Guide to GloMoSim

Disclaimer

Undeniably, there are already several existing GloMoSim tutorials and manuals out there, and the purpose of this guide is not to undermine the efforts and knowledge of previous authors, but to provide additional experiences from a novice user, working with routing protocols. ☺

If you are very familiar with the Linux operating system, then you should skip this guide totally and proceed on to "A Comprehensive GloMoSim Tutorial" – a compilation by Jorge Nuevo.

The installation procedures as follows apply only for Linux machines. In addition, the parameters herein serves only as a guideline for simulating network protocols. If you are planning to simulate for other purposes, the parameters to be used may differ, and you should consult relevant literature in your area of research for the commonly used parameters.

Installing GloMoSim

Download the latest version of GloMoSim from:
http://pcl.cs.ucla.edu/projects/glomosim/academic/topsecret_download.html.
*As of 24 August 2004, the latest version is GloMoSim-2.03.

Unzip the installation file as follows:
$ gunzip –dfv glomosim-2.03.tar.gz
$ tar –xvf glomosim-2.03.tar

The main problem with installing GloMoSim is linking the Parsec compiler. By now, you should be able to see a folder called "glomosim-2.03".
$ cd glomosim-2.03

You should be able to see the following two folders, glomosim and parsec.
$ cd parsec

There are a few available versions of parsec:

- Aix
- Freebsd-3.3
- Irix-6.4
- Redhat-6.0
- Redhat-7.2
- Solaris-2.5.1
- Solaris-2.5.1-cc
- Solaris-gcc303
- Windowsnt-4.0-vc6
- X86-solaris-2.5.1

After identifying the correct version of Parsec which you should be using, copy the appropriate folder to a targeted destination. Suppose that you are using the Redhat-7.2 version, and you want the targeted destination to be /home/parsec.

Go to the home directory and create the folder called "parsec":
$ mkdir parsec

Then, go to the directory where the Parsec that came with GloMoSim was unzipped, and copy the folder as follows:
$ cp redhat-7.2 /home/parsec

This will copy the contents inside the folder redhat-7.2 to the destination /home/parsec.

Now, you have to change some environmental parameters according to the destination folder which you put the Parsec. You can create a file called ".bashrc" inside the root or home folder (if it does not already exist), and include the following:

```
export PCC_DIRECTORY=/home/parsec
PATH=$PATH:$PCC_DIRECTORY/bin
export PATH
```

*Your PCC_DIRECTORY should be set to the targeted destination folder which you created earlier.

Now, type:
$ pcc

You should be able to see the following output
"No input file."

If you do not see the output as above, then add the following inside your ".bashrc" file:

```
LD_LIBRARY_PATH=/usr/local/lib:/opt1/lib
export LD_LIBRARY_PATH
export PS1='\u@\h:\W$ '
```

Try the following command again:
$ pcc

Check that you obtain the correct output before proceeding with your installation.

After this, you just have to compile GloMoSim by going to the main folder of GloMoSim:
$ cd glomosim-2.03/glomosim/main
$ make clean
$ make

Then, run GloMoSim by going to the bin folder:
$ cd ../bin
$ ./glomosim config.in

If you do not want the output to be shown on the command window, you can direct the output to an appropriate output file, as follows:
$ ./glomosim config.in > out.txt

Configuration parameters

The GloMoSim configuration file "config.in" can be located under the path glomosim-2.03/glomosim/bin. It provides the configuration parameters for the simulation.

SIMULATION TIME
Adjust this according to the time period which you want to run your simulations. A typical simulation time is about 300 seconds, but this should vary according to the type of statistics which you are trying to obtain.

SEED
While this value is initially set to 1, you are free to change it to allow other initialization configurations for your node positions. In practice, we normally run about 10 simulations for each scenario, using different seed numbers, and then average the statistics being collected. This helps to reduce any arbitrary randomness of the nodes.

TERRAIN-DIMENSIONS
The units are in metres. A terrain size of 2000 metres by 2000 metres will usually work quite fine. If you would like to vary the average node density, you can either (i) adjust the number of nodes in the network; or (ii) adjust the terrain size.

NUMBER-OF-NODES
This is the number of nodes to be simulated in the network.

NODE-PLACEMENT
The usual practice is to use uniform or random placement of nodes. However, if you are planning to use some other mobility models not provided in the default GloMoSim package (such as Reference Point Group Mobility model), you can use a node generation software (such as BonnMotion, which will be explained later) to create a file containing the placement of nodes. The input file should look something like this:

Node	**0**	**x**	**y**	**z**
0	0	(10.2,	0.8,	0.11)
1	0	(21.3,	1.3,	21.7)

Then, uncomment the following lines as follows:
NODE-PLACEMENT FILE
NODE-PLACEMENT-FILE ./nodes.input

The file "nodes.input" should be placed inside the bin folder of GloMoSim.

MOBILITY
For static networks, mobility should be set to NONE as follows:
MOBILITY NONE

GloMoSim provides the Random Waypoint mobility model, in which nodes move towards a destination with a randomly generated speed (within the specified limits), and then pause there for some time. If you plan to use the Random Waypoint model, the parameters for mobility should be set as follows:
MOBILITY NONE
MOBILITY RANDOM-WAYPOINT
MOBILITY-WP-PAUSE 30S
MOBILITY-WP-MIN-SPEED 0
MOBILITY-WP-MAX-SPEED 10

By using a pause time of 0S (0 seconds), continuous random motion can be emulated. The maximum speed is usually around 20 ms^{-1}, which is equivalent to 72 kmh^{-1} (approximately the average speed of a vehicle).

If you would like to use another mobility model (easily obtained by using the BonnMotion software), specify the input trace files as follows:
MOBILITY TRACE
MOBILITY-TRACE-FILE ./mobility.in

A typical mobility trace file looks like this:

Node	**Time**	**x**	**y**	**z**
10	100S	(10.2,	0.8,	0.11)
10	200S	(21.3,	1.3,	21.7)

The mobility trace file, "mobility.in" should also be put inside the bin folder.

PROPAGATION-PATHLOSS
There are 3 pathloss models available in GloMoSim. By default, we use the Two-Ray (or Ground Reflection) model, which is a more realistic measure of the propagation loss experienced by packets. Refer to Jorge Nuevo's tutorial if you want a more detailed explanation of these models. ☺

RADIO-TYPE
There are 2 types of radios, the standard radio model and the abstract radio model. The default for GloMoSim is the standard radio model, while the abstract model is compatible with the current version (2.1b5) of the ns-2 radio model.

RADIO-TX-POWER
The default value for the transmission power is set to 15 dBm. This is approximately 376.782 metres. To find out the distance for the various transmission powers that you have set, you can do the following:

1. Modify the RADIO-TX-POWER in "config.in".
2. Go to the bin directory in your command line window.
3. $ ./radio_range config.in

You should be able to see the radio range from there.

ROUTING-PROTOCOL
The following routing protocols are available in GloMoSim:

- Bellman Ford
- AODV (Ad Hoc On Demand Distance Vector)
- DSR (Dynamic Source Routing)
- LAR1 (Location Aided Routing)
- WRP (Wireless Routing Protocol)
- Fisheye
- ZRP (Zone Routing Protocol)

If you are planning to use ZRP, you should specify the ZONE-RADIUS, which is set to 2 by default. The zone radius n determines the proactive region of each node in the network, ie: the number of n-hop neighbours which it needs to maintain routes to.

APP-CONFIG-FILE
The application configuration file specifies the types of data traffic that will be used in the simulations. GloMoSim provides FTP, Telnet, HTTP and CBR (Constant Bit Rate) data traffic.
In most simulations, we use CBR data traffic and send data packets at regular time intervals. The "app.conf" file located in the bin folder provides a good explanation of how you can specify data traffic.
One thing to keep in mind, when determining the data traffic, is whether there is enough data flow to actually keep the network saturated. This depends on the number of nodes in the network, the overall bandwidth (which is 2 Mbps by default), etc.

STATISTICS
This section allows you to specify the types of statistics which you are interested in at the end of the simulation. The statistics will be consolidated in the "glomo.stat" file, which can be found in the "bin" folder at the end of each simulation. The types of statistics available include:

- Application
- TCP
- UDP
- Routing
- Network
- MAC
- Radio
- Channel
- Mobility

<u>Using BonnMotion to Generate Mobility Models</u>

As mentioned previously, GloMoSim provides the Random Waypoint mobility model, which may not be suitable for all types of simulations. The BonnMotion software provides a generator for other kinds of mobility models, thus eliminating the need to write your own script file. ☺

BonnMotion can be downloaded from http://web.informatik.uni-bonn.de/IV/Mitarbeiter/dewaal/BonnMotion/

At the time of writing, the BonnMotion provides the following mobility models:

- Gauss Markov
- Manhattan Grid
- Random Waypoint
- RPGM (Reference Point Group Mobility)
- Static network (no movements)

In general, there are two classes of mobility patterns:

- entity mobility models – mobile nodes have movements that are independent of each other;
- group mobility models – mobile nodes have movements that are dependent of each other.

To install BonnMotion, you must first have Java installed on your machine. While the current download version of BonnMotion can be used on both the Linux and Windows platforms, there appears to be some bugs for the latter version. Hence, the reader is encouraged to use the Linux version. ☺

Unzip and install the BonnMotion file by running the Install script. You will be asked to input the path of your Java.

In general, the syntax for the commands is:
bm <parameters> <application> <application parameters>

The scenario parameters are as follows:

-d <scenario duration>
-i <number of seconds to skip>
-n <number of nodes>
-x <width of simulation area>
-y <height of simulation area>
-R <random seed>

The various application parameters for the different mobility models is given below:

Mobility model	Parameters:
Random Waypoint	-a <attractor parameters> -c [circular] -o <dimension: 1: x only, 2: x or y, 3: x and y>
RPGM	-a <average no. of nodes per group> -c <group change probability> -r <max. distance to group center> -s <group size standard deviation>
Static	-a <attractor parameters> -l <no. density levels>
Manhattan Grid	-c <speed change probability> -e <min. speed> -m <mean speed> -o <max. pause> -p <pause probability> -q <update distance> -s <speed standard deviation> -t <turn probability> -u <no. of blocks along x-axis> -v <no. of blocks along y-axis>
Gauss Markov	-a <angle standard deviation> -h <max. speed> -q <speed, angle update frequency> -s <speed standard deviation>

To create a scenario file, use the following command:
$ bm –f filename MOBILITY-MODEL –n 100 –d 300 –i 3600
where 'MOBILITY-MODEL' can be one of the following:

- RandomWaypoint
- ManhattanGrid
- GaussMarkov
- RPGM
- Static

This will produce 2 files:

- filename.params (contains the parameters used in the simulations)
- filename.movements.gz (contains the movement data)

To convert the output files to those that are compatible with GloMoSim, use the following command:
$ bm GlomoFile –f filename

This will produce 2 files:

- filename.glomo_nodes
- filename.glomo_mobility

which should be placed in the bin folder of GloMoSim.

*The .glomo_nodes and .glomo_mobility files are equivalent to the nodes.input and mobility.in files, respectively. These files should be specified in the config.in file (under NODE-PLACEMENT and MOBILITY) as described in the previous section.