**RC Frame Simulations**

**Instructions Manual**

By: Maha Kenawy

University of Nevada, Reno

March 3, 2020

This document includes instructions for running different types of simulations of reinforced concrete moment frames subjected to various loading conditions. 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 in parallel on NERSC’s CORI. The third set covers instructions for running a single response history simulation when a detailed output is desired (e.g., moments and rotations at all beam-column joints), and the final set covers running simulations of other types of analyses, including pushover, cyclic loading, free vibration, and dynamic analysis.

**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

**Instructions for running a ground motion simulation with detailed output**

DETAILED BUILDING OUTPUT

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

* Opensees.exe
* Directory with one ground motion .txt file with a single component named in the same format as before (GMfiles\_GMsetComponent)
* Excel sheets with building and structural member information
* timeincr.txt and numPts.txt files
* MATLAB scripts for generating the structural model and other simulations input (main\_detailedGMs.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 postprocessing

(MomRot.m, memRotLim.m, connect.m)

* **STEP 1: Run everything through main\_detailedGMs**
  + Set basic parameters in main\_detailedGMs.m
    - Set the building name, GMset, component (FN or FP)
    - Set name for the simulation set after the selected ground motion (technically you can run multiple ground motions, but the output script MomRot would need to be reconfigured to separate multiple ground motion outputs)
  + Set output plot parameters in MomRot.m (optional)
    - plottype = ‘def’ for deformed shape, or ‘undef’ for undeformed shape
    - large = numeric scale of plastic hinge rotation circles
    - dispscale = numeric scale of displacements
    - timestep = step at which to plot deformations and rotation values (line 105 of script)
  + Run main\_detailedGMs.m
  + Output:
    - Plot of deformed shape and plastic hinge rotations

Note: to plot already existing results, make sure to set the correct buildingName, GMset, component, and sim in MomRot.m, and run MomRot with newSim = 0.

**Instructions for running other types of single building simulations**

Analysis types: pushover, cyclic, free vibration, dynamic, dynamic pushover

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

* Opensees.exe
* Excel sheets with building and structural member information
* MATLAB scripts for generating the structural model and other simulations input (main.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 postprocessing

(postprocess.m)

* **STEP 1: Run everything through main**
  + Set basic parameters in main.m
    - Set the building name
    - Set the analysis type: pushover, cyclic, freevib, dynamic, dynamicpushover
    - Set a name for the simulation (optional)
  + Run main.m
  + Output:
    - Plots that depend on the analysis type