**RC Frame Ground Motion Simulations**

**Instructions Manual**

By: Maha Kenawy

University of Nevada, Reno

Updated: November 24, 2020

This document includes instructions for running simulations of reinforced concrete moment frames subjected to ground motion acceleration series. The first set of instructions covers running in-series response history dynamic simulations of any RC frame subjected to multiple ground motions; the second set covers running the same type of simulations remotely in parallel on NERSC’s CORI.

**Instructions for running multiple GMs in the EQSIM scenario format**

FOR RUNNING MULTIPLE SIMULATIONS

The following files must be in the directory for creating the model, running the simulation and processing the output (this is run in /multipleGM\_simulations)

* Opensees.exe
* Directory with ground motion .data files (If .txt files with a single component are available, skip step 1)
* Excel sheets with building and structural member information
* MATLAB scripts for preparing the ground motions: (prepareGMs.m) – unless the ground motions are already prepared in .txt format.
* MATLAB scripts for generating the structural model and other simulations input (main\_multipleGMs.m, generateModel.m, readBuildingInfo.m, colSectionInfo.m, beamSectionInfo.m, framInfo.m, writeFrameInfo.m, memParameters.m, readcoldata.m, readIMKParameters.m, readbeamdata.m, beamRftGeometry.m, colRftGeometry.m, flexuralStrength.m, shearStrength.m, IMKCalibration.m, processIMKParameters.m)
* Tcl scripts for running the simulations

(multipleGMs.tcl, runGMs.tcl, LumpedModel.tcl, rotSpring2DModIKModel.tcl, LibUnitsNmm.tcl, LibUnits.tcl, LibAnalysisDynamicParameters.tcl)

* MATLAB scripts for getting ground motion intensity measures:

(specDriver.m, integrate.m, spectrum.m, parseDATA\_mk.m, spectra\_T1)

* MATLAB scripts for getting the pulse classification: *classification scripts by Jack Baker and collaborators.*
* MATLAB scripts for postprocessing

(output\_multipleGMs.m, extractMaxDrift.m, plot\_drift\_envelope\_vs\_distance.m, plot\_drift\_vs\_distance\_color.m, hazMap.m, generic\_map.m, plot\_drift\_IMs.m, plot\_drift\_envelopes.m, plotLimits.m, bar\_ticks)

* **STEP 1: Prepare the ground motion files** (this needs to be done once for each ground motion set)
  + Open prepareGMs.m (this file will generate the ground motion time series needed for simulation - if the GMfiles already exist, skip this step)
    - set name of GMset – this is the name of the directory containing the ground motions .data files and should be in the same directory as prepareGMs.m
    - set the component to FN or FP
  + run prepareGMs.m
  + output:
    - GMfiles\_GMsetComp - contains the records for all GMs in .txt format with only the selected acceleration component present.
    - numPts.txt – contains number of points in each record
    - timeincr.txt – contains the time increment of each record
    - pathToGMs.tcl – contains the path to the ground motion files, which will be called by the analysis script runGMs.tcl
* **STEP 2: Generate the structural model**
  + main\_multipleGMs.m (this script generates the actual building model, and spectral ordinates for the building)
    - Set the building name and fundamental period; excel files with the input info for the building and its members must exist first.
    - Set the GMset
    - Set the component (FN or FP)
    - Set a unique name for the simulation set (optional)
  + Run main\_multipleGMs.m
  + Output:
    - frameInfo.tcl – contains the building information
    - memParameters.tcl – contains the structural member information
    - a .mat data file for each column and beam
    - a .mat data file for saving the building info
    - a .mat file for saving SA(T1) and SV(T1)
    - outputDir.tcl – contains the name of the data directory for saving the simulations output, which will be called by runGMs.tcl
* **STEP 3: Run nonlinear response history simulations on Opensees**
  + Run Opensees and source multipleGMs.tcl
  + Output: An output directory named driftOutput\_buildingName\_GMsetComponentSimulation – contains all the building and inter-story drift histories
* **STEP 4: Process output and create plots**
  + output\_multipleGMs.m – this output processing script is separated from the workflow to facilitate extraction of results from previous simulations – the script will call other scripts to generate the SA, SV spectra and PGV, and classify pulse ground motions (if they don’t exist already)
    - Edit user input section appropriately (set buildingName, structPeriod, GMset, component, sim)
    - Set ground motion subset for plots, if desired. The options are:
      * ‘near’: creates plots for GM stations within 10 km normal to fault
      * ‘far’: creates plots for GM stations beyond 10 km normal to fault
      * ‘’: default – creates plots for all stations
    - Set newSim = 1 if extracting results of new simulations; set newSim = 0 if using results extracted and saved previously.
    - Edit the number of x and y ground motion stations and location of the hypocenter if needed
    - Edit plot\_ind to specify indices of specific stations at which drift envelopes will be plotted.
  + Run output\_multipleGMs.m
  + Output:
    - maxDrifts\_buildingName\_GMsetComp.mat – contains maximum drifts
    - GMsetComponent\_SA\_SV\_PGV.mat – contains acceleration and velocity response spectra, and peak ground velocities
    - Pulse\_class\_GMset – contains pulse classification of all ground motions
    - Various plots

**Instructions for running multiple GMs in the EQSIM scenario format on CORI**

STEP 1 is performed in $CSCRATCH directory: The following files must be in that directory for generating the needed ground motion files:

* unzipped directory with ground motion .data files
* MATLAB scripts for preparing the ground motions (prepareGMs.m,) – unless the ground motions are already prepared in .txt format.
* **STEP 1: Prepare the ground motion files** (this step needs to be done once for each ground motion set)
  + Open prepareGMs.m (this file will generate the ground motion time series needed for simulation - if the GMfiles already exist, skip step 1)
    - set name of GMset – this is the name of the directory containing the ground motions .data files and should be in the same directory as prepareGMs.m
    - set the component to FN or FP
  + Open matlab:

salloc -q interactive -N 1 -c 32 -C haswell -t 30:00

module load matlab

matlab

* + prepareGMs
  + output:
    - GMfiles\_GMsetComponent - contains the records for all GMs in .txt format with only the selected acceleration component present.
    - numPts.txt – contains number of points in each record
    - timeincr.txt – contains the time increment of each record
    - pathToGMs.tcl – contains the path to the ground motion files, which will be called by the analysis script runGMs.tcl

STEP 2 is performed on CORI in the $CSCRATCH directory: The following files should be in that directory for running the simulations:

* The directory containing the ground motion files, named as: GMfiles\_GMsetComponent – this is generated in the previous step
* The time increment and number of step files: (timeincr.txt, numPts.txt)
* Tcl scripts containing the building properties

(frameInfo3st.tcl, memParameters3st.tcl, frameInfo12st.tcl, memParameters12st.tcl)

* Tcl scripts for running the simulations

(runGM.tcl, LumpedModel.tcl, rotSpring2DModIKModel.tcl, LibUnitsNmm.tcl, LibUnits.tcl, LibAnalysisDynamicParameters.tcl)

* run.sh – calls opensees
* gen.sh – creates the parallel simulation files and the simulations task list
* batch.sh – allocates resources on CORI
* createSimFiles.py – python file which creates the parallel simulation files
* user\_input.py – to specify the building and GMset and component
* **STEP 2: run the simulations**
* edit user\_input.py - specify the buildingName, GMset, component, sim
* in command line, type *./gen.sh* - this generate the inputlist.txt (this also generates parallelized files and other needed input by running createSimFiles.py)
* change the number of nodes if needed in batch.sh
* *sbatch batch.sh* – submits the job which includes running the parallel simulations

STEP 3 is performed on CORI in the $CSCRATCH directory: The following files should be in that directory for running the simulations:

* Matlab files to extract the drift output and save the maximum drifts: (output\_CORI.m, extractMaxDrift.m (*a function*))
* **STEP 3: extract the output**
  + Load and run the matlab postprocessing script to extract the output files

module load matlab

srun -n 1 -c 32 matlab -nodisplay -r < output\_CORI.m -logfile output\_CORI.log

* download maxDrifts\_buildingName\_GMsetComponentSim.mat to /Opensees\_CORI
* Output:
* maxDrifts\_buildingName\_GMsetComponentSim.mat – contains maximum building and inter-story drifts

STEP 4 is performed locally in /CORI\_simulations, and requires the following files:

* directory with all ground motion .data files (for generating the spectra and pulse classification)
* .mat file containing maximum drifts: maxDrifts\_buildingName\_GMsetComponentSim – downloaded from $CSCRATCH
* MATLAB scripts for post-processing:
* (output\_multipleGMs.m, plot\_drift\_envelope\_vs\_distance.m, plot\_drift\_vs\_distance\_color.m, hazMap.m, generic\_map.m, plot\_drift\_IMs.m, plot\_drift\_envelopes.m, plotLimits.m, bar\_ticks)
* MATLAB scripts for getting ground motion intensity measures:

(specDriver.m, integrate.m, spectrum.m, parseDATA\_mk.m, spectra\_T1)

* MATLAB scripts for getting the pulse classification: *classification scripts by Jack Baker and collaborators.*
* **STEP 4: Process output and create plots** (same as STEP 4 in running simulations locally)
* output\_multipleGMs.m
  + Edit user input section appropriately (set buildingName, structPeriod, GMset, component, sim)
  + Set ground motion subset for plots, if desired. The options are:
    - ‘near’: creates plots for GM stations within 10 km normal to fault
    - ‘far’: creates plots for GM stations beyond 10 km normal to fault
    - ‘’: default – creates plots for all stations
  + Set newSim = 0 (because results have already been extracted on CORI)
  + Edit the number of x and y ground motion stations and location of the hypocenter if needed
  + Edit plot\_ind to specify indices of specific stations at which drift envelopes will be plotted.
* Run output\_multipleGMs.m

Note: to generate new buildings for CORI simulations (locally in CORI\_simulations):

* Set the buildingName in generateModel\_CORI.m – must match name on design excel sheets in the same directory
* Run generateModel\_CORI.m
* Move frameinfoBuildingName.tcl and memParametersBuildingName.tcl to $CSCRATCH/Opensees\_models
* Output:
  + frameinfoBuildingName.tcl – defines building properties
  + memParametersBuildingName.tcl – defines structural member properties
  + buildingName\_info.mat – saves some building properties for output processing