Tips on using ATOM code

1. How to configure simulation parameters?

* The simulation parameter we use is the Inter arrival rate which controls the number of vehicles being inserted in to the network we use. It is given with the router command as the value of “period” whose default value is 1. If the simulation time is 3600 and we give period value 2, then after every 2 minutes of simulation time each vehicle will be inserted into the network according to the network condition and a total of 1800 vehicles will be loaded after 3600 seconds. If we give period value is 0.5, after every 0.5 minute of simulation time, a vehicle will be inserted into the network and a total of 7200 vehicles will be loaded into the network. So use period value to control network congestion.

1. How to run the software?

* According to the network used, check for changes in sumo configuration (cfg) file, the python file and change “period” value as needed. Also in cfg file change the summary file name as needed and in atom.add.xml file change the output file name as per need as well. Then run the atom.sh file.

1. How to check the simulation and collecting numerical data?

* After running the simulation we will get per edge total values in “output” file referred to in atom.add.xml and summary values in “summary” file referred to in cfg file. Also we get the command line output which is the total values of the whole network in the total simulation time. Note down the command line output values like inserted, loaded, route length and duration. To check the real time that was used for the simulation run, you can use the duration parameter in the command line output.

1. How to make the charts using the collected data?

* We have service rate chart, average waiting time chart and average number of stopped vehicles in network chart with x-axis as Inter arrival rate. Then for real road network case alone we had two more charts that is number of vehicles running in the network chart and number of vehicles that completed journey chart all with x axis time.

Service rate is calculated as the ratio of inserted and loaded values in command line output.

Average waiting time is the waiting value in “output” file. Divide it by 1000 to change to seconds from milliseconds.

Average number of stopped vehicles can be obtained from the “summary” file “step\_halting” parameter. You can use the average value.

For the time charts , number of vehicles running in the network can be obtained from “summary” file “step\_running” parameter which shows how many vehicles are running currently in the network. And “step\_ended” shows the number of vehicles that completed their journey so far during the time.