do the following steps before he/she starts the GUI program:

1. Set up environment variables:

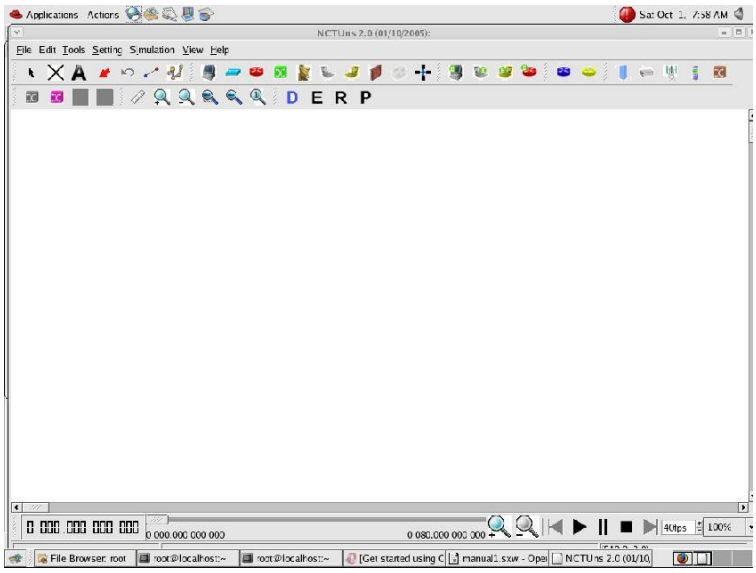
Before the user can run up the dispatcher, coordinator, or
NCTUns GUI program he/she must set up the
NCTUNSHOME environment variable.

2. Start up the dispatcher on terminal 1.

3. Start up the coordinator on terminal 2.

4. Start up the nctunsclient on terminal 3.

After the above steps are followed, the starting screen of NCTUns disappears and the



user is presented with the working window as shown below:

Drawing A Network Topology

To draw a new network topology, a user can perform the following steps:

Choose Menu->File->Operating Mode-> and make sure that the “Draw Topology” mode is checked. This is the default mode of NCTUns when it is launched. It is only in this mode that a user can draw a new network topology or change an existing simulation topology. When a user switches the mode to the next mode “Edit Property”, the

simulation network topology can no longer be changed.

1. Move the cursor to the toolbar.

2. Left-Click the router icon on the toolbar.

3. Left-Click anywhere in the blank working area to add a router to the current network topology. In the same way we can add switch, hub,WLAN access point,WLAN mobile node , wall (wireless signal obstacle) etc.

4. Left-Click the host icon on the toolbar. Like in step 4, add the required number of hosts to the current topology.

5. To add links between the hosts and the router, left-click the link icon on the toolbar to select it.

6. Left-Click a host and hold the mouse button. Drag this link to the router and then release the mouse left button on top of the router. Now a link between the selected host and the router has been created.

7. Add the other, required number of links in the same way. This completes the creation of a simple network topology.

8. Save this network topology by choosing Menu->File->Save. It is saved with a .tpl extension.

9. Take the snapshot of the above topology.

Editing Node's Properties

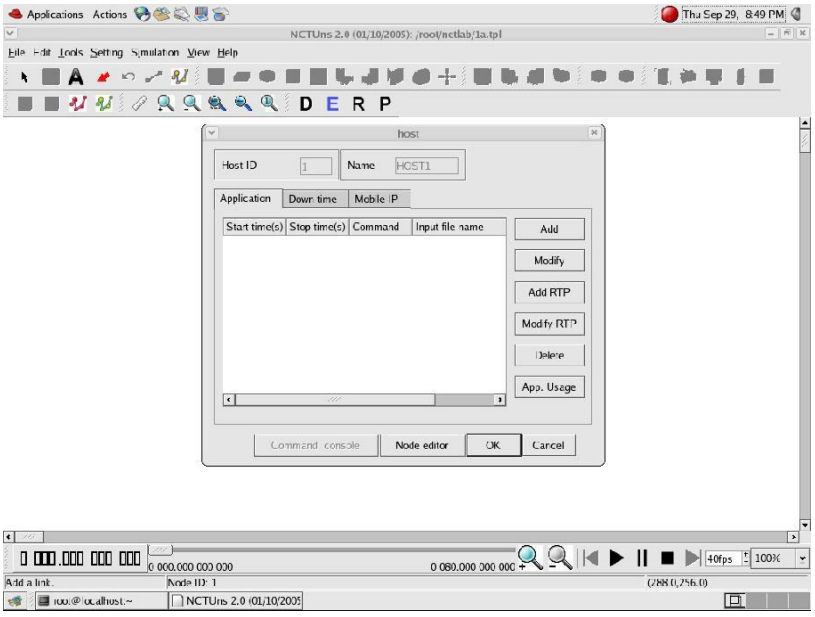
1. A network node (device) may have many parameters to set. For example, we may have to set the maximum bandwidth, maximum queue size etc to be used in a network

interface. For another example, we may want to specify that some application programs (traffic generators) should be run on some hosts or routers to generate network traffic.

2. Before a user can start editing the properties of a node, he/she should switch the mode from the “Draw Topology” to “Edit Property” mode. In this mode, topology changes can no longer be made. That is, a user cannot add or delete nodes or links at this time.

3. The GUI automatically finds subnets in a network and generates and assigns IP and MAC addresses to layer 3 network interfaces.

4. A user should be aware that if he/she switches the mode back to the “Draw Topology” mode when he/she again switches the mode back to the “Edit Topology” mode, node's IP and MAC addresses will be regenerated and assigned to layer 3 interfaces.



Therefore the application programs now may use wrong IP addresses to communicate with their partners.

Running the Simulation

1. When a user finishes editing the properties of network nodes and specifying application programs to be executed during a simulation, he/she can start running the simulation.

2. In order to do so, the user must switch mode explicitly from “Edit Property” to “Run Simulation”. Entering this mode indicates that no more changes can (should) be made to the simulation case, which is reasonable. This simulation is about to be started at this moment; of course, any of its settings should be fixed.

3. Whenever the mode is switched to the “ Run Simulation” mode, the many simulation files that collectively describe the simulation case will be exported. These simulation files will be transferred to the (either remote or local) simulation server for it to execute the simulation. These files are stored in the “ main File Name.sim” directory, where main Filename is the name of the simulation case chosen in the “Draw Topology” mode.

Playing Back the Packet Animation Trace

After the simulation is finished, the simulation server will send back the simulation result files to the GUI program after receiving these files, the GUI program will store these files in the “results directory” .It will then automatically switch to “play back mode”.

1. These files include a packet animation trace file and all performance log files that the

user specifies to generate. Outputting these performance log files can be specified by checking some output options in some protocol modules in the node editor. In addition to this, application programs can generate their own data files.

3. The packet animation trace file can be replayed later by the packet animation player. The performance curve of these log files can be plotted by the performance monitor.

Post Analysis

1. When the user wants to review the simulation results of a simulation case that has been finished before, he /she can run up the GUI program again and then open the case's topology file

2. The user can switch the mode directly to the “Play Back” mode. The GUI program will then automatically reload the results (including the packet animation trace file and performance log file.

3. After the loading process is finished, the user can use the control buttons located at the bottom of the screen to view the animation.

Simulation Commands

The following explains the meaning of each job control command:

Run: Start to run the simulation.

Pause: Pause the currently -running simulation.

Continue: Continue the simulation that was just paused.

- **Stop**: Stop the currently -running simulation

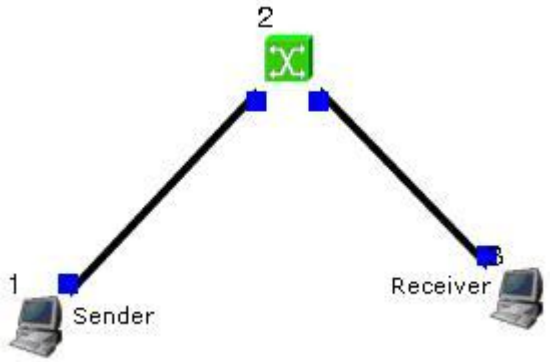
- **Abort**: Abort the currently running simulation. The difference between “stop” and “abort” is that a stopped simulation job's partial results will be transferred back to GUI
- files.

- **Reconnect**: The Reconnect command can be executed to reconnect to a simulation job that was previously disconnected. All disconnected jobs that have not finished their simulations or have finished their simulations but the results have not been retrieved back to be a GUI program by the user will appear in a session table next to the "Reconnect" command. When executing the reconnect command, a user can choose a disconnected job to reconnect from this session table.

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

Experiment: 1

Simulate a three node point to point network with a duplex link between them. Set the queue size and vary the bandwidth and find the number of packets dropped.



Step1: Drawing topology

1. Select/click the HOST icon on the toolbar and click the left mouse button on the editor, to place a HOST1 on the editor.

   Repeat the above procedure and place another host "HOST2" on the editor.

2. Select/click the HUB icon on the toolbar and click the left mouse button on the editor, to place HUB1 on the editor.

3. Click on the LINK icon on the toolbar and connect HOST1 to HUB1 and HUB1 to HOST2

4. Click on the "E" icon on the toolbar to save the current topology
   e.g: file1.tpl
   (Look for the ******.tpl extension.)

NOTE: Changes cannot / (should not) be done after selecting the "E" icon.

Step2: Configuration

1. Double click the left mouse button while cursor is on HOST1 to open the HOST window.
2. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

   stg –u 1024 100 1.0.1.2

3. Click OK button on the command window to exit and once again click on the OK button on the HOST window to exit.

4. Double click the left mouse button while cursor is on HOST2 to open the HOST window.

5. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

   rtg –u –w log1

6. Click OK button on the command window to exit.

7. Click NODE EDITOR Button on the HOST window and select the MAC tab from the modal window that pops up.

8. Select LOG STATISTICS and select checkboxes for Number of Drop Packet and Number of Collisions in the MAC window

9. Click OK button on the MAC window to exit and once again click on the

OK button on the HOST window to exit.

Note: To set QUEUE size

1. Double click the left mouse button while cursor is on HOST2 to open the HOST window.

2. Click NODE EDITOR Button on the HOST window and select the FIFO tab from the modal window that pops up.

3. Change Queue size (Default 50).
4. Click OK button on the FIFO window to exit and once again click on the OK button on the HOST window to exit.

Step3: Simulate

i. Click "R" icon on the tool bar

ii. Select Simulation in the menu bar and click/ select RUN in the dropdown list to execute the simulation.

iii. venTo start playback select "►" icon located at the bottom right corner of the editor.

iv. To view results, Open up new TERMINAL window, move to file1.results folder and open collision and drop log files in separate TERMINAL window.

Caution: file1 is the hypothetical name given to this simulation. **(Refer Step 1.4)**

Changing configurations

Change 1

1. Open the above file,
2. Do not change the topology or any other configuration,
3. Select E icon on the toolbar
4. Reduce the bandwidth at link2 by double clicking the left mouse button while cursor is on link2 .(Change bandwidth on both tabs Uplink/Downlink)
5. Repeat Step3 (Simulate)

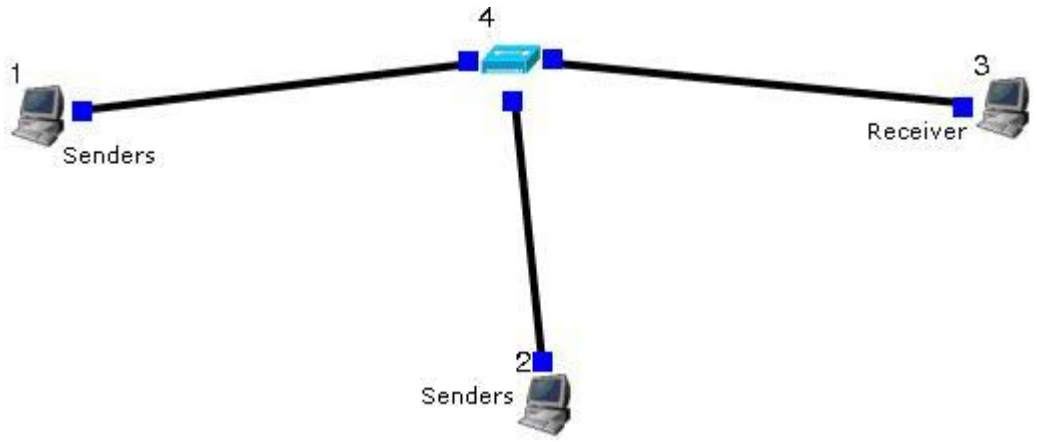
Change 2

1. Open the above file,
2. Remove HUB and replace it with SWITCH.
3. Do not change anything in the configuration
4. Repeat Step3(Simulate)

Experiment: 2

Simulate a four node point to point network and connect the link as follows Apply a TCP agent between n0 to n3 and apply a UDP agent between n1 and n3.

Apply relevant applications over TCP and UDP agents changing the parameters and determine the number of packets sent by two agents.



Step1: Drawing topology

1. Select/click the HOST icon on the toolbar and click the left mouse button on the editor, to place a host on the editor.

Repeat the above procedure and place two other hosts “HOST2” and “HOST3” on the editor.

2. Select/click the HUB (or SWITCH) icon on the toolbar and click the left mouse button on the editor, to place a HUB (or SWITCH) on the editor.

3. Click on the LINK icon on the toolbar and connect HOST1 to HUB, HOST2 to HUB and HUB to HOST3

4. Click on the "E" icon on the toolbar to save the current topology **e.g:** file2.tpl
   (Look for the ******.tpl extension.)

NOTE: Changes cannot / (should not) be done after selecting the "E" icon.

Step2: Configuration

1. Double click the left mouse button while cursor is on HOST1 to open the HOST window.

2. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

   stcp –p 21 –l 1024 1.0.1.3

3. Click OK button on the command window to exit

4. Click NODE EDITOR Button on the HOST window and select the MAC tab from the modal window that pops up.

5. Select LOG STATISTICS and select checkbox for output throughput in the MAC window

6. Click OK button on the MAC window to exit and once again click on the OK button on the HOST window to exit.

7. Double click the left mouse button while cursor is on HOST2 to open the HOST window.

8. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

   stg –u 1024 100 1.0.1.3

9. Click OK button on the command window to exit

10. Click NODE EDITOR Button on the HOST window and select the MAC tab from the modal window that pops up.

11. Select LOG STATISTICS and select checkbox for output throughput in the MAC window

12. Click OK button on the MAC window to exit and once again click on the OK button on the HOST window to exit.

Double click the left mouse button while cursor is on HOST3 to open the HOST window.

14. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

rtcp –p 21 –l 1024

15. Click OK button on the command window to exit.

16. Also add the following command on

HOST3 rtg –u –w log1

17. Click NODE EDITOR Button on the HOST window and select the MAC tab from the modal window that pops up.

18. Select LOG STATISTICS and select checkbox for input and output throughput in the MAC window

19. Click OK button on the MAC window to exit and once again click on the OK button on the HOST window to exit.

Step3: Simulate

i. Click "R" icon on the tool bar

ii. Select Simulation in the menu bar and click/ select RUN in the dropdown list to execute the simulation.

iii. To start playback select "►" icon located at the bottom right corner of the editor.

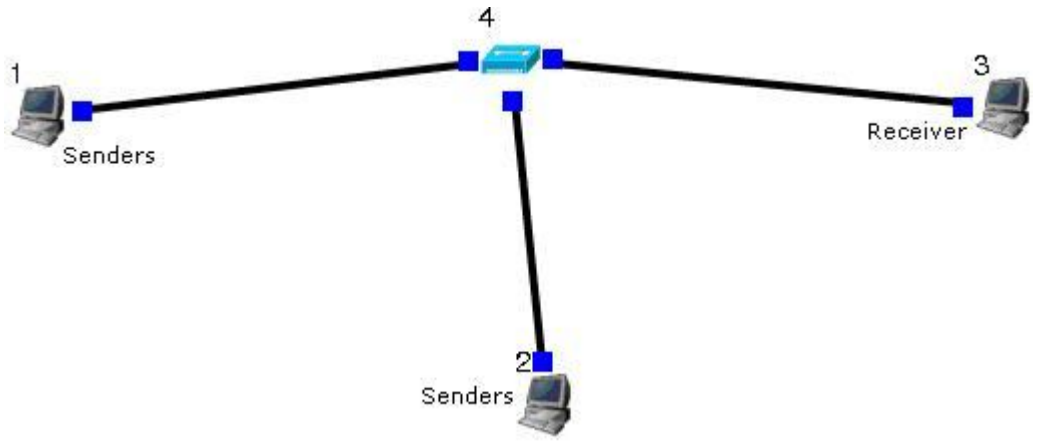
iv. To view results, Open up new TERMINAL window, move to

file2.results folder and open input and output throughput log files in separate TERMINAL window.

Caution: file2 is the hypothetical name given to this simulation. **(Refer Step 1.4)**

Experiment: 3

Simulate the different types of internet traffic such as FTP, TELNET over a network and analyze the throughput.



Step1: Drawing topology

1. Select/click the HOST icon on the toolbar and click the left mouse button on the editor, to place a host on the editor.

Repeat the above procedure and place another host "HOST2" on the editor.

2. Select/click the HUB (or SWITCH) icon on the toolbar and click the left mouse button on the editor, to place a HUB (or SWITCH) on the editor.

3. Click on the LINK icon on the toolbar and connect HOST1 to HUB and HUB to HOST2

4. Click on the "E" icon on the toolbar to save the current topology **e.g:** file3.tpl
   (Look for the ******.tpl extension.)

NOTE: Changes cannot / (should not) be done after selecting the “E” icon.

Step2: Configuration
(FTP application)

1. Double click the left mouse button while cursor is on HOST1 to open the HOST window.

2. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

   stcp –p 21 –l 1024 1.0.1.2

3. Click OK button on the command window to exit and once again click on the OK button on the HOST window to exit.

4. Double click the left mouse button while cursor is on HOST2 to open the HOST window.

5. Select Add button on the HOST window to invoke the command window and provide the following command in the command textbox.

   rtcp –p 21 –l 1024

6. Click OK button on the command window to exit.

7. Click NODE EDITOR Button on the HOST window and select the MAC tab from the modal window that pops up.

8. Select LOG STATISTICS and select checkbox for output throughput in the MAC window

9. Click OK button on the MAC window to exit and once again click on the OK button on the HOST window to exit.

Step3: Simulate

i. Click “R” icon on the tool bar
ii. Select Simulation in the menu bar and click/ select RUN in the

dropdown list to execute the simulation.

iii. To start playback select "►" icon located at the bottom right corner of the editor.

iv. To view results, Open up new TERMINAL window, move to file3.results folder and open output throughput log files in separate TERMINAL window.

Caution: file3 is the hypothetical name given to this simulation.

Changing configurations

Change 1 (TELNET application)

i. Open the above file,
ii. Do not change the topology or any other configuration,
iii. Select E icon on the toolbar
iv. Change port number to 23 for TELNET application

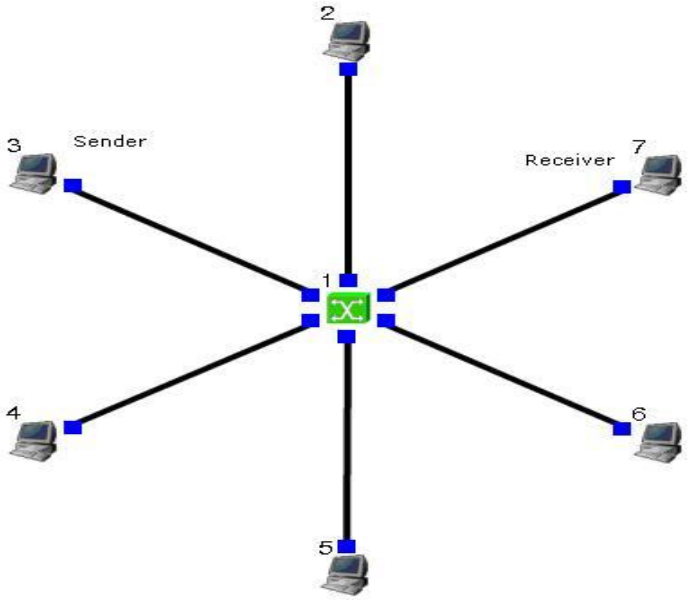
i.e.
stcp –p 23 –l 1024 1.0.1.2

rtcp –p 23 –l 1024

v. Repeat Step3 (Simulate)

Experiment: 4

Simulate the transmission of ping messages over a network topology consisting of 6 nodes and find the number of packets dropped due to congestion.



Step1: Drawing topology

1. Select/click the SUBNET icon on the toolbar and click the left mouse button on the editor, to place a SUBNET on the editor.

2. A pop up window appears requesting the number of nodes and radius for the subnet

   Set number of nodes=6;
   Set radius of subnet >150

3. Click on the "E" icon on the toolbar to save the current topology **e.g:** file4.tpl
   (Look for the ******.tpl extension.)

   NOTE: Changes cannot / (should not) be done after selecting the "E" icon.

Step2: Configuration

4. Double click the left mouse button while cursor is on a HOST to open the HOST window.

5. Click NODE EDITOR Button on the HOST window and select the INTERFACE tab (1^{st} tab) from the modal window that pops up.

6. Determine the IP address of the selected host.

7. Click OK button on the INTERFACE window to exit and once again click on the OK button on the HOST window to exit.

8. Repeat the above step for 2 other HOSTS

9. Also click NODE EDITOR Button on the HOST window and select the MAC tab from the modal window that pops up.

10. Select LOG STATISTICS and select checkbox for drop and collision log statistics in the MAC window

11. Click OK button on the MAC window to exit and once again click on the OK button on the HOST window to exit.
12. Repeat steps 6 to 9 for the other hosts selected at step 5.

13. Select G_Setting from the menu bar and select Simulation from the drop down list

    Set simulation time>600sec

Step3: Simulate

i. Click "R" icon on the tool bar

ii. Select Simulation in the menu bar and click/ select RUN in the dropdown list to execute the simulation.

During simulation, double click the mouse button on a HOST, the HOST window pops up, select / click on command console button located at the bottom.

iv. A terminal window appears, type **ping IP address** of a host in the subnet at the command prompt.

v. To view results, Open up new TERMINAL window, move to file4.results folder and open drop and collision log files in separate TERMINAL window.

Caution: file4 is the hypothetical name given to this simulation.
(Refer Step 1.3)