NISRAF

NEES Integrated Seismic Risk Assessment Framework

User Manual

Beta Version

by

Sheng-Lin Lin
Jian Li
Amr S. Elnashai
Billie F. Spencer

DEPARTMENT OF CIVIL AND ENVIRONMENTAL ENGINEERING
UNIVERSITY OF ILLINOIS AT URBANA-CHAMPAIGN
URBANA, ILLINOIS

JUNE 2011

TABLE OF CONTENTS

1. Introduction 1
2. Installation of NISRAF 2
3. Architecture of NISRAF 3
3.1 File Menu 5
3.2 Strong Motion Menu 8
3.3 Hazard Characterization Menu 9
3.3.1 Seismic Hazard Analysis 9
3.3.2 Synthetic Time History 16
3.3.3 Hazard Map Generation 20
3.4 Structure Model Menu 22
3.4.1 Import from Zeus file 22
3.4.2 New Model 23
3.4.3 New Model from Template 24
3.4.4 View 24
3.4.5 Structure Model 29
3.5 Model Calibration Menu 35
3.5.1 Modal Analysis 35
3.5.2 System Identification 36
3.5.3 Model Updating 37
3.6 Hybrid Simulation Menu 38
3.6.1 Dynamic Load 39
3.6.2 Static Load 40
3.6.3 Hybrid Model 40
3.6.4 Simulation 45
3.6.5 Results 46
3.7 Fragility Analysis Menu 47
3.7.1 Define Limit States 47
3.7.2 Run Hybrid Simulation 48
3.7.3 Hybrid Fragility Curves 48
3.8 Impact Assessment Menu 49
3.8.1 MAEviz 49
3.8.2 HAZUS 49
3.9 Help Menu 50
4. Tutorial Examples 50
4.1 Introduction 50
4.1.1 Building Information 50
4.1.1 Site Condition 51
4.2 Strong Motion 53
4.3 Hazard Characterization 53
4.3.1 Synthetic Ground Motion 53
4.3.2 Hazard Map 55
4.4 Structure Model 56
4.5 Model Calibration 56
4.5.1 System Identification 56
4.5.2 Model Updating 57

4.6 Hybrid Simulation & Fragility Analysis 59
4.6.1 Performance Limit State 59
4.6.2 Seismic Input 59
4.6.3 Hybrid Simulation 59
4.6.4 Hybrid Fragility Analysis 60
4.7 Impact Assessment 62
5. Reference 63

NISARF Update History

April 2010

- Hazard@NISRAF Beta Version release

June 2011

- NISRAF Beta Version release

1. Introduction

NEES Integrated Seismic Risk Assessment Framework (NISRAF), a completed MATLAB (The MathsWork, Inc.) GUI-driven software platform has been developed for the purpose of making impact assessment more efficient and more reliable. Several components—instrumentation, advanced hazard characterization, system identification, model updating, hybrid simulation, advanced hybrid fragility analysis and impact assessment tool—have been implemented and tailored with novel methods to build the seamless, transparent and extensible framework. Below, the architecture, methodologies, communication protocols and analysis platforms of NISRAF are discussed first. Next, a tutorial of NISRAF will be presented.

This document is available only for the beta version of NISRAF. The beta version of NISRAF is released to get feedback from various users for improvement of NISRAF. There could be unexpected bugs in the beta version of NISRAF, so it is desirable to use NISRAF only for simulation cases. If there are any questions or comments, please feel free contact with main developers of NISRAF (Sheng-Lin Lin, shenglin0110@gmail.com ; Jian Li, jianli3@illinois.edu).

2. Installation of NISRAF

To install NISRAF, run the executable installation file (i.e., NISRAFSetup.exe). **When you install software, do not change the default folder location. The software should be installed in C:\NISRAF**. After installation is complete, the user should be able to run the examples illustrated in this document.

The installed NISRAF folder should be in the search path of MATLAB. To update MATLAB path, run a MATLAB and click ***File-Set Path-Add with Subfolders*** in the menu bar and select *C:\NISRAF* and click ***OK*** button.

NISRAF has been developed based on MATLAB R2009a (v7.8.0), so there could be unexpected bugs when it is running in lower version of MATLAB.

3. Architecture of NISRAF

As shown in Figure 3-1 and Figure 3-2, free-field measurements (**I1**) along with nonlinear site response analysis (**SR**) are used to generate the advanced hazard map and ground motion records (**AH**). The measured and synthetic records are then used in hybrid simulation and fragility analysis. Meanwhile, the structural model is calibrated with the measured structural response (**I2**). Next, hybrid simulations (**HS**) are performed with the most critical component of the structural system tested in the laboratory and the remainder of the structure simulated analytically. These simulations are conducted to derive the mean seismic intensity value (PGA, for example) of the corresponding performance limit state. The fragility curves (**FA**) of the structure are then generated using the hybrid simulation data and the dispersions from the literature. Finally, the hybrid fragility curves and the calibrated hazard map are fed into the impact assessment tool, such as MAEviz or HAZUS (**IA**).

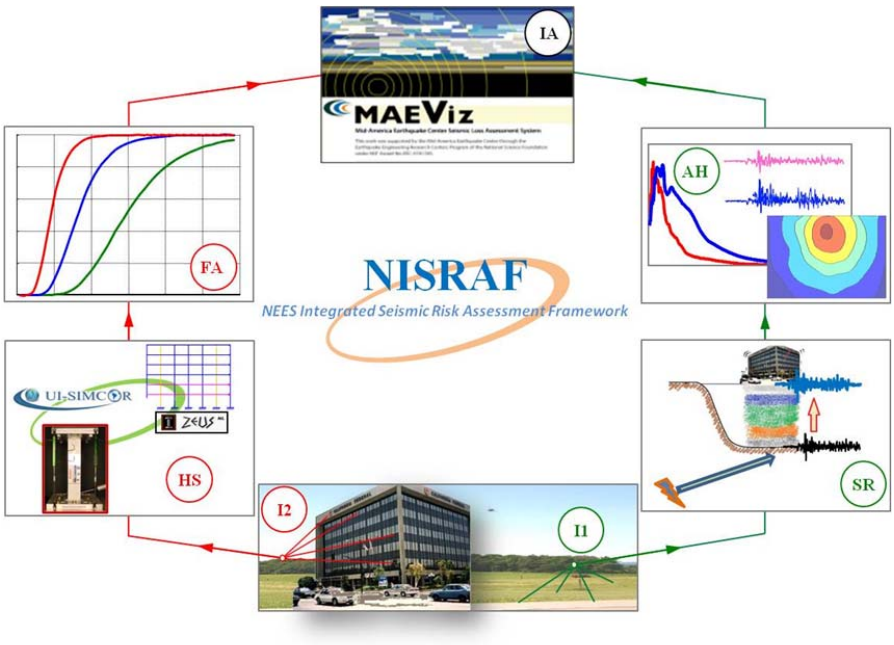


Figure 3-1 Schematic of the proposed integrated framework

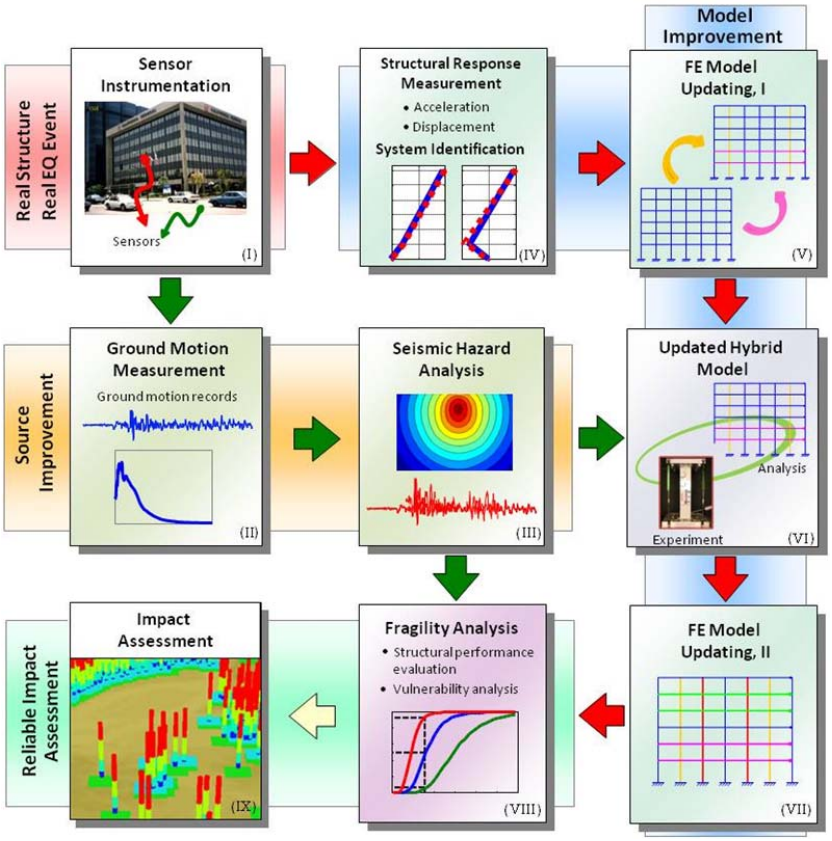


Figure 3-2 Architecture of NISRAF

Clearly, NISRAF is composed of five main parts: namely, (i) instrumentation (**I1**, **I2**), (ii) seismic hazard analysis (**AH**), (iii) model calibration and hybrid simulation (**HS**), (iv) fragility analysis (**FA**) and (v) earthquake impact assessment (**IA**). For ease of use, nine main menus with submenus are designed and arranged, following the analysis sequences (Figure 3-3):

- ***File:*** Contains general menus (such as Open, Save, Save As, Page Setup, Print Review, Print and Exit).
- ***Strong Motion:*** Provides an interface to download measured data from instrumentation databases (ANSS, COSMOS, CESMD and PEER).
- ***Hazard Characterization:*** Contains three menus (Seismic Hazard Analysis, Synthetic Time Histories and Hazard Map Generation) to perform hazard analysis.
- ***Structure Model:*** Contains five menus (Import from ZEUS File, New Model from Template, New Model, View and Structure Model) to import, develop and view the FE model.
- ***Model Calibration:*** Contains three menus (Modal Analysis, System Identification and Model Updating) to improve the FE model.
- ***Hybrid Simulation:*** Contains five menus (Dynamic Load, Static Load, Hybrid Model, Simulation and Results) to develop the hybrid model, run simulation and check results.

- ***Fragility Analysis:*** Contains three menus (Define Limit States, Run Hybrid Simulation and Hybrid Fragility Curves) to derive hybrid fragility relationships through hybrid simulation testing.
- ***Impact Assessment:*** Contains two menus (MAEviz and HAZUS) to perform the earthquake impact assessment.
- ***Help:*** Contains three menus (NISRAF Manual, UI-SIMCOR Manual, and About NISRAF) to assist users in performing the analysis. Copyright and version information are also included here.

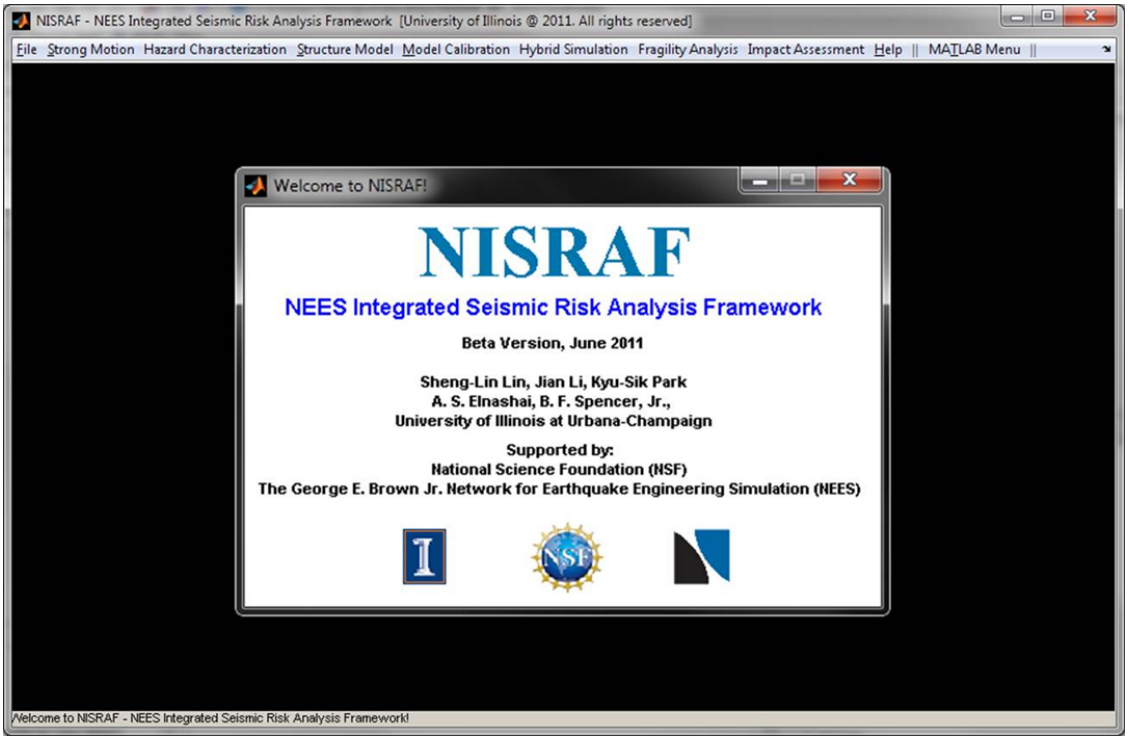


Figure 3-3 Welcome window and main window of NISRAF

In NISRAF, each main menu is modularized. Moreover, each method and algorithm implemented in sub-menu is also developed in module unit. This module feature makes it easy to understand analysis algorithms as well as to maintain this versatile and integrated program. Most importantly, it enables the latest research finding and computation techniques to be easily implemented. Below, development of each main menu is presented with a focus on the novel manners used to tailor and integrate components to build the seamless framework.

3.1 File Menu

File menu contains the general menus, such as **Open**, **Save**, **Save As**, **Page Setup**, **Print Review**, **Print** and **Exit**, as shown in Figure 3-4. These submenus provide the basic functionalities to manage files, such as opening an existing file, saving and printing the current working file, and exiting and closing NISRAF.

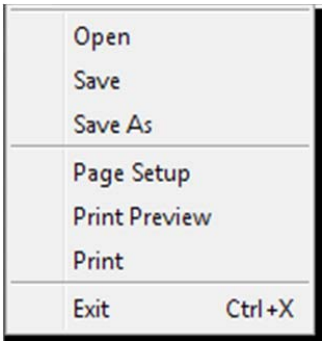
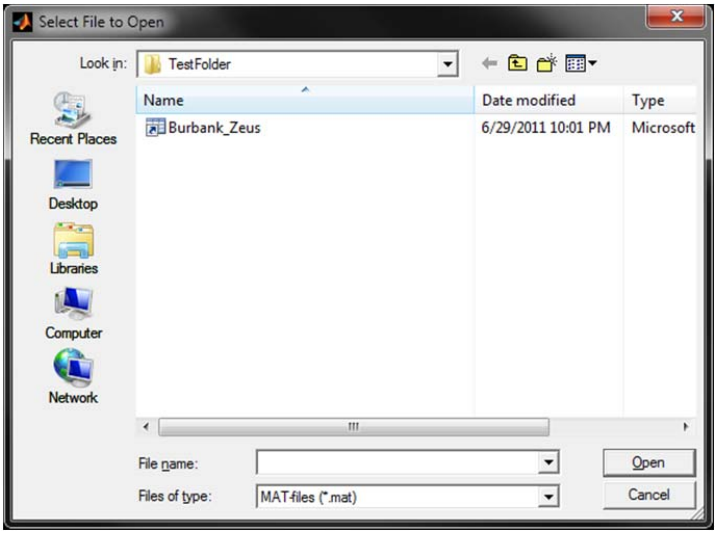


Figure 3-4 File submenus

- *File-Open*: Open existing NISRAF file

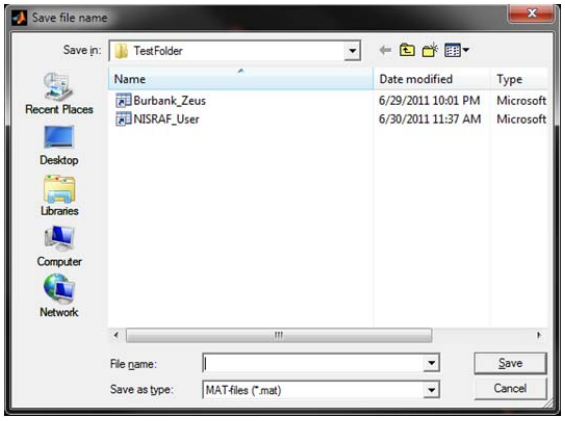


When existing NISRAF file is opened, NISRAF data (strong motion, hazard, structural information, fragility, and other information) will be loaded.

- *File-Save*: Save NISRAF data

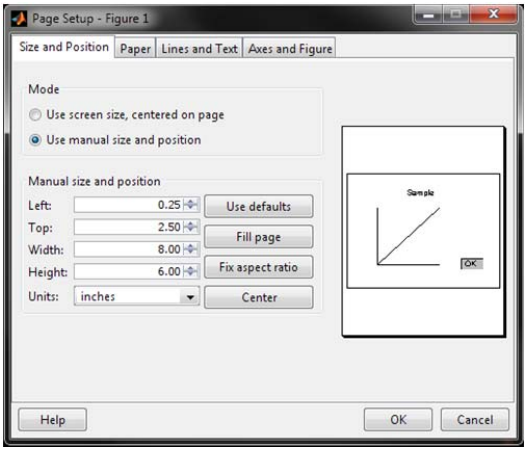
Current NISRAF data will be saved in as default file name (i.e., NISRAF_User.mat).

- *File-Save as*: Save NISRAF data as user defined file name

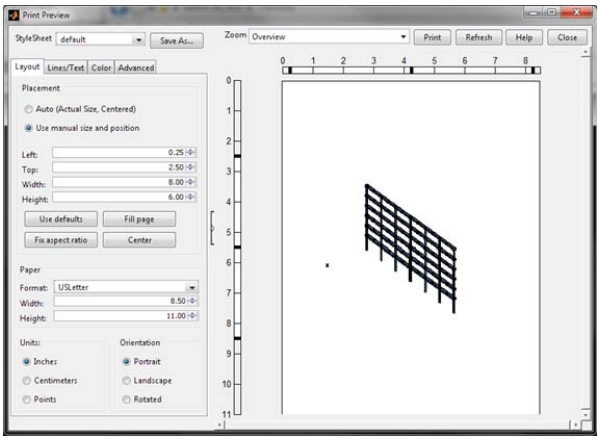


Current NISRAF data will be save in user defined file name (i.e., filename.mat).

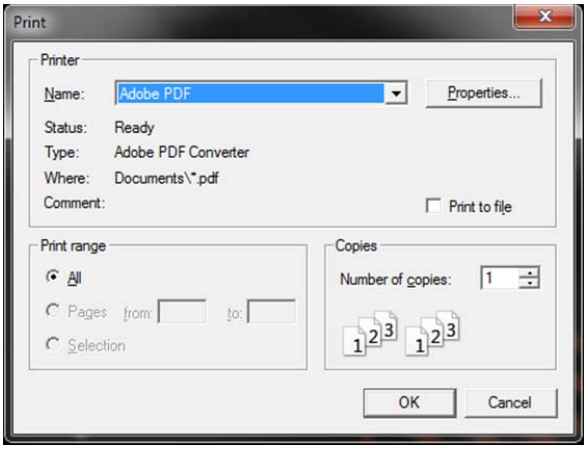
- *File-Page Setup*: Setup page for printing



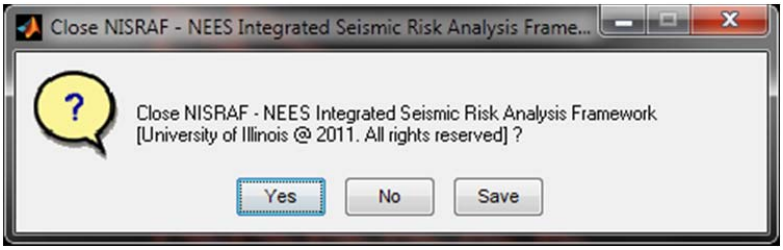
- *File-Print Preview*: Print preview



- *File-Print*: Print NISRAF GUI



- *File-Exit*: Exit NISRAF



3.2 Strong Motion Menu

In Strong Motion menu, as shown in Figure 3-5, the user is prompted to connect to a web-based instrumentation database. Through this linkage, the user can easily download records. Meanwhile, NISRAF allows the user to create a new folder to deposit the downloaded records as well as other basic project information, which facilitates maintenance. Two different types of records are required to perform analysis in NISRAF. Ground motion (free-field) records are used to calibrate hazard models, while structural measurements are used to calibrate structural models. The incorporation of the instrumented data into NISRAF is not only to increase its usage, but also to improve hazard and structural model.

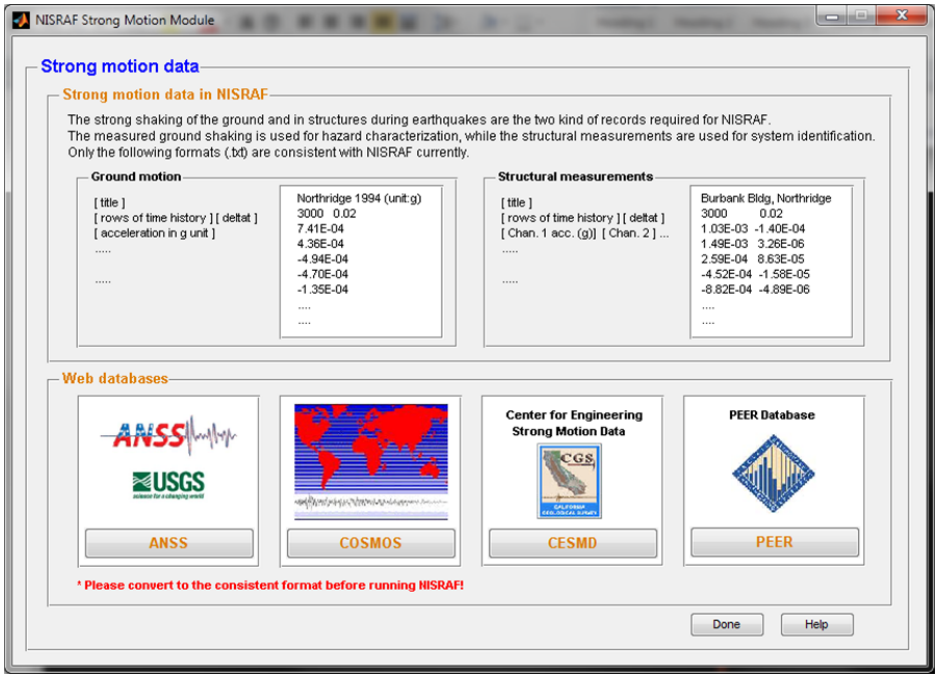


Figure 3-5 Strong motion data GUI in Strong Motion menu

3.3 Hazard Characterization Menu

Hazard characterization menu is composed of three main parts: namely, Seismic Hazard Analysis, Synthetic Time Histories and Hazard Map Generation, as shown in Figure 3-6. Methodologies and analysis procedures of hazard characterization analysis have already been illustrated and verified in Lin (2010). One of the features of the proposed advanced hazard analysis approach is its ease of use. By tailoring the hazard models as well as ensuring connection and compatibility between them, it simplifies the complicated and tedious procedures in the conventional analysis. Consequently, with an interactive interface to define inputs, hazard analysis becomes efficient and straightforward. Below, analysis procedures in the three submenus are presented with GUIs and illustrations.

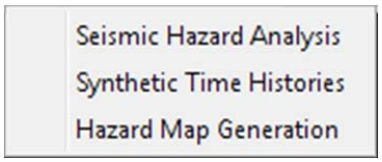


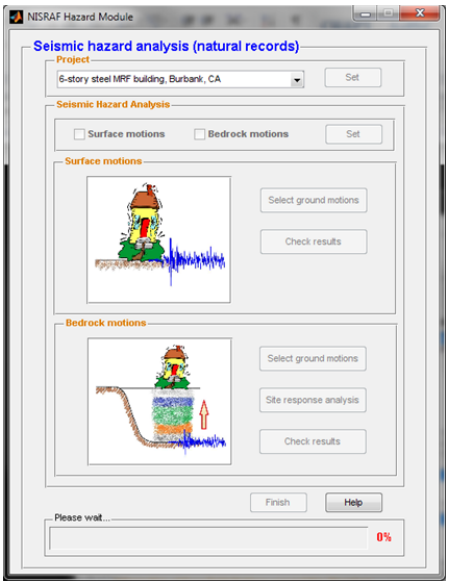
Figure 3-6 Hazard Characterization submenus

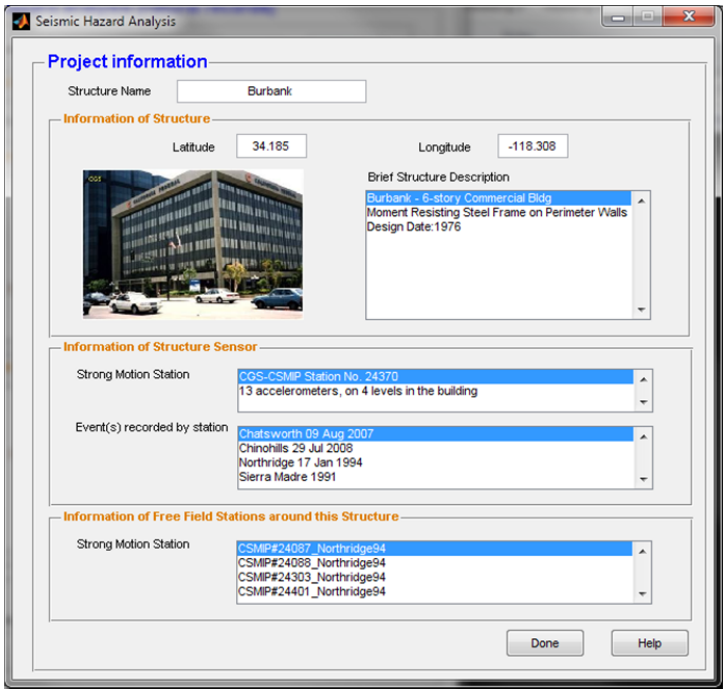
3.3.1 Seismic Hazard Analysis

In Seismic Hazard Analysis, user can check the natural records on the surface, and perform local site effect for records on the bedrock. After clicking Seismic Hazard Analysis, user will be prompted to select project for analysis. Based on the selected project, NISRAF will list all the information and recorded strong motions for this

structure. Afterward, users can check the surface records or perform local site effect on the bedrock records.

- Click ***Set*** to confirm the selected project (i.e., 6-story steel MRF building, Burbank, CA).

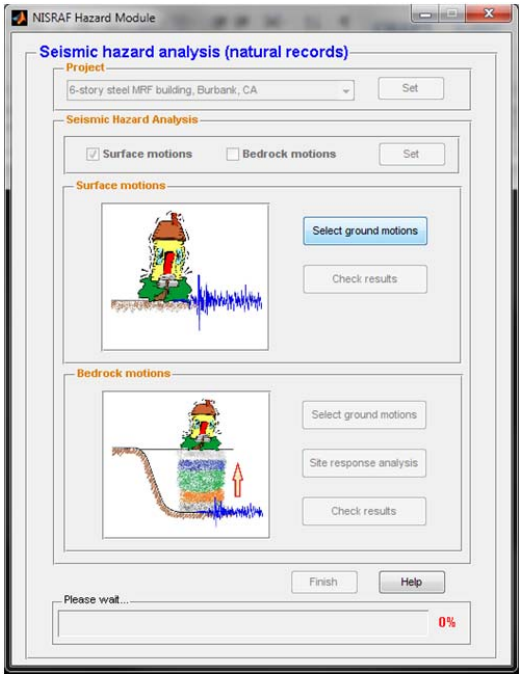




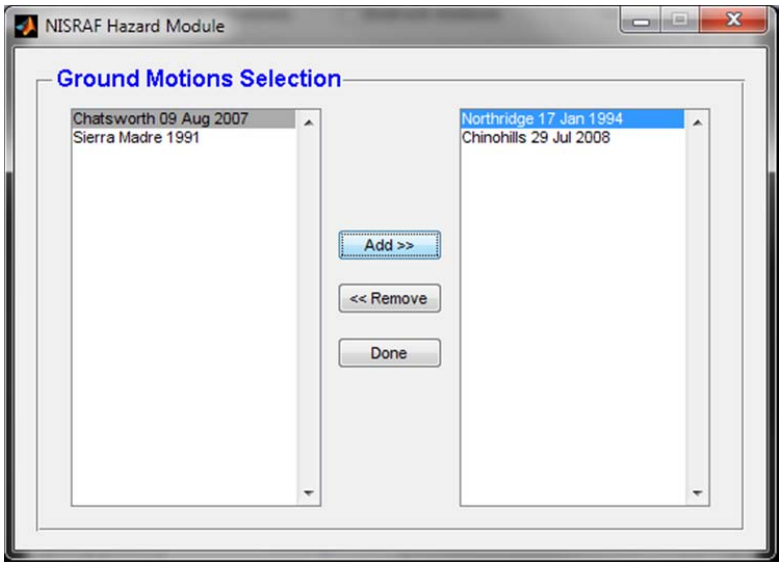
- Choose surface motions and/or bedrock motions for further analysis. Click ***Set*** to confirm.

3.3.1.1 Surface motions

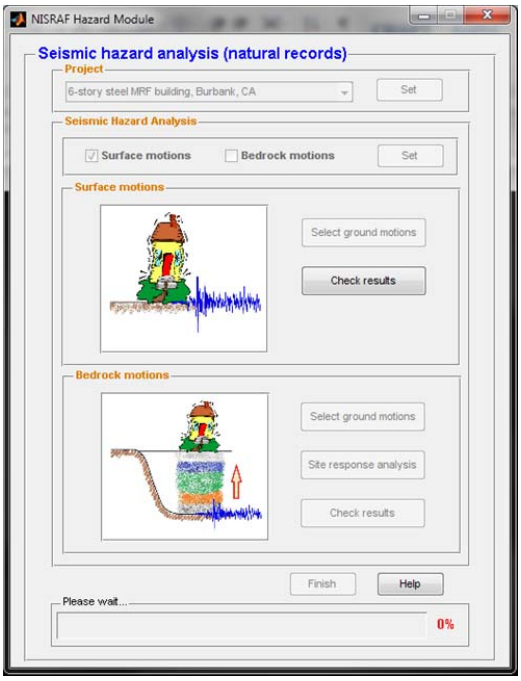
- Click ***Select ground motions***

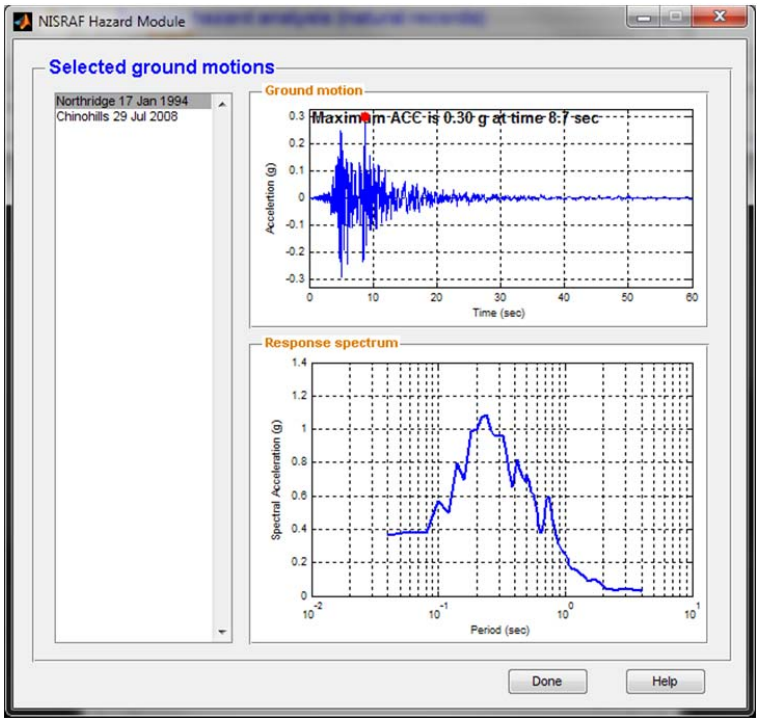


- Select surface motions, click ***Done*** to continue



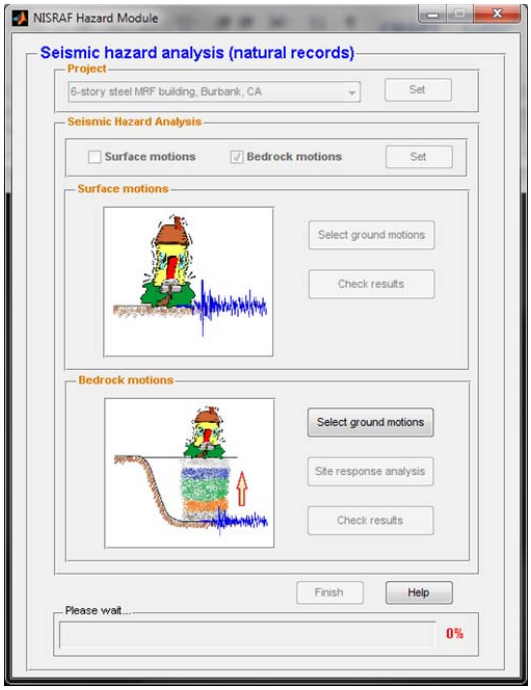
- Click ***Check results*** to check time history and response spectra for the selected motion



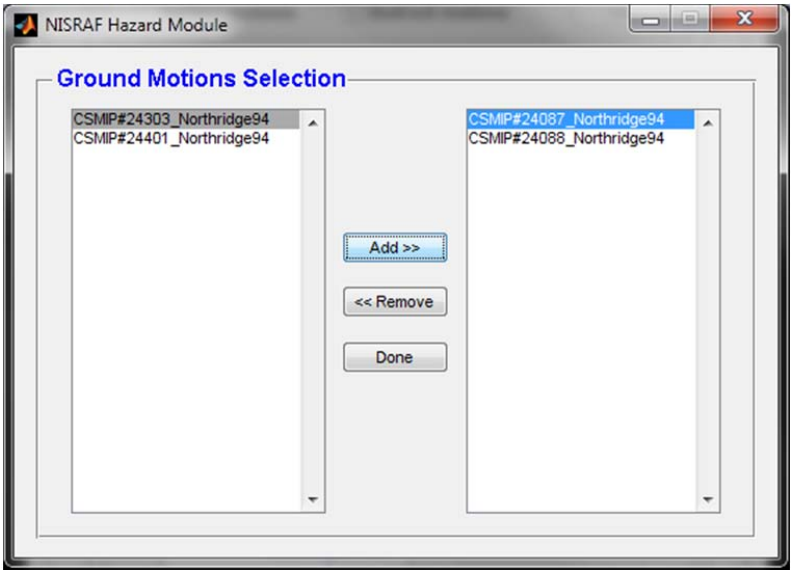


3.3.1.2 Bedrock motions

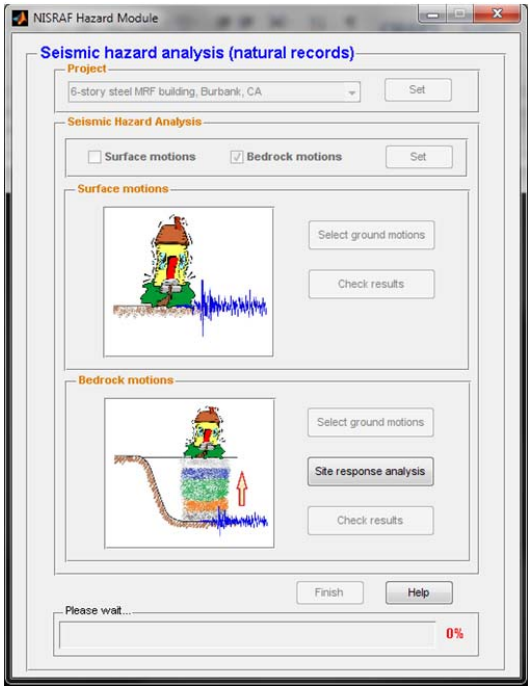
- Click ***Select ground motions***



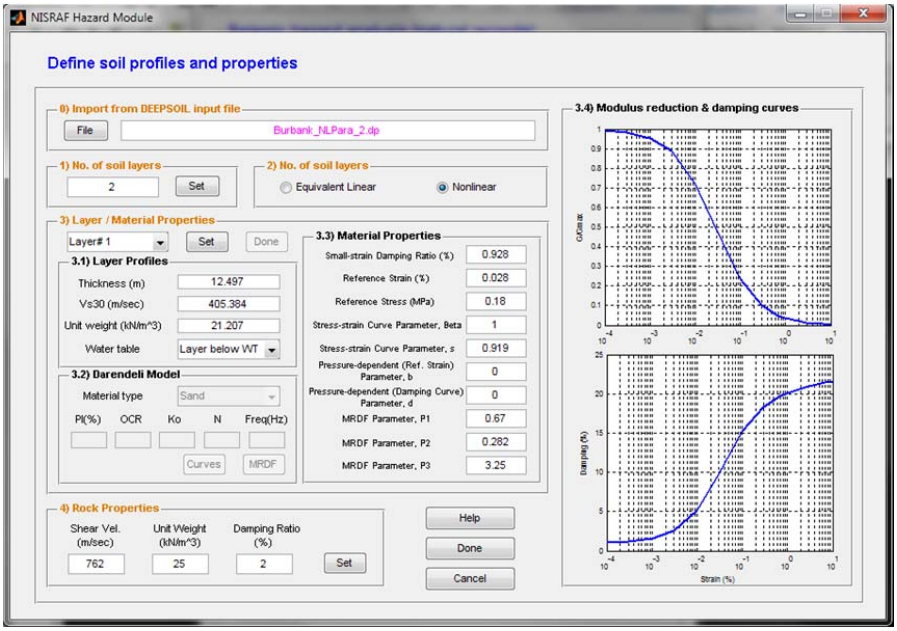
- Select bedrock motions for analysis, Click ***Done*** to continue



- Click ***Site response analysis***



- Before performing site response analysis, user need to define soil profiles and material properties

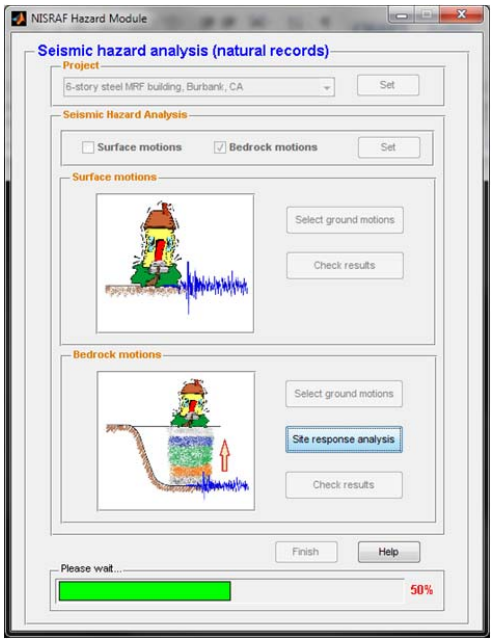


To define soil profiles and material properties, user can import input files for DEEPSOIL (*.dp files, for example, Burbank_DP_2.dp and Burbank_NLPara_2.dp in

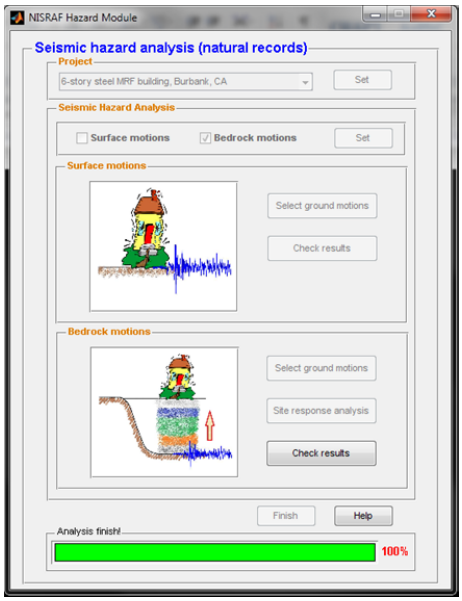
C:\NISRAF\Hazard\Projects\Burbank) or define step by step. Users need to follow the number of each panel to fill in all the parameters. For more information about each parameter, users can refer to the user manual of DEEPSOIL (Hashash et al., 2009).

After defining all the parameters, a DEEPSOIL file (*.dp) will be generated and saved into current folder.

- After that, NISRAF will perform site response analysis



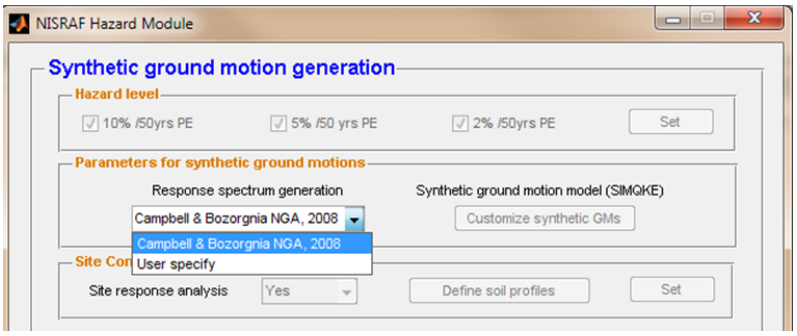
- Click ***Check results*** to check surface motions with local site effect



3.3.2 Synthetic Time History

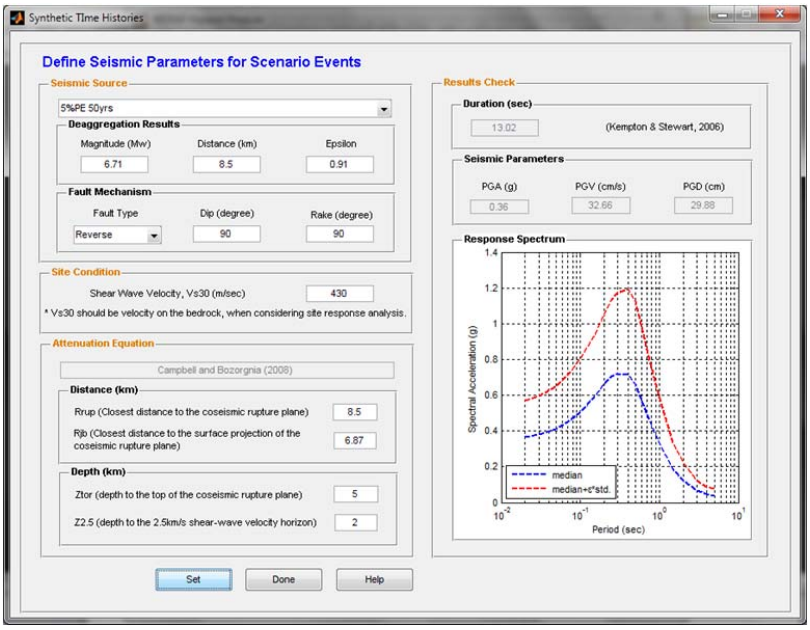
NISRAF allows user to generate synthetic time histories for three different hazard levels (i.e., 10%/50 yrs PE, 5%/50 yrs PE, or 2%/50 yrs PE). The following describes more details for each step and the required information.

- Select the hazard level for synthetic time histories, click ***Set*** to continue
- Response spectrum generation, users can generate spectrum by NGA model or define the spectrum by discrete points



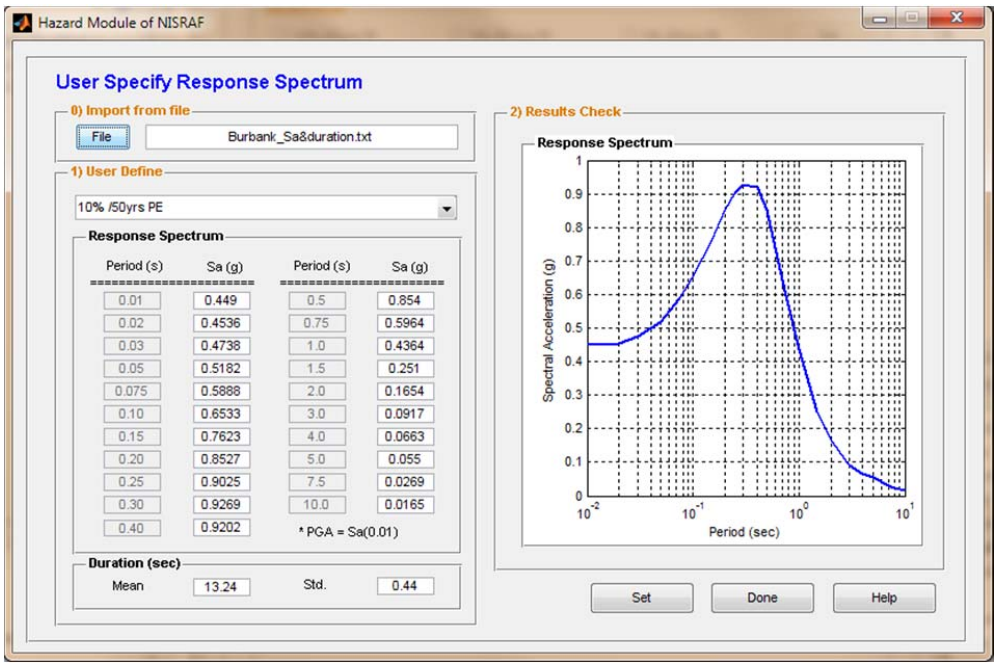
- **Using NGA model to generate spectrum:**
  Here user only provides parameters for Seismic Source and Site Condition panels. Click ***Set*** to save parameters and check duration and spectrum for each hazard level, and click ***Done*** to next step.

  User can get information for seismic source panel by conduct deaggregation analysis for the project site.

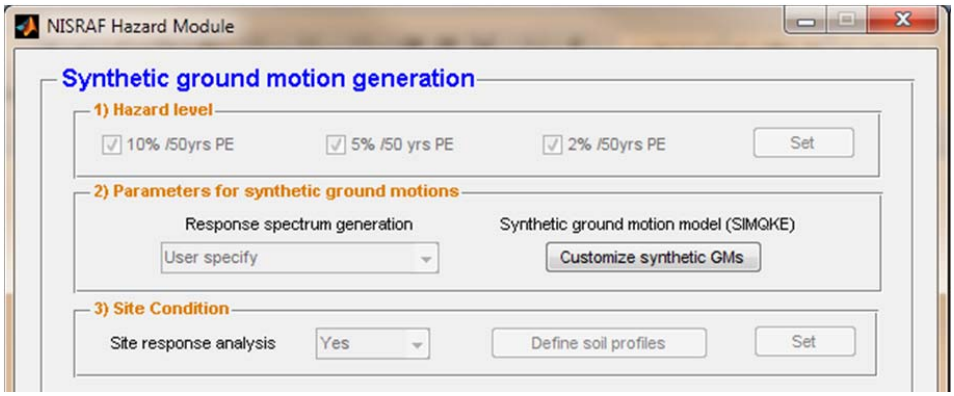


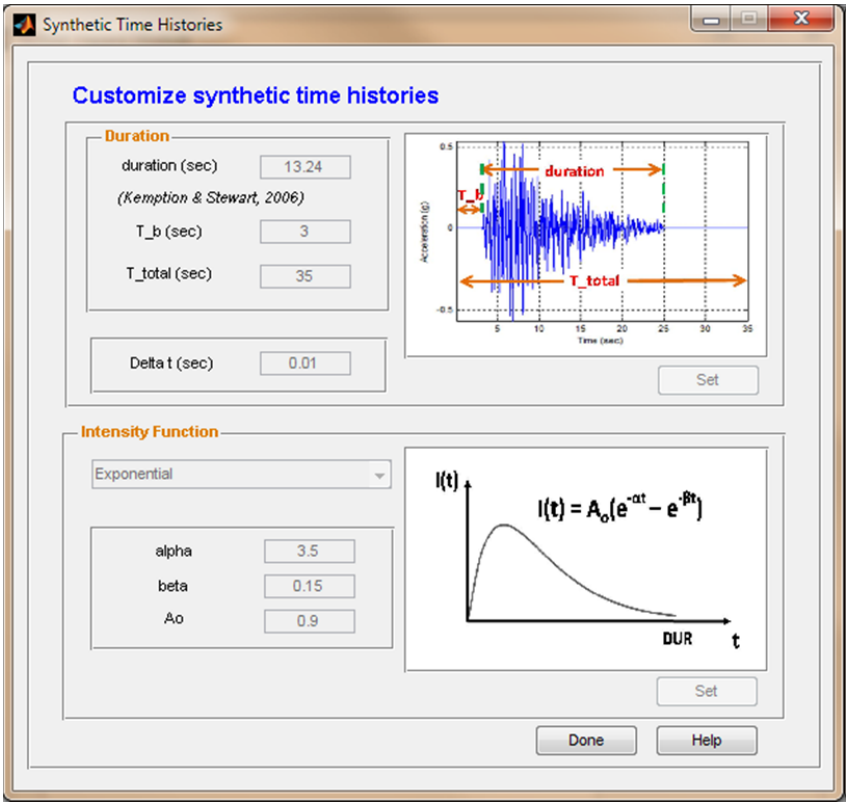
- **User specify spectrum:**
  Here user can define spectrum by importing from *.txt files or define the discrete points through the interface.

  When importing from .txt files, users need to pay attention the format requirement for the files. Please refer to Burbank_Sa&duration.txt under *C:\NISRAF\Hazard\Projects\Burbank* for the compatible format.

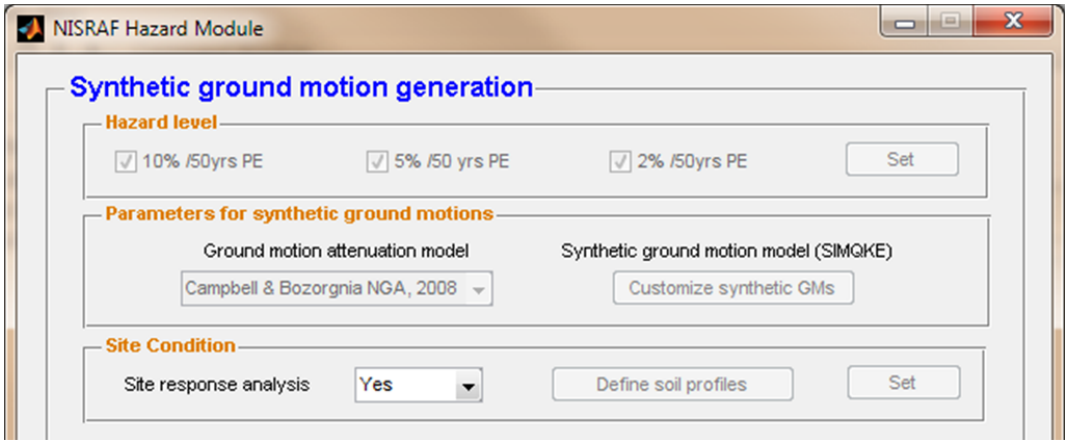


- Click ***Customize synthetic GMs*** to define the intensity function for synthetic ground motions

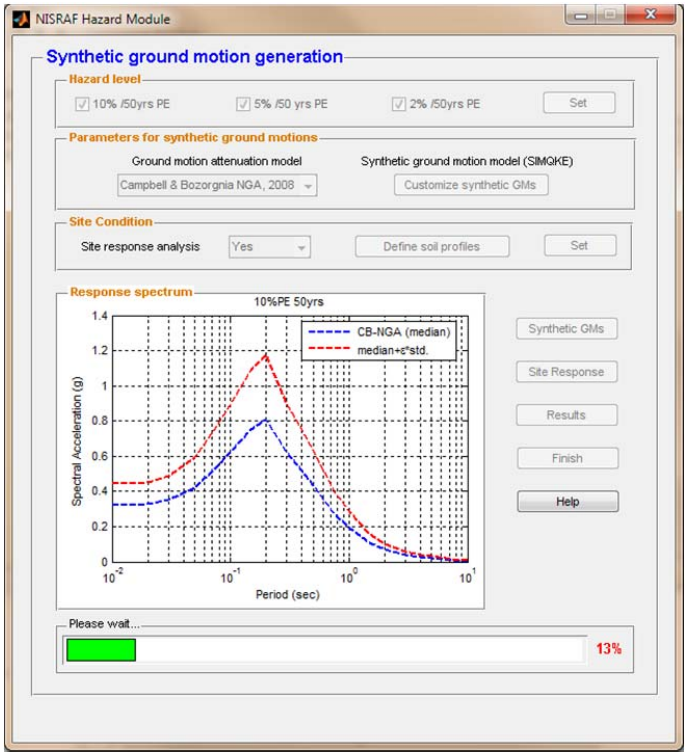




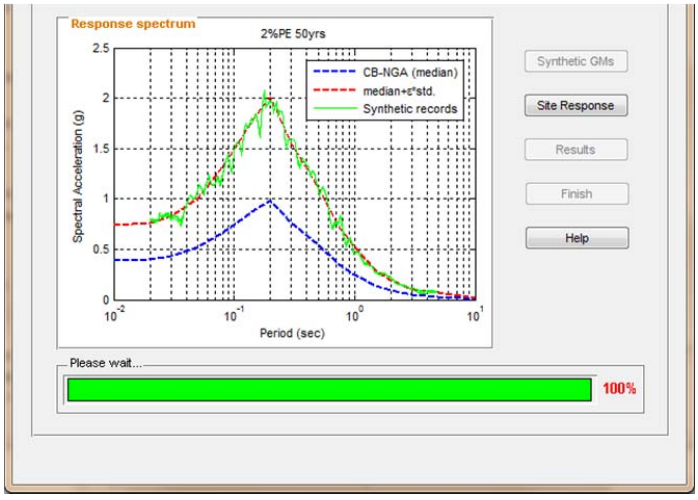
For Site Condition panel, users decide if the site response analysis will be considered or not. Users need to provide soil profiles and material properties if the site response analysis is considered. Please refer to section 3.3.1.2 for more detail about the definition of soil condition.



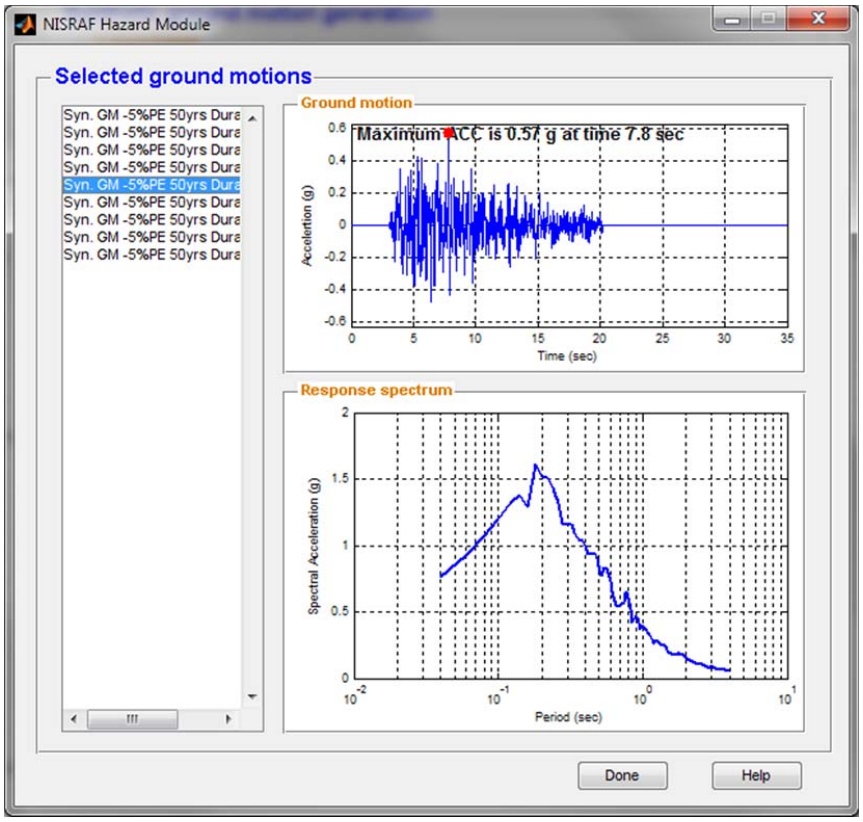
- Click ***Site Response*** to conduct local site effect. GMs with different duration will be generated and saved as *.txt files in *C:\NISRAF\StrongMotion\[Project name]\Synthetic Ground Motion*



- Click ***Site Response*** to conduct local site effect. GMs with different duration will be generated and saved as *.txt files in *C:\NISRAF\StrongMotion\[Project name]\Synthetic Ground Motion\Syn_Conv*



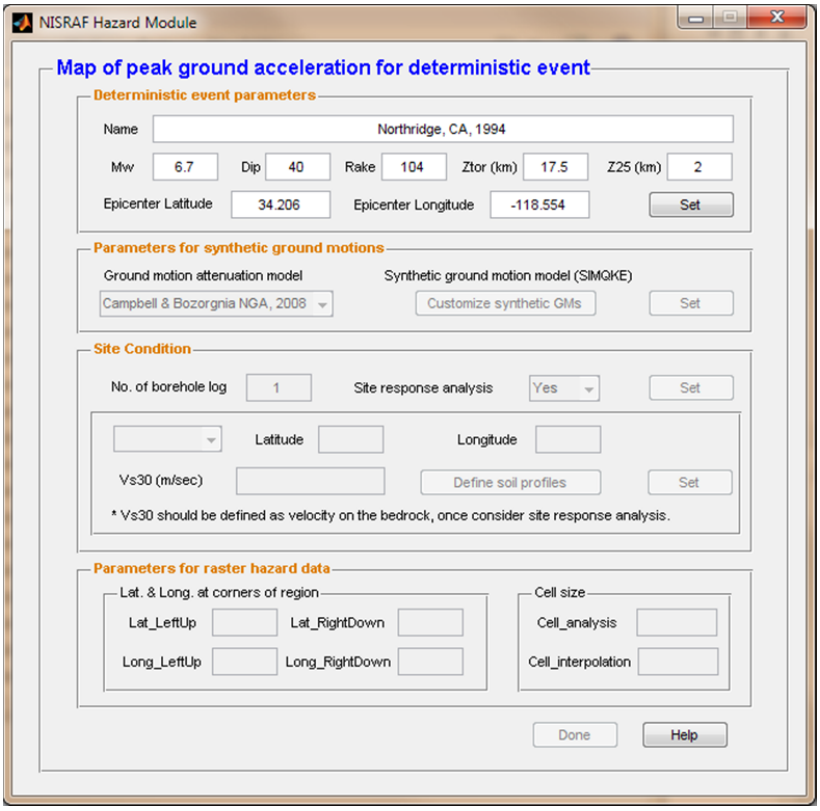
- After all the analysis, users can check the time histories and response spectrum for each ground motion.

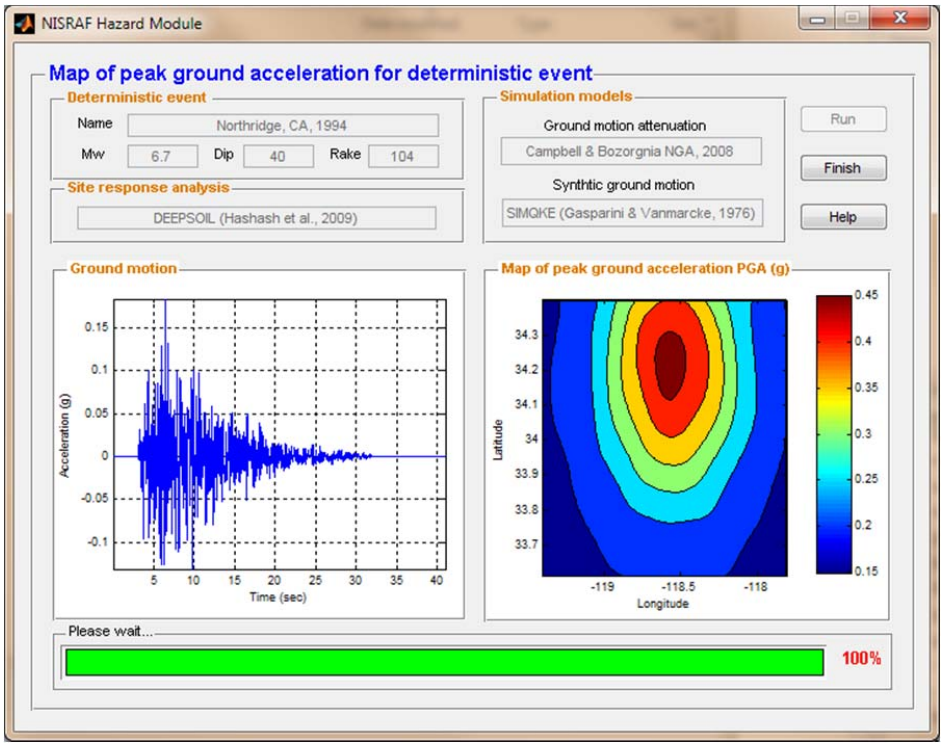


3.3.3 Hazard Map Generation

NISRAF allows user to generate hazard map for deterministic event. The following describes more details for each step and the required information.

- First, user need to provide parameters for the scenario event.
- Second, define parameters for synthetic ground motions and site condition (please refer to previous section for more detail)
- In addition, users need to provide information for the boundary of the map and the size of cell.
- After defining all the parameters, hazard map will be generated and saved into current folder (*.asc)


NISRAF Hazard Module
Map of peak ground acceleration for deterministic event
Deterministic event parameters
Name
Northridge, CA, 1994
Mw
6.7
Dip
40
Rake
104
Ztor (km)
17.5
Z25 (km)
2
Epicenter Latitude
34.206
Epicenter Longitude
-118.554
Set
Parameters for synthetic ground motions
Ground motion attenuation model
Campbell & Bozorgnia NGA, 2008
Synthetic ground motion model (SIMQKE)
Customize synthetic GMs
Set
Site Condition
No. of borehole log
1
Site response analysis
Yes
Set
Latitude
Longitude
Vs30 (m/sec)
Define soil profiles
Set
* Vs30 should be defined as velocity on the bedrock, once consider site response analysis.
Parameters for raster hazard data
Lat. & Long. at corners of region
Lat_LeftUp
Lat_RightDown
Long_LeftUp
Long_RightDown
Cell size
Cell_analysis
Cell_interpolation
Done
Help


NISRAF Hazard Module
Map of peak ground acceleration for deterministic event
Deterministic event
Name
Northridge, CA, 1994
Mw
6.7
Dip
40
Rake
104
Site response analysis
DEEPSOIL (Hashash et al., 2009)
Simulation models
Ground motion attenuation
Campbell & Bozorgnia NGA, 2008
Synthtic ground motion
SIMQKE (Gasparini & Vanmarcke, 1976)
Run
Finish
Help
Ground motion
Acceleration (g)
Time (sec)
Map of peak ground acceleration PGA (g)
Latitude
Longitude
Please wait...
100%

3.4 Structure Model Menu

The finite element model is a prerequisite for model calibration. To create an FE model, the user is allowed to import an existing ZEUS-NL model or create a new model in NISRAF. Submenus for creating an FE model (such as **Import from Zeus file**, **New Model from Template**, **New Model**, **View** and **Structural Model**) are based on SimBuild (Park et al., 2007), a pre- and post-processor for UI-SIMCOR.

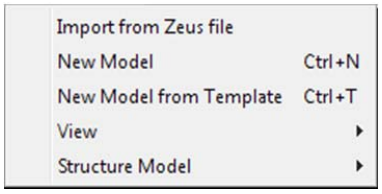
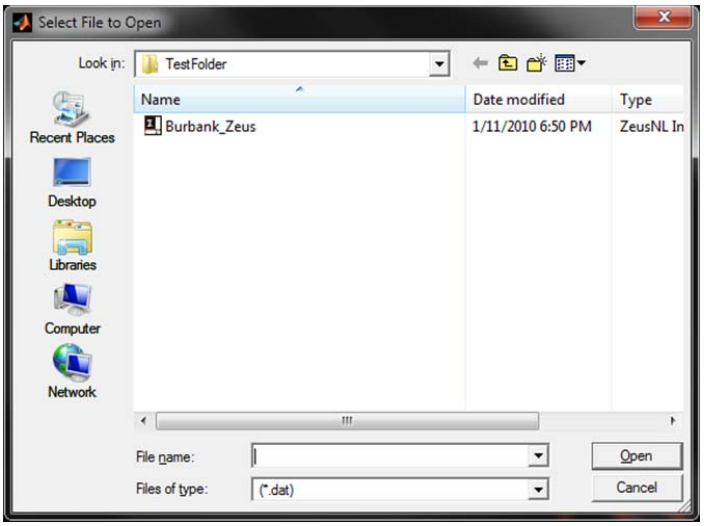


Figure 3-7 Structural Model submenus

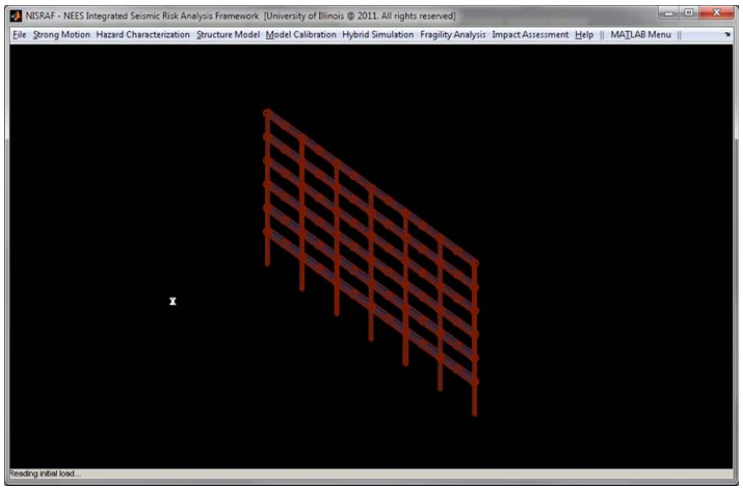
3.4.1 Import from Zeus file

NISRAF allows user to import existing Zeus file and transfer all the structural information required for NISRAF.

- After clicking ***Import from Zeus file***, select the existing Zeus file



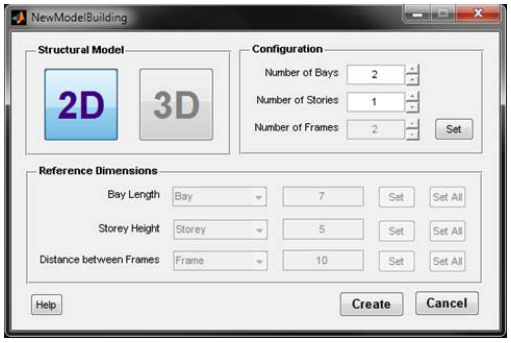
- NISRAF collect structural information from Zeus file and present through its interactive interface



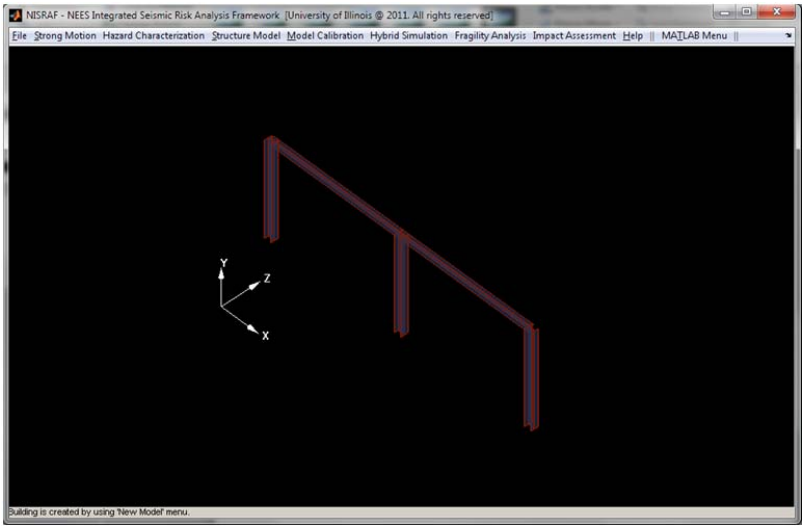
3.4.2 New Model

NISRAF allows user to create new structure model.

- User can select structure type as building or bridge.
- Bridge type structure is implemented as test purpose in this version of NISRAF, so there could be unexpected bugs in the bridge structure.
- If building type is selected, following GUI for creating building structure will be shown.



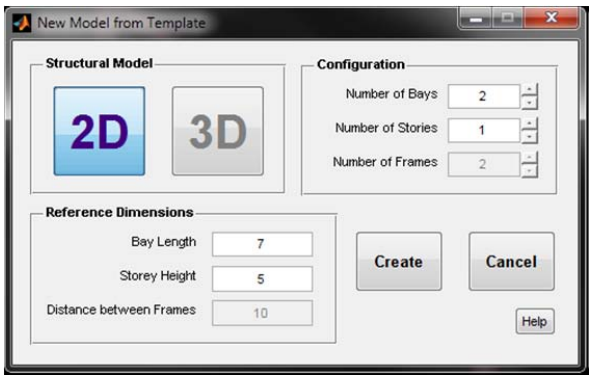
- When structure is created, NISRAF GUI is updated as follows.



3.4.3 New Model from Template

NISRAF also allows user to create structure model from template.

- If building type is selected, following GUI will be shown.



- This menu is similar to Structure Mode/New Model, but more simple which the properties will be defined as same as all components (i.e., bay length, story height, frame distance).

3.4.4 View

User can check the structural information such as node, element, boundary condition, and others by using **View** menu. Hybrid model including simulation platform of each substructure can be shown in this menu by using **Hybrid Model** submenu. Disabled submenus will be enabled when it is available.

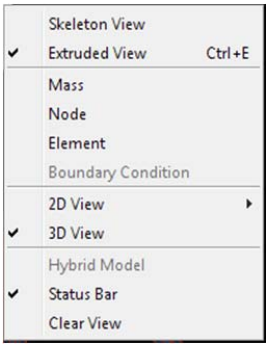
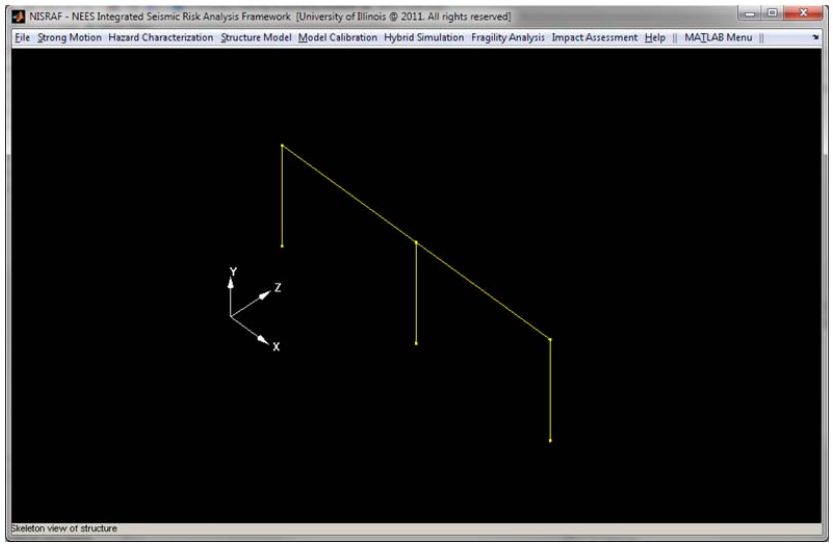
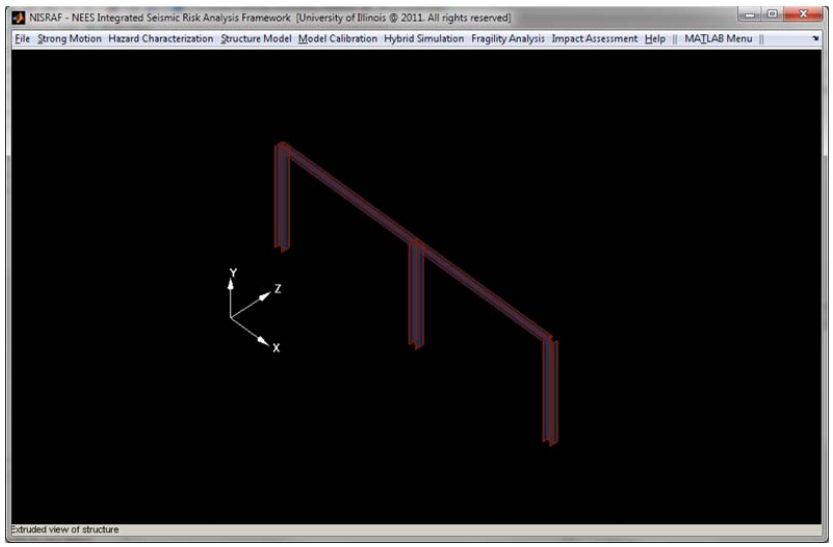


Figure 3-8 View submenus

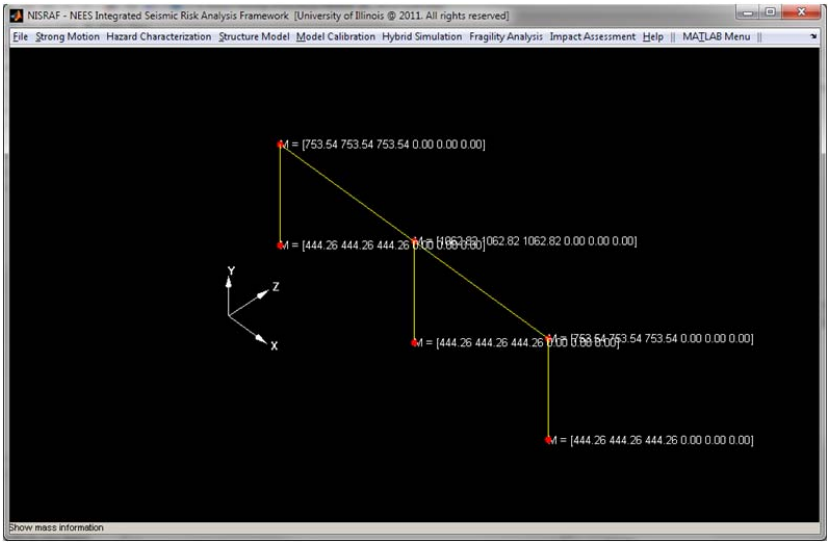
- *View-Skeleton View:* Skeleton view of structure



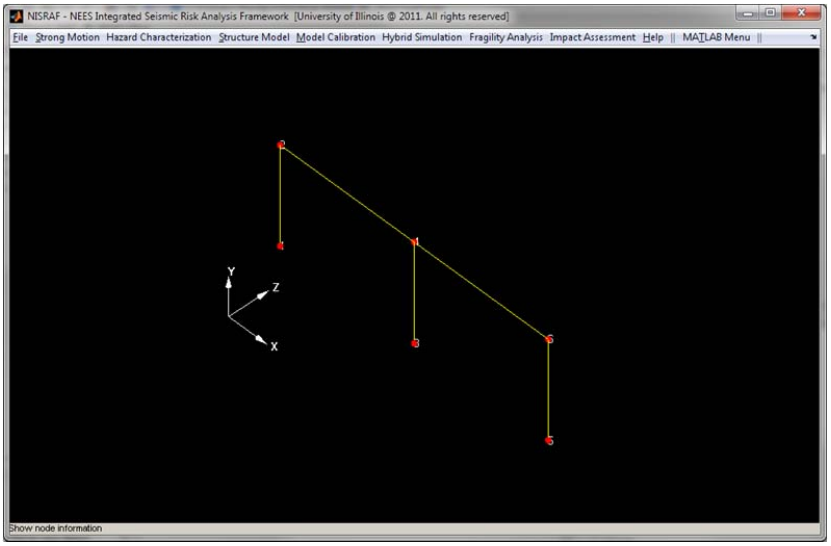
- *View-Extruded View:* Extruded view of structure



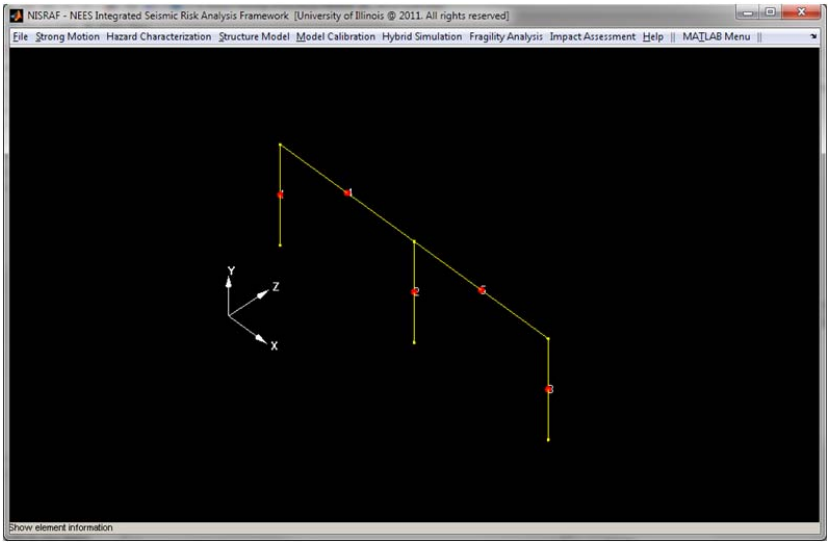
- *View-Mass:* View or hide nodal mass



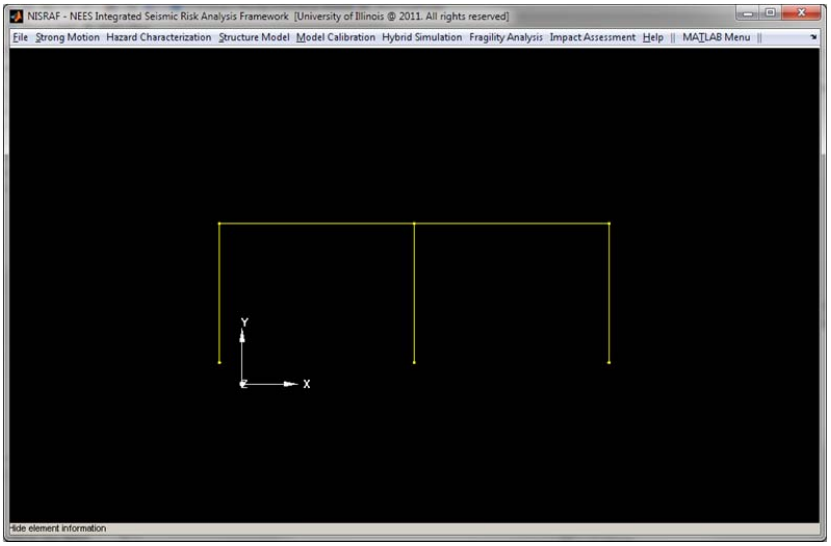
- *View-Node:* View or hide node number and coordinate



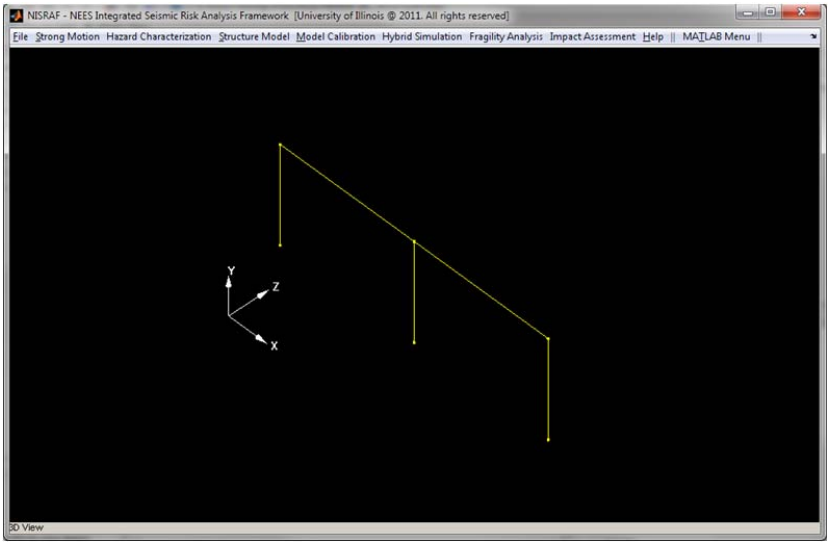
- *View-Element:* View or hide element number



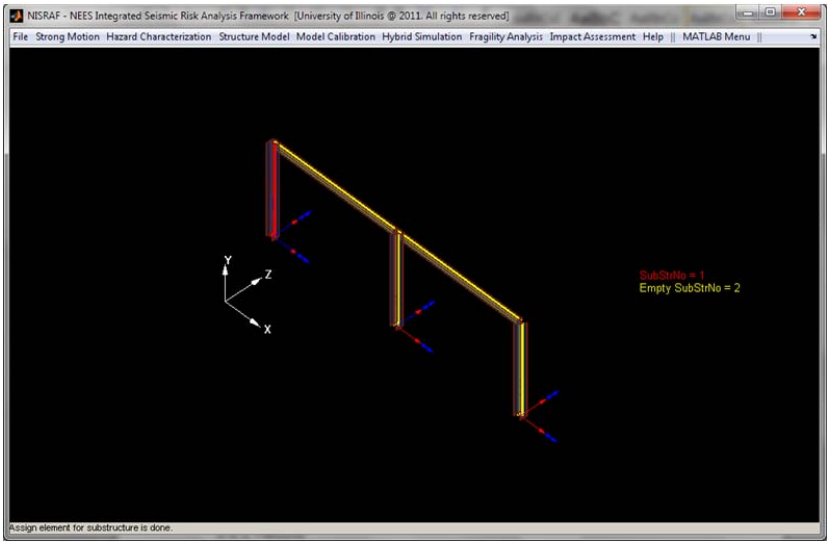
- *View-2D View-XY or YZ or ZX:* 2D view of structure



- *View-3D View:* 3D view of structure



- *View-Hybrid Model:* view or hide hybrid model (this submenu will be enabled when setting up the hybrid model is finished)



- *View-Status Bar:* View or hide status bar

- *View-Clear View:* Clear view except structure

3.4.5 Structure Model

The default structure properties can be seen and updated by using **Structure Model** menu. For example, user can update material, section, node, element, connectivity, boundary condition, mass, and damping properties. User can also refine mesh for simulation module by using **Refine Mesh** submenu.

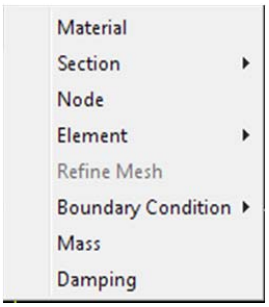
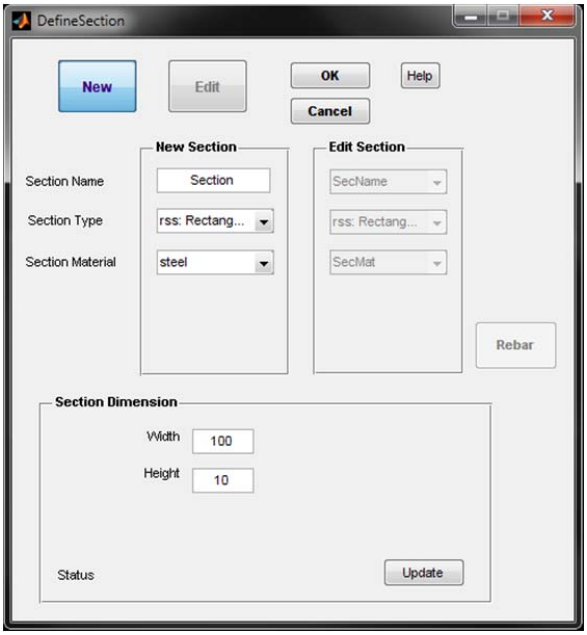


Figure 3-9 Structure Model submenus

- *Structure Model-Material:* Edit or add new material

User can update existing material properties or add new materials. '*stl1*' and '*con2*' materials defined in Zeus are available in this version of NISRAF. More materials will be available in the later version.

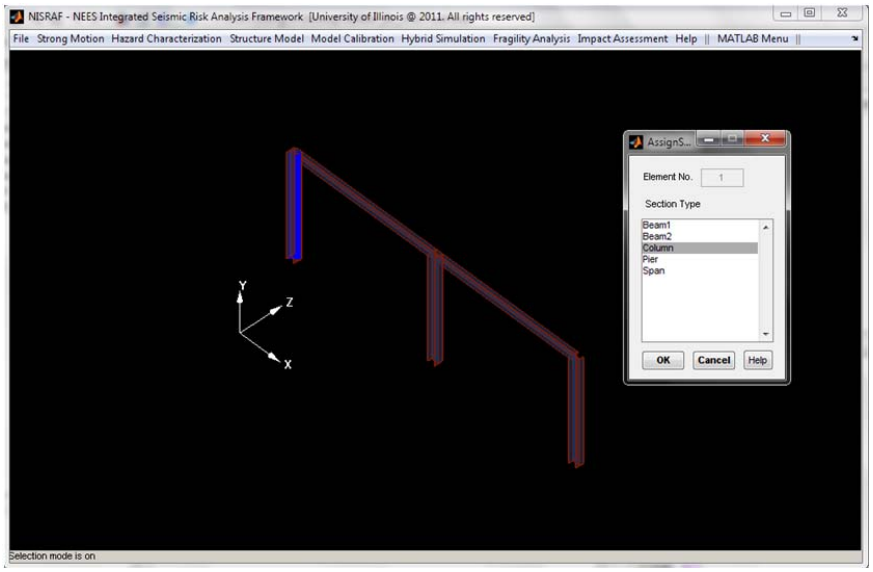
- *Structure Model-Section-Edit:* Edit or add new section



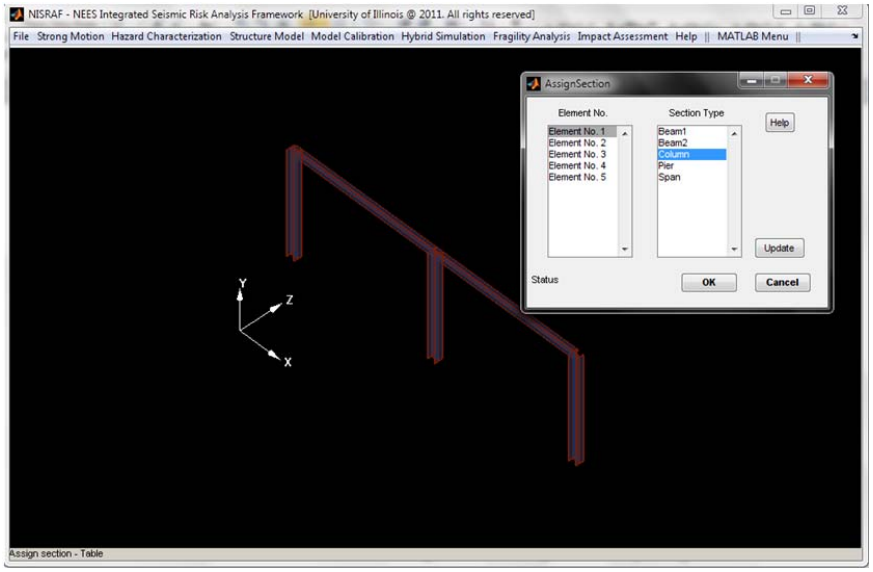
User can update existing material properties or add new materials. '*css*', '*rss*', '*sits*', and '*rcrs*' sections defined in Zeus are available in this version of NISRAF. More sections will be available in the later version.

- *Structure Model-Section-Assign-Update:* Update section properties of structure

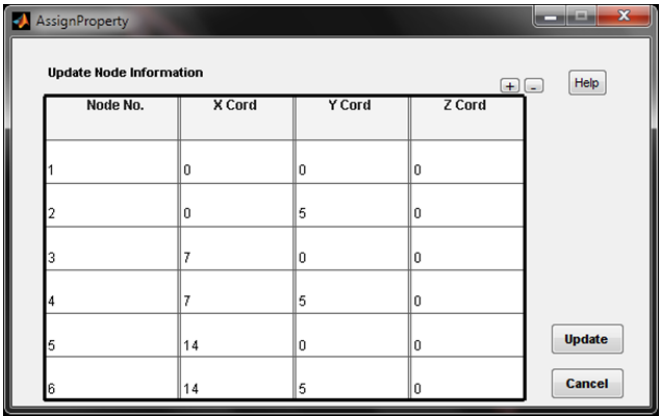
- *Structure Model-Section-Assign-One by One:* Assign section by selecting element



- *Structure Model-Section-Assign-Table:* Assign section by using table

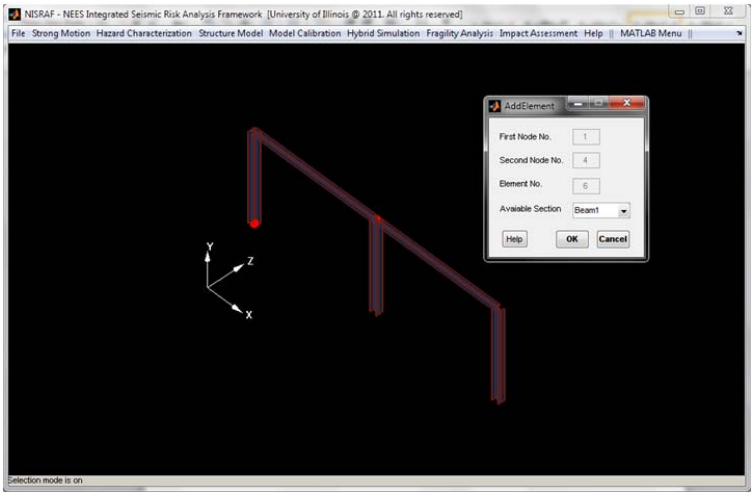


- *Structure Model-Node:* Update nodal coordinates

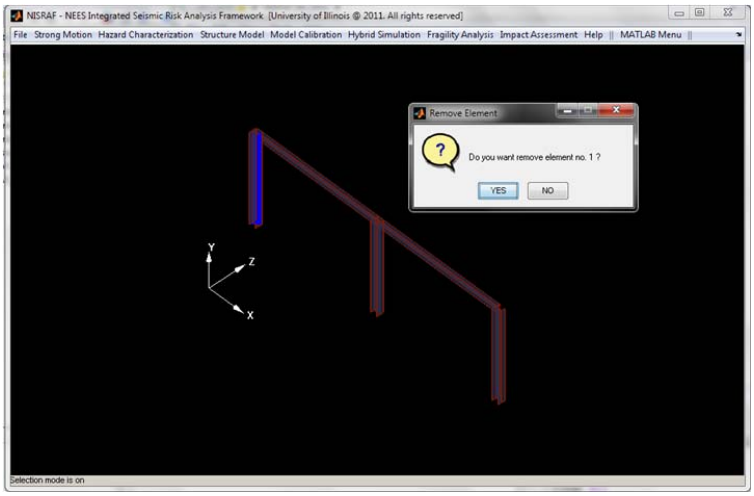


Node No.	X Cord	Y Cord	Z Cord
1	0	0	0
2	0	5	0
3	7	0	0
4	7	5	0
5	14	0	0
6	14	5	0

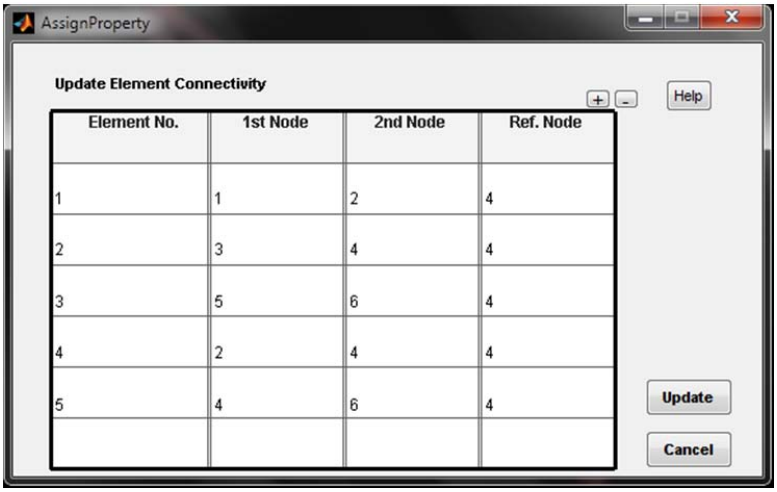
- *Structure Model-Element-Add:* Add element



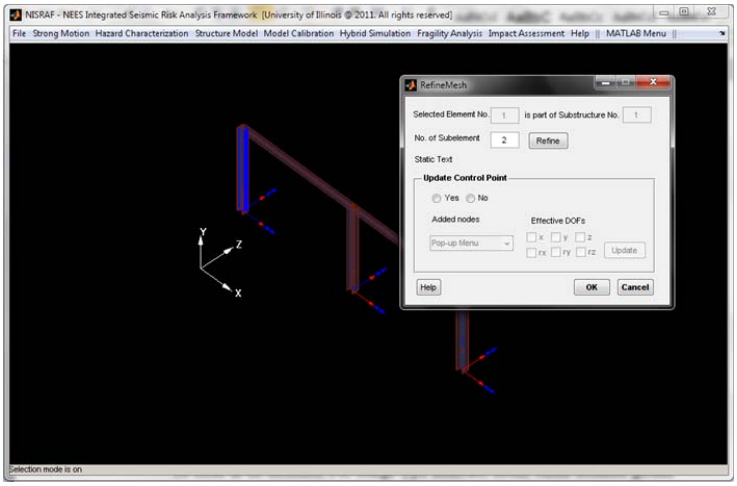
- *Structure Model-Element-Remove:* Remove element



- *Structure Model-Element-Connectivity:* Update element connectivity

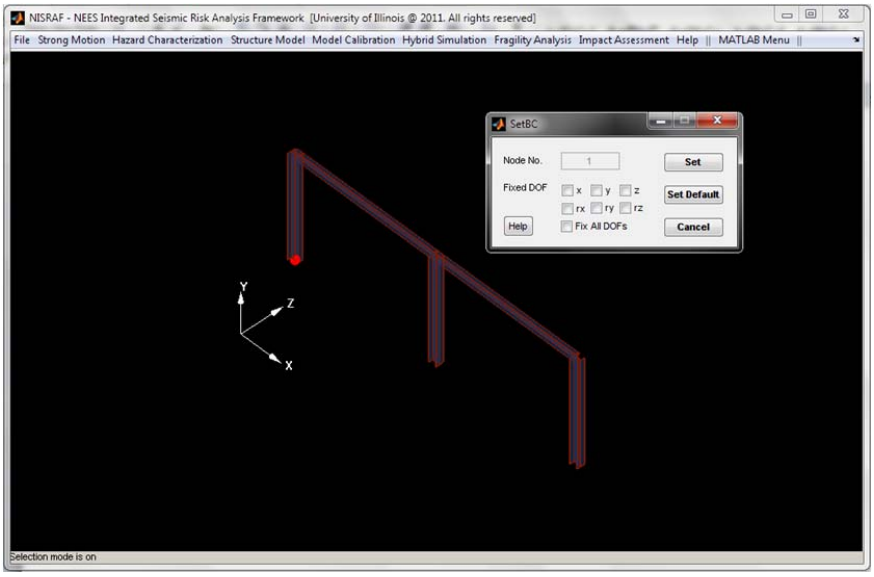


- *Structure Model-Refine Mesh:* Refine mesh for simulation module (this submenu will be enabled after defining substructure)

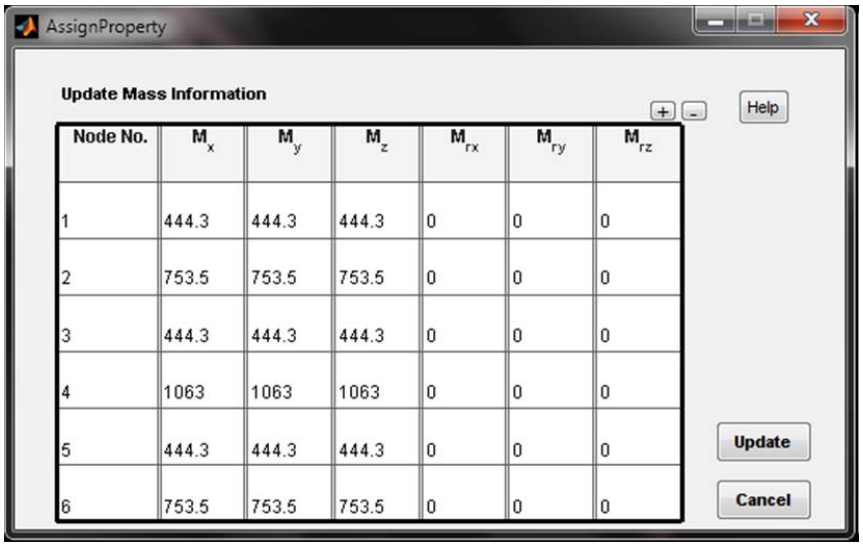


- *Structure Model-Boundary Condition-Default:* Set the boundary condition as default. For building type structure, nodes which attached ground (i.e., y=0) will be fixed in all direction. For bridge type structure, nodes which attached ground and two abutment nodes will be fixed in all directions except x- and rz-DOF of right abutment node.

- *Structure Model-Boundary Condition-One by One:* Set the boundary condition of selected node. When node is selected, following GUI will be shown.

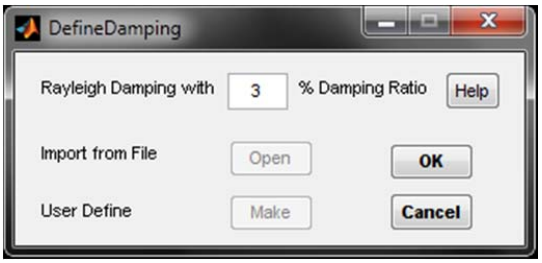


- *Structure Model-Mass:* Update nodal mass



Node No.	M_x	M_y	M_z	M_{rx}	M_{ry}	M_{rz}
1	444.3	444.3	444.3	0	0	0
2	753.5	753.5	753.5	0	0	0
3	444.3	444.3	444.3	0	0	0
4	1063	1063	1063	0	0	0
5	444.3	444.3	444.3	0	0	0
6	753.5	753.5	753.5	0	0	0

- *Structure Model-Damping:* Define damping. Only Rayleigh damping is supported in this version. More options for damping will be available for later version.



3.5 Model Calibration Menu

An automatic approach for system identification and model updating is developed and incorporated into NISRAF. Based on the instrumented data, the finite element model defined in previous section can be calibrated in NISRAF analysis platform.

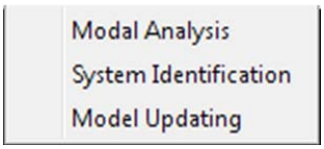
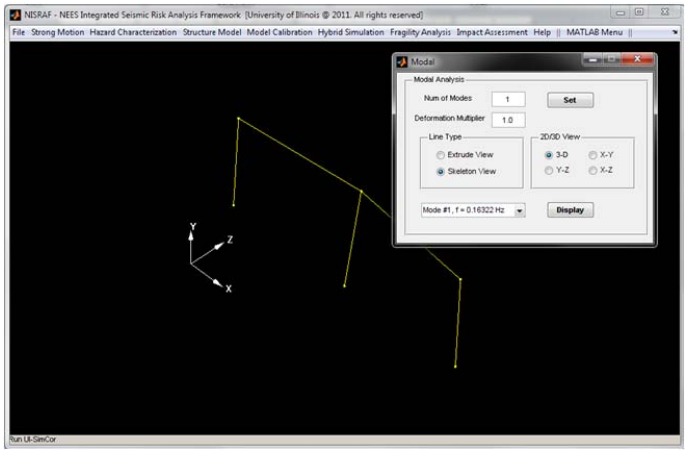


Figure 3-10 Model Calibration submenus

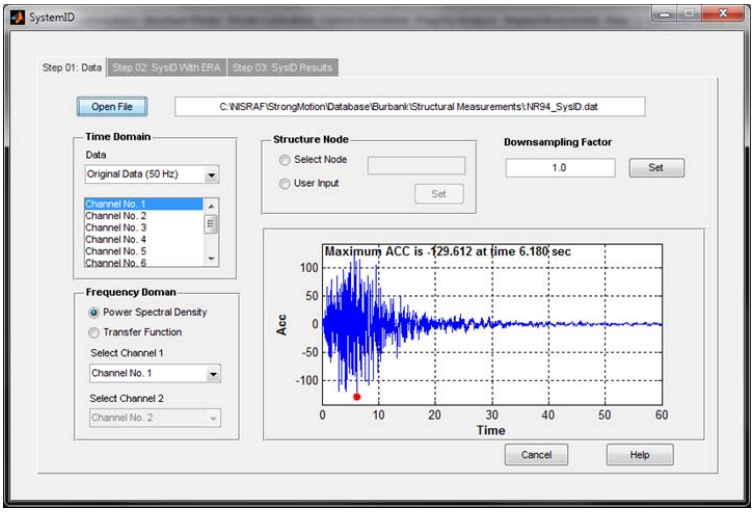
3.5.1 Modal Analysis

Before conducting system identification and model updating, the Modal Analysis allows user to check the modal information of structure (i.e., the mode shape and frequency). User is allowed to define the number of interested modes, deformation multiplier, line type, and 2D/3D view.

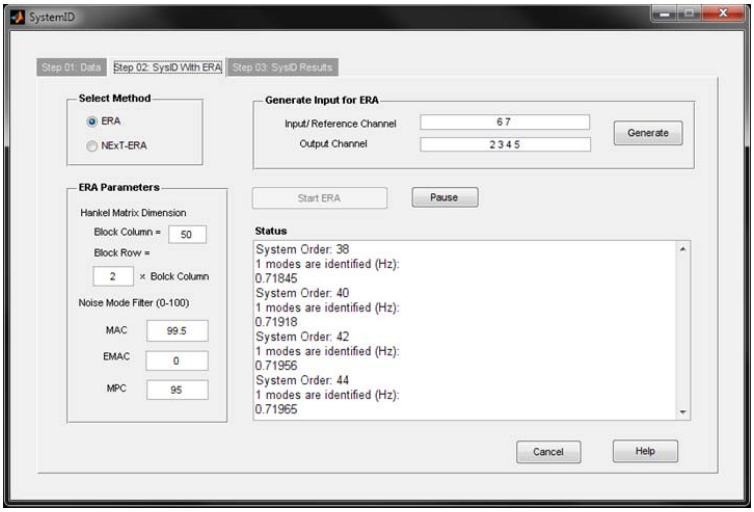


3.5.2 System Identification

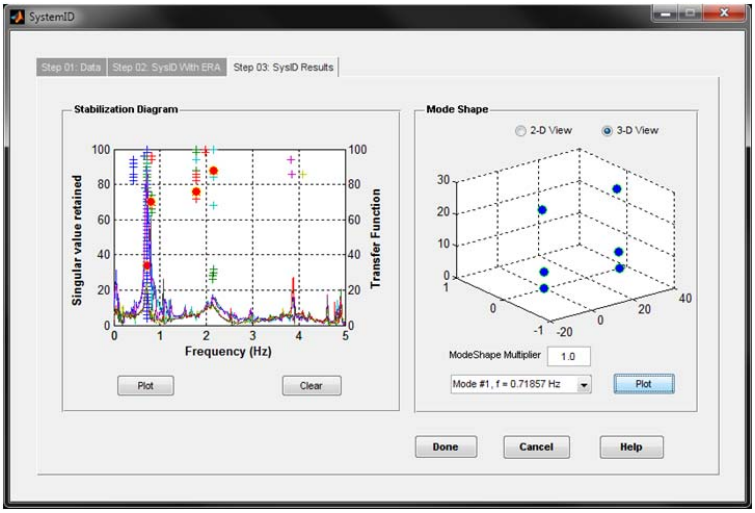
- The first step in **System Identification** is to import the instrumented sensor data.
- Next, downsampling factor is defined to downsample raw data.
- After that user need to locate the input and output channels to the related structural nodes.



- The second step in **System Identification** is to perform system identification via ERA method.

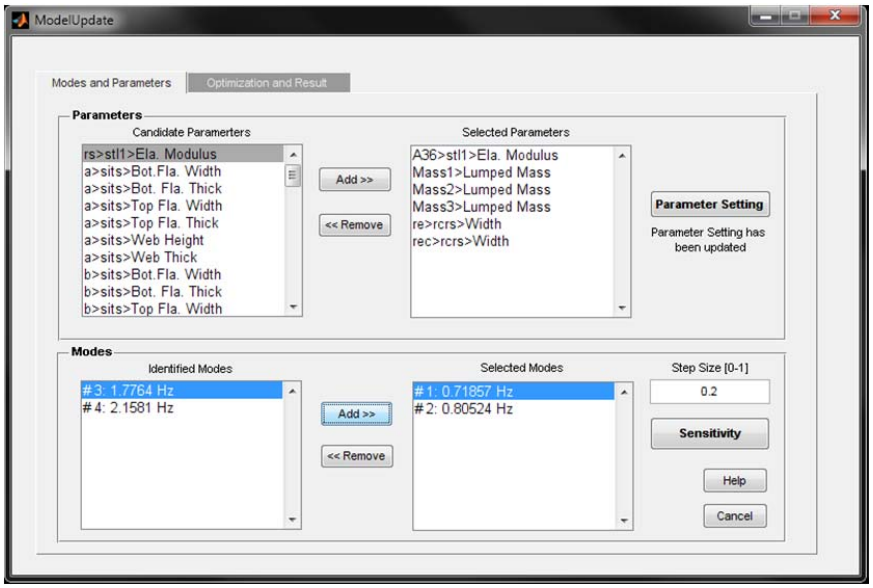


- After **System Identification** is completed, user can check the results.

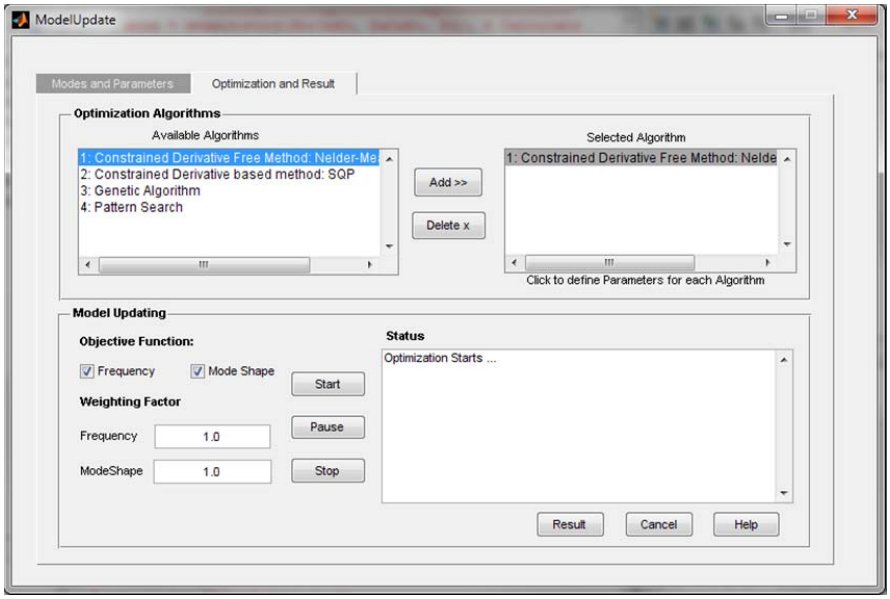


3.5.3 Model Updating

- The first step in **Model Updating** is to define the candidate parameters and identified modes.



- Next, an optimization algorithm is defined by user to conduct model updating.
- After defining objective function, NISRAF will perform model updating.
- User can check the progress and results during and after analysis.



3.6 Hybrid Simulation Menu

Under **Hybrid Simulation** Menu, NISRAF will assist user to create hybrid model including definition of substructures, platform of simulation parts and auxiliary module (i.e., cameral and data acquisition system). User can select element and/or joint to assign element to substructure. Furthermore, unassigned elements of structures will be assigned to the empty substructure by using **Auto Assignment** submenu.

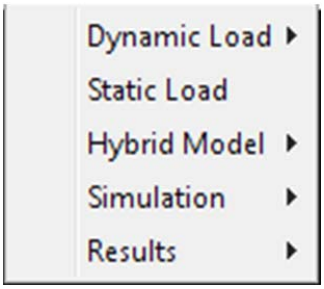


Figure 3-11 Hybrid Simulation submenus

3.6.1 Dynamic Load

Loading scenarios for hybrid simulation can be defined in this menu. Existing loading file can be used or user can create load.

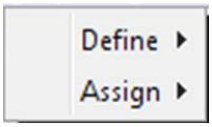
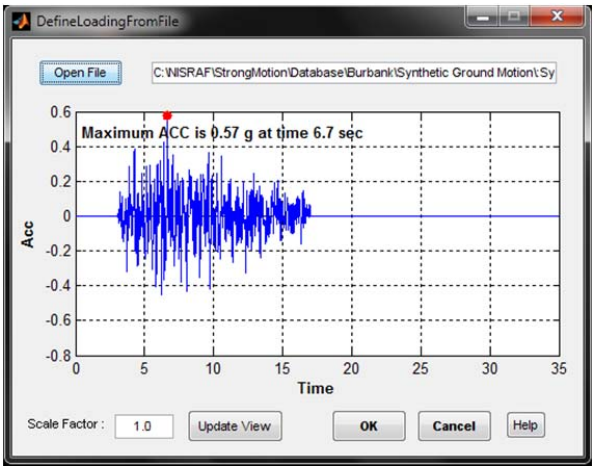
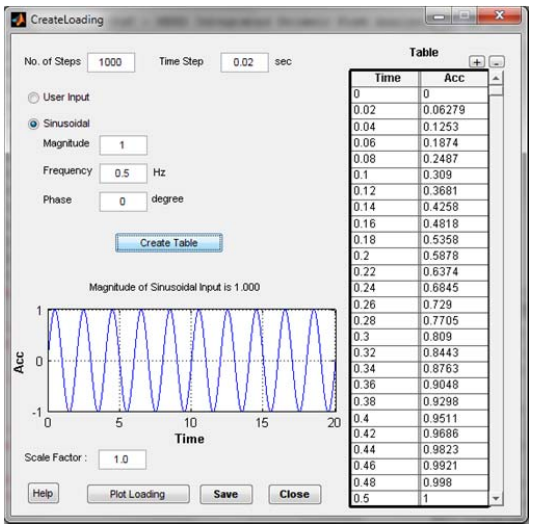


Figure 3-12 Dynamic Load submenus

- *Hybrid Simulation-Dynamic Load-Define-From FIle:* Open existing load file



- *Hybrid Simulation-Dynamic Load-Define-Create:* Create loading history using table



- *Hybrid Simulation-Dynamic Load-Assign-x-dir (or y-dir, z-dir, rx-dir, ry-dir, rz-dir):* Assign load in any direction.

3.6.2 Static Load

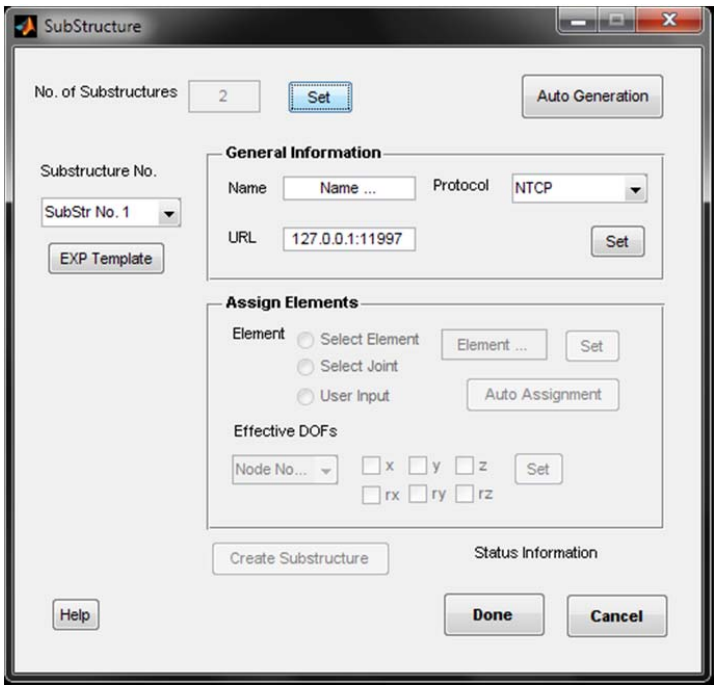
Assign static (gravity) load when conducting hybrid simulation. Only static load applied by importing Zeus file is allowed in current version NISRAF.

3.6.3 Hybrid Model

After defining the load, under **Hybrid Model** menu, user will be prompted to define substructures, analysis platform and auxiliary for hybrid simulation.

Figure 3-13 Hybrid Model submenus

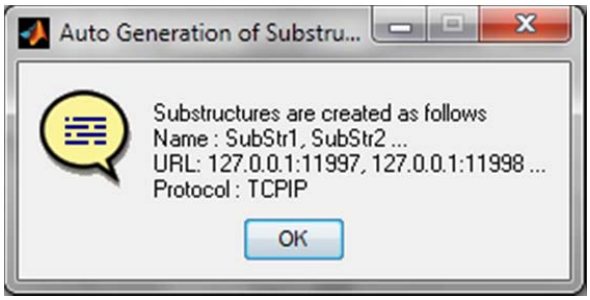
- *Hybrid Simulation-Hybrid Model-Define Substructure:* Define substructure



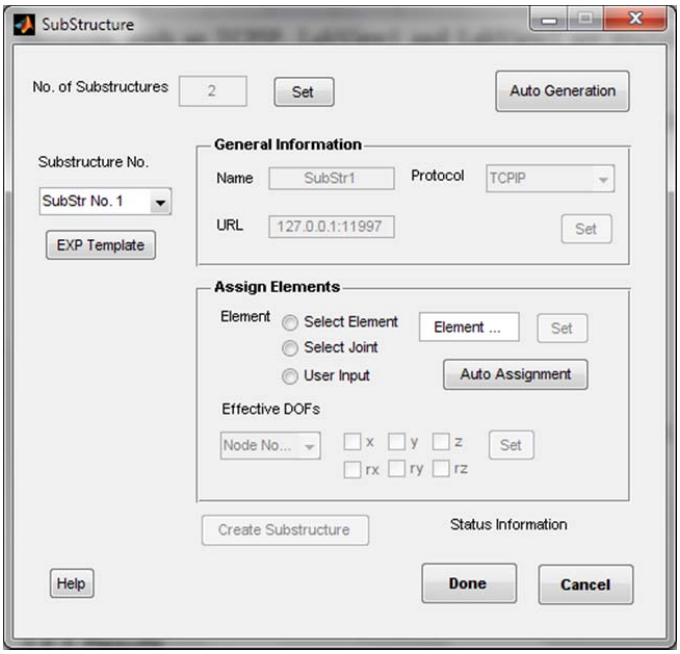
First, the number of substructure should be defined. Then panel for general information will be enabled.

General information of substructure such as name, communication protocol, IP and port number can be defined in this panel. Three communication protocols, such as TCPIP, LabView1 and LabView2 are available in this version of NISRAF.

If Auto Generation button is clicked, the name, protocol, and URL of each substructure are defined automatically as follow.

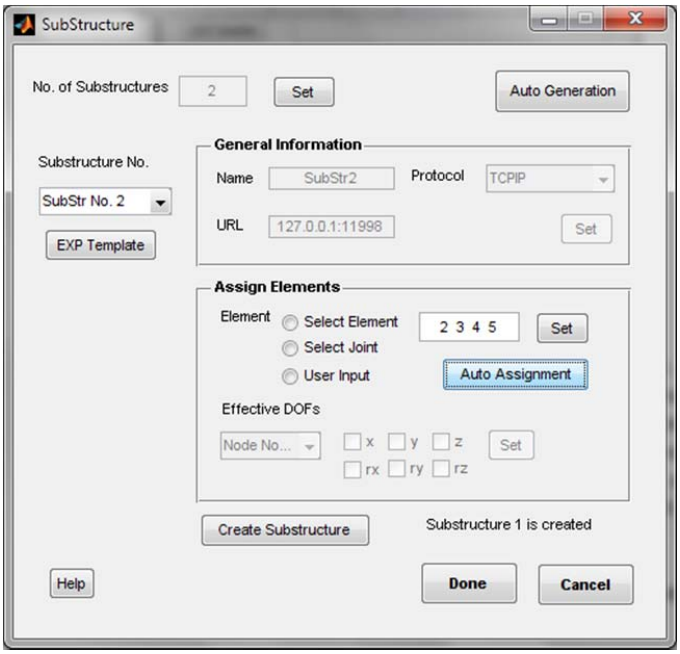


Once general information is defined, this panel will be disabled and panel for advanced information of substructure will be enabled.



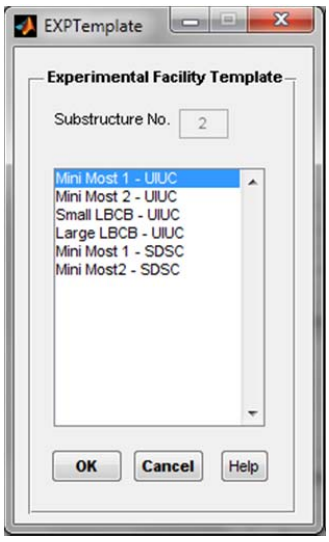
There are four methods to assign elements to the substructure. User can select any element and/or joint to assign element to the substructure. If user knows specific element number, then use can type in the element box. Finally, all of unassigned elements can be assigned to the empty substructure by clicking **Auto Assignment** button. This is same function of **Hybrid Model-Assign Substructure-Select Element** or **Select Joint** or **Auto Assignment**. These will be explained in the later.

If **Select Element** option is selected, following, user can select any element of structure.

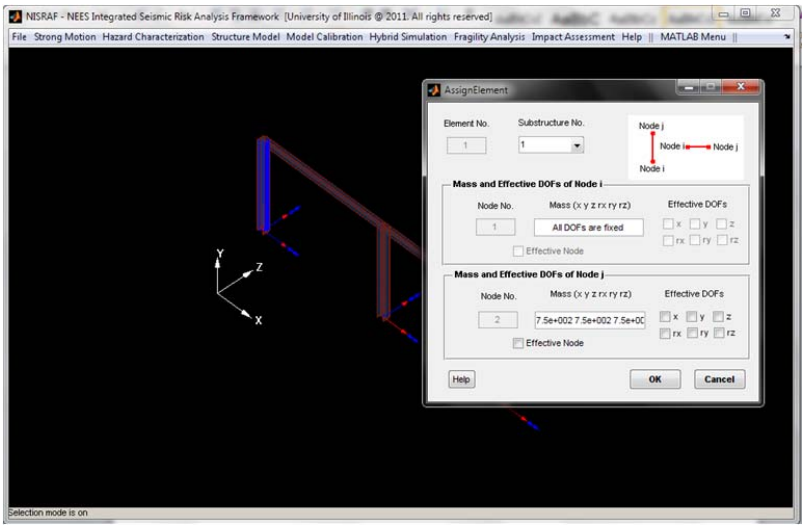


The selected element is updated into element box. User can add more elements. After defining elements, click **Create Substructure** button to save the defined substructure. User can continue to define another substructure if applicable.

Meanwhile, user can use experimental template to load predefined setup of experimental facilities. If **Exp Template** button is clicked, following GUI will be shown. It's recommended to check all the experimental setup before running test in the laboratory.

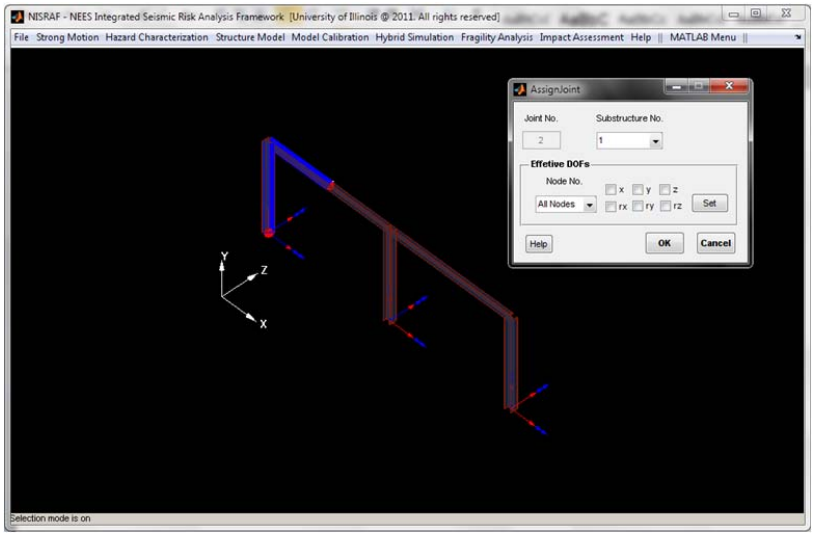


- *Hybrid Simulation-Hybrid Model-Assign Substructure-Select Element:* Assign selected element to substructure. When the element is selected, following GUI will be shown.



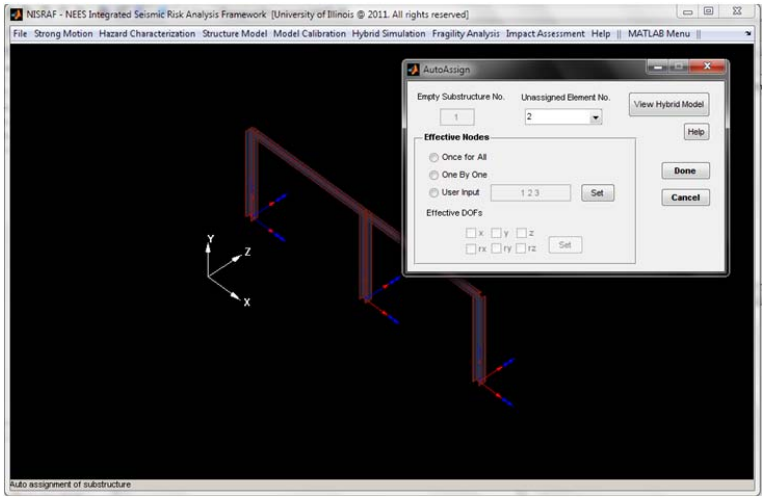
The selected element is highlighted. Also, the selected element number and available substructure number are shown. User can update mass and effective DOFs of each node. The first node of this element is one of boundary condition which all DOFs are fixed, so effective DOFs check boxes of this node (i.e., Node 1) are disabled.

- *Hybrid Simulation-Hybrid Model-Assign Substructure-Select Joint:* Assign element to substructure by selecting joint. When joint is selected, following GUI will be shown.

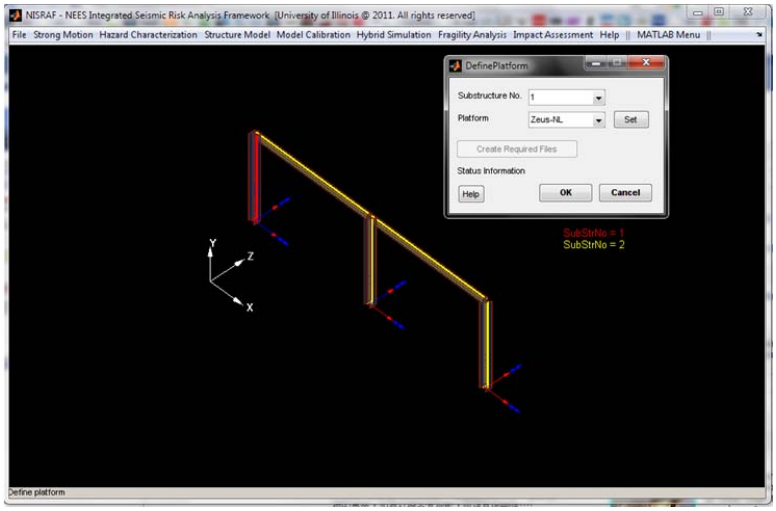


When any joint is selected, adjacent elements will be divided into two to move mass on joint node to end node. The selected joint number and available substructure number are displayed. User needs to update effective DOFs.

- *Hybrid Simulation-Hybrid Model-Assign Substructure-Auto Assignment:* Auto assignment of the unassigned element to empty substructure. This is very useful when the structure is complicated.



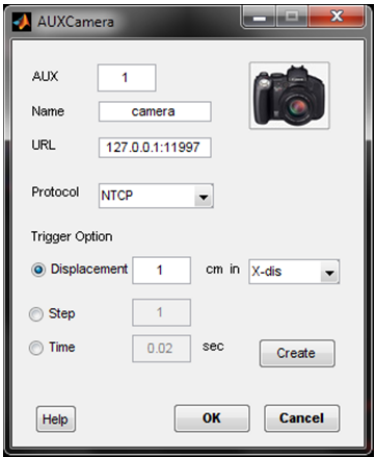
- *Hybrid Simulation-Hybrid Model-Define Platform:* Define platform of each substructure.



User can define platform of substructure as Zeus-NL, OpenSees, FedeasLab, Abaqus, and Experiment. Currently, Zeus-NL, OpenSees, FedeasLab, and Experiment platform are available in this version. If the platform of substructure is defined as Zeus-NL, OpenSees, and FedeasLab (i.e., simulation module), the required files for hybrid simulation (conducted by UI-SimCor) such as input file of static analysis module and configuration

file will be generated automatically within the folder which name is same as name of substructure. If 'Experiment' is selected as platform, only folder which name is same as name of substructure will be created.

- *Hybrid Simulation-Hybrid Model-Auxiliary-Camera (or DAQ):* Define camera or DAQ module. Only camera module is supported in this version. DAQ module will be supported in the later version.



3.6.4 Simulation

After defining the load and substructures, under **Simulation** menu, user will be prompted to run hybrid simulation.

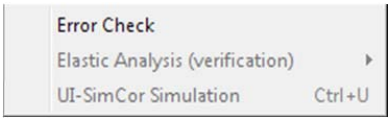
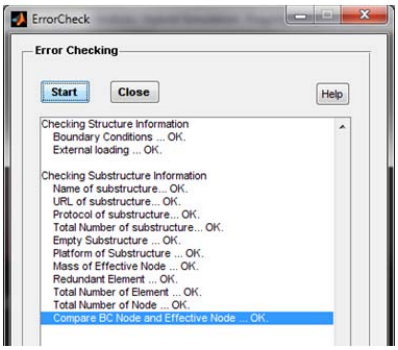


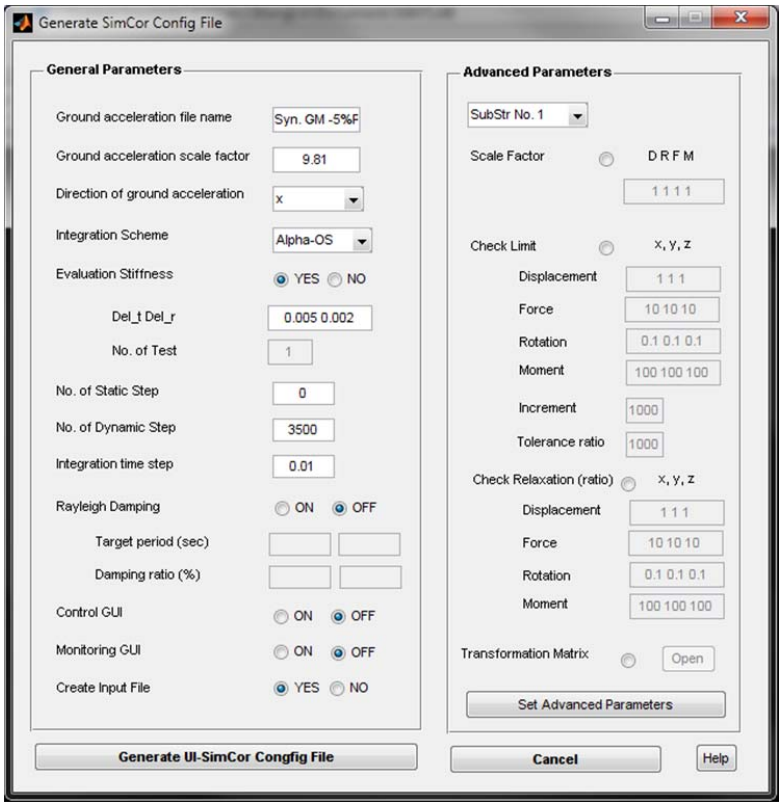
Figure 3-13 Simulation submenus

- *Hybrid Simulation-Simulation-Error Check:* Check error of hybrid simulation environment.



- *Hybrid Simulation-Simulation-Elastic Analysis (verification):* Allow user to run static or dynamic analysis to verify the defined hybrid model.

- *Hybrid Simulation-Simulation-UI-SimCor Simulation:* After define and/or update the required information, NISRAF will be ready to conduct hybrid simulation.



3.6.5 Results

After finishing hybrid simulation, NISRAF allows user to check simulation results through displacement/force history plot, animation, and photos if applicable.

3.7 Fragility Analysis Menu

Fragility Analysis menu is composed of three main parts: namely, **Define Limit States**, **Run Hybrid Simulation** and **Hybrid Fragility Curves**, as shown in Figure 3-14. Methodologies and analysis procedures of fragility analysis can be found in Lin (2010). One of the features of the proposed advanced fragility analysis approach is its ease of use. With structural information available from **Structural Model**, the user defines interested Interstory drift angle (ISDA) through the interactive structural model. Meanwhile, when performing hybrid simulation in order to derive mean seismic intensity, NISRAF calculates ISDAs, compares with target ISDA, calculates scale factor, and asks to continue the next simulation. The above designs avoid the heavy and tedious calculations. The "***hold on***" feature allows the user to have time to replace the experimental specimen in the laboratory, which is really a useful and practical design.

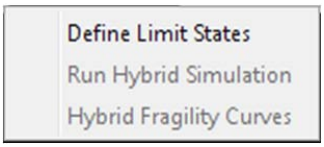
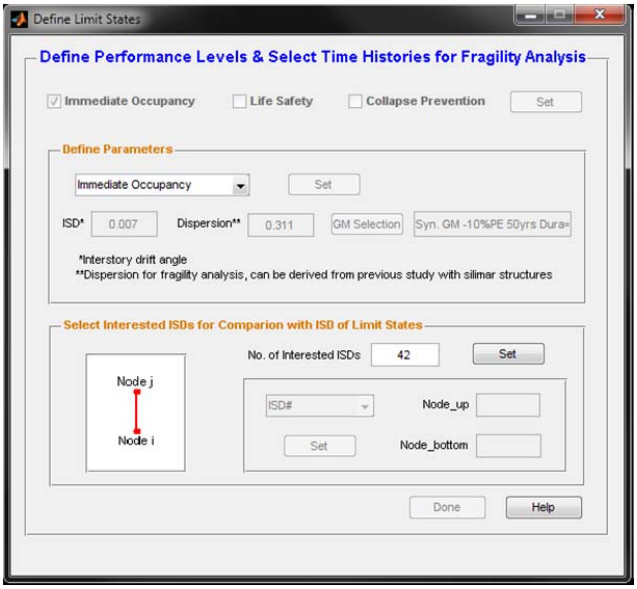


Figure 3-14 Fragility Analysis submenus

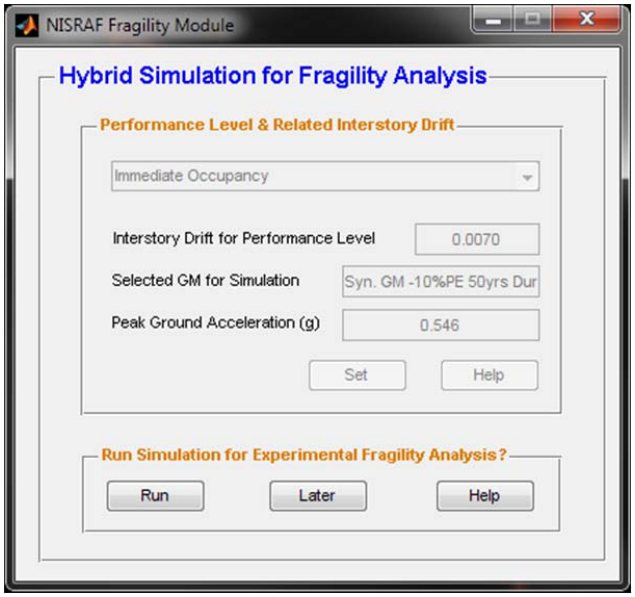
3.7.1 Define Limit States

- To derive hybrid fragility curves, user need to define parameters and select time history for interested performance levels.
- For building type, the interstory drift (ISD) is used to make comparison between the target performance level and the hybrid simulation results. Therefore, user is prompted to define information of the interested ISDs (i.e., the up and bottom node).



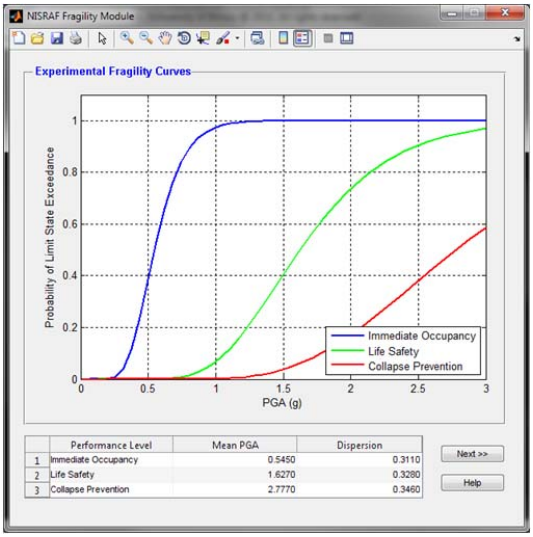
3.7.2 Run Hybrid Simulation

- After finishing the definition of limit states and the setup of hybrid simulation (i.e., substructure, platform, and others), NISRAF will run hybrid simulation in order to derive fragility curves.



3.7.3 Hybrid Fragility Curves

- After several hybrid simulation tests in order to meet the target performance levels, NISRAF will based the mean PGA value and the dispersion defined by user to generate the interested fragility curves. The derived fragility curves are compatible and ready to be used in MAEviz.



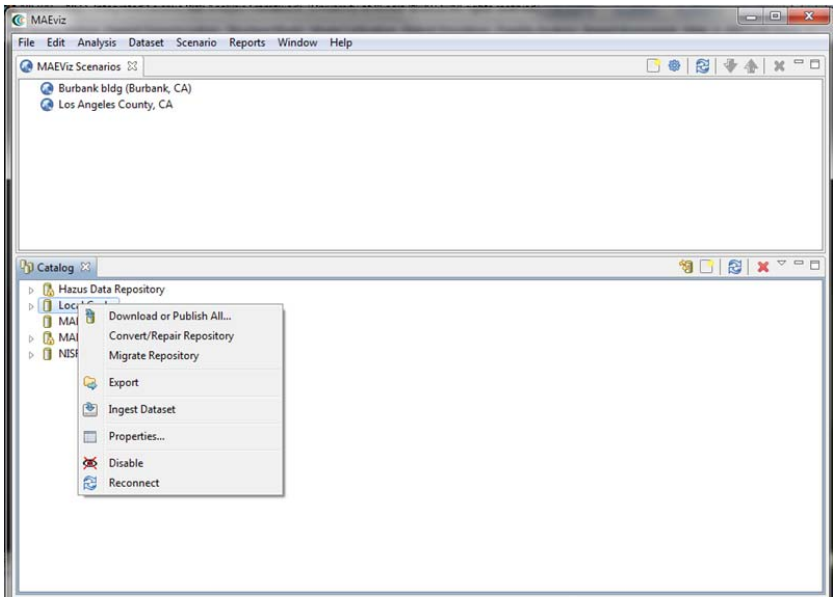
3.8 Impact Assessment Menu

Finally, fragility curves from **Fragility Analysis** and the hazard map from **Hazard Characterization** are fed into earthquake impact assessment packages (MAEviz, for example) to evaluate the seismic loss (Figure 3-15).

Figure 3-15 Impact Assessment submenus

3.8.1 MAEviz

- The first step to perform MAEviz under NISRAF is to ingest the generated hazard map and fragility curves into MAEviz. Both hazard map and fragility curves are compatible with MAEviz, no additional format transformation is required.



- Once the required information is fed, user can perform all the functions of MAEviz under NISRAF in order to evaluate the seismic losses.

3.8.2 HAZUS

HAZUS is not available in this version of NISRAF.

3.9 Help Menu

Manuals of NISRAF and UI-SIMCOR are available. In addition, **About NISRAF** states the copyright as well as version information.

4. Tutorial Examples

An instrumented building was selected to demonstrate NISRAF in this example. In the following sections, background information about this building and site conditions are presented first. Thereafter, step by step analysis in NISRAF is performed.

4.1 Introduction

4.1.1 Building Information

A six-story commercial building in Burbank, California (latitude = 34.185°, longitude = -118.308°), was selected for this study (Figure 4-1). This is a steel moment resisting frame building, in which the perimeter frames are the primary lateral load resisting system, and the internal frames are only resisting gravity load, as shown in Figure 4-2. Reference is made to Anderson and Bertero (1991) for detailed information about this building. This building is instrumented by the California Strong Motion Instrumentation Program (CGS - CSMIP Station No. 24370) in 1980 with 13 sensor channels as shown in Figure 4-3. Several significant earthquakes were captured, such as Whittier (1987), Sierra Madre (1991) and Northridge (1994). Data are available in the Center for Engineering Strong Motion Data (CESMD, www.strongmotioncenter.org).

Figure 4-1 Photo of 6-story steel moment frame building in Burbank, California

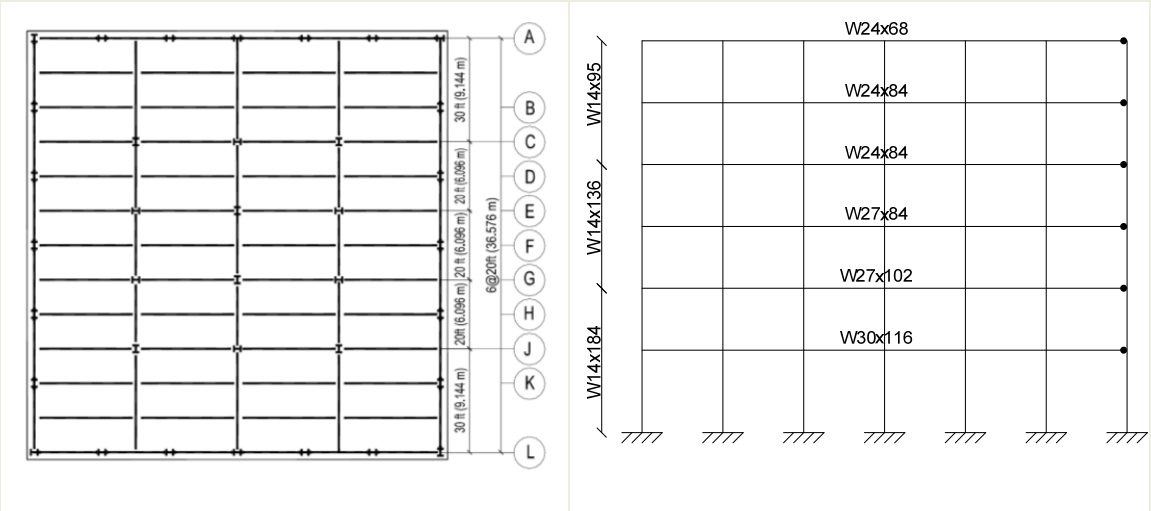


Figure 4-2 Elevation and plan view of Burbank building

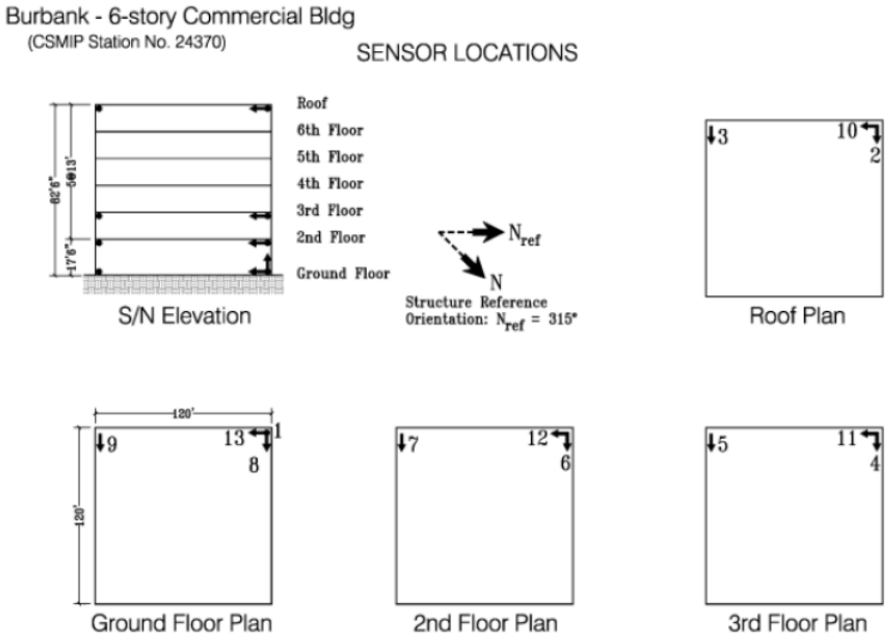


Figure 4-3 Sensor location of Burbank building (CESMD)

4.1.1 Site Condition

Based on the SMIP geotechnical report No. 131 (Fumal et al., 1979), the soil deposits at the Burbank site is Pleistocene alluvium. The borehole log (Figure 4-4) shows the soil profile for the top 30 meters at this site. The water table is assumed 20 feet below the ground surface, based on the geologic criteria for Burbank with soil deposits of similar Pleistocene age (Department of Conservation, Division of Mines and Geology, 1998).

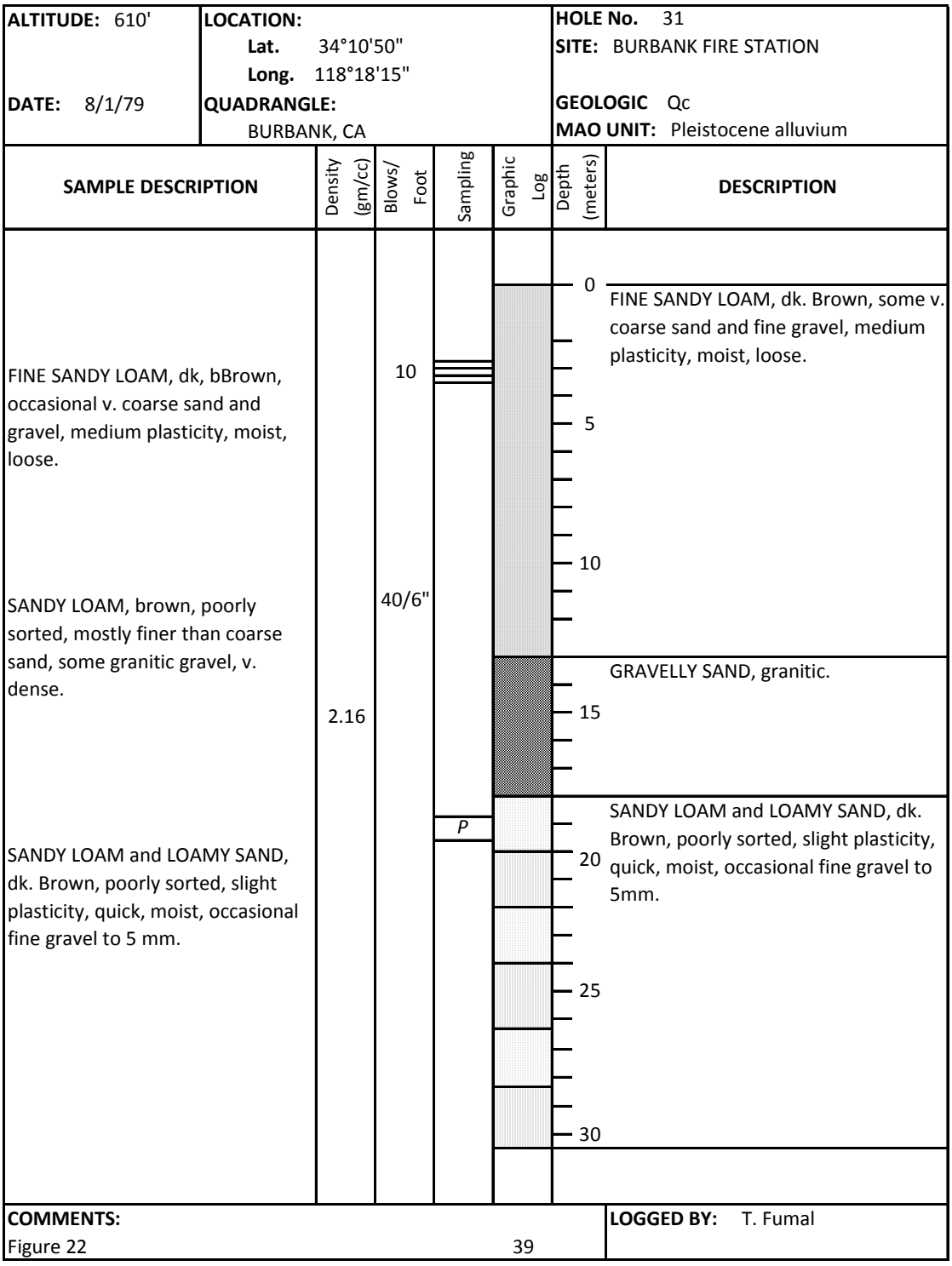


Figure 4-4 Borehole log of the Burbank site (adapted from Fumal et al., 1979)

4.2 Strong Motion

Either **Strong Motion** or **Structure Model** must be the first step in NISRAF. **Strong Motion** was selected as the first step in this example. Through the linkage to web-database, free-field station records around the Burbank building site and structural sensor histories during the past earthquakes were downloaded and deposited in NISRAF. After that, an interactive window with already-downloaded information allows user to add some information (background, description and image), as shown in Figure 4-5.

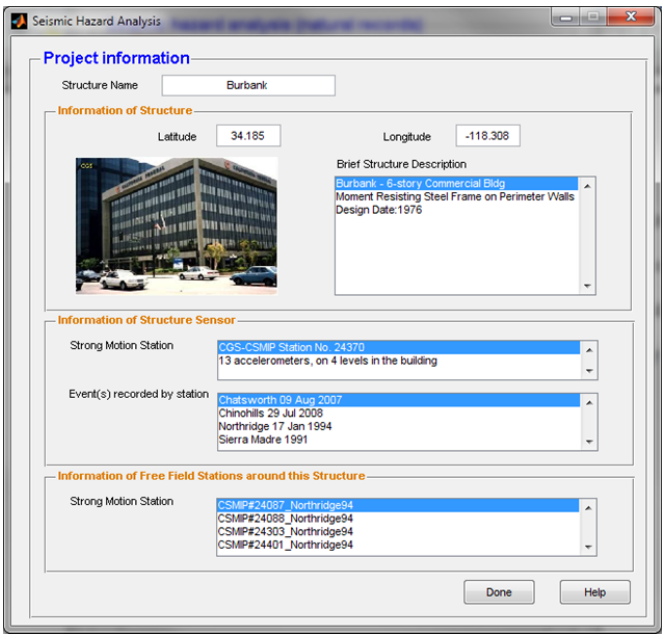


Figure 4-5 GUI to manage project and downloaded records

4.3 Hazard Characterization

With instrumented strong-motion records from **Strong Motion**, the **Hazard Characterization** analysis was undertaken. Synthetic ground motions with various hazard levels were generated for further use in **Hybrid Simulation** and **Fragility Analysis**. The hazard map for the Northridge earthquake in the Burbank area was generated for further use in **Impact Assessment**.

4.3.1 Synthetic Ground Motion

Ground motions with various hazard levels are generated based on the seismic information specified by the users. The deaggregation results for different hazard levels (10%, 5%, and 2% probability of exceedance in 50 years), as shown in Table 4-1 and Table 4-2, at the Burbank site were fed into this advanced hazard method. Next, sets of synthetic ground motions, including site response analysis and varying with duration and hazard levels were generated automatically. These motions with compatible format were

further used in the hybrid simulation and fragility analysis. Figure 4-6 presents one of the generated synthetic ground motions and its response spectrum.

Table 4-1 Deaggregation results at Burbank site

	Return Period (yrs)	M	R (km)	Epsilon
2%/ 50yrs	2475	6.73	6.9	1.18
5%/ 50yrs	975	6.71	8.5	0.91
10%/ 50yrs	475	6.71	10.6	0.63

Table 4-2 Contributed fault information based on deaggregation results

Name	Type	F_{RV}	F_{NM}	$f_{hng,\delta}$	f_{sed}
Verdugo Char	Reverse	1	0	1	0
Elysian Park Char	Blind trust (reverse)	1	0	1	0

**assume $\delta = 90°$, $Z_{2.5} = 2km$*

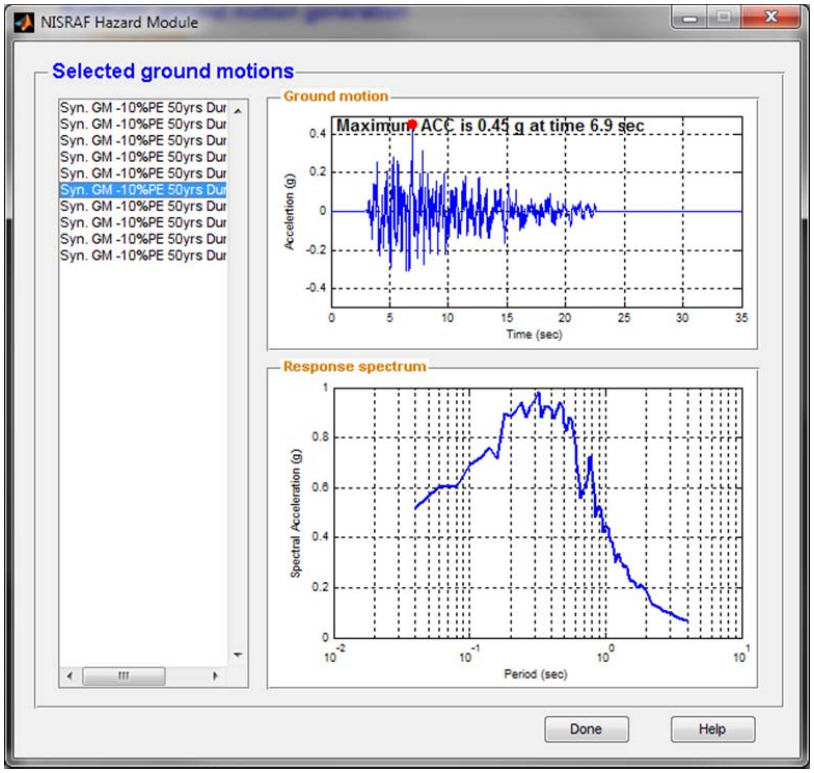


Figure 4-6 Synthetic ground motion and its response spectrum

4.3.2 Hazard Map

Hazard map, the exposure when calculating earthquake loss, is one of the indispensable components of regional impact assessment. The map of PGA for the 1994 Northridge earthquake in the Burbank area in standard gravity (g) was generated in this application. This map is not only served to demonstrate the proposed method, but also used for impact assessment on the selected building.

SMIP geotechnical report (Fumal et al., 1979) was used again to illustrate local site characteristics. Step-by-step procedures to generate the hazard map were then performed. The Northridge earthquake mechanism, the site conditions (soil profiles and material properties) and map information (such as interested region scope and cell size) were defined in the first step. Next, the CB-NGA (Campbell and Bozorgnia, 2008) and duration prediction equation along with SIMQKE (Gasparini and Vanmarcke, 1976) and DEEPSOIL (Hashash et al., 2009), were performed for each cell. Finally, PGA values were collected and hazard map of the Burbank area was presented, as shown in Figure 4-7.

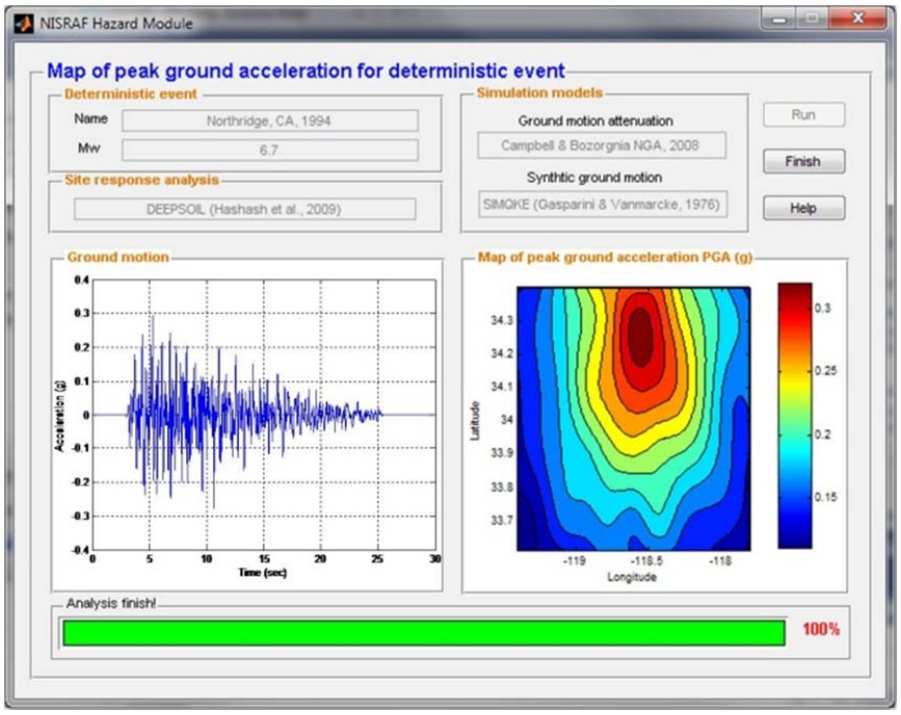


Figure 4-7 Hazard map at Burbank area

4.4 Structure Model

A finite element model was created in NISRAF, as shown in Figure 4-8. Due to the fact that only the perimeter frames are used for the lateral load resisting system, a 2-D model of the exterior frame was modeled to represent the whole structure. Section dimensions and material properties for each beam and column were based on design documents. Lumped mass was used and applied at every beam-column connection. Concrete slabs were modeled and connected to steel girders using rigid elements, to account for their contribution of stiffness.

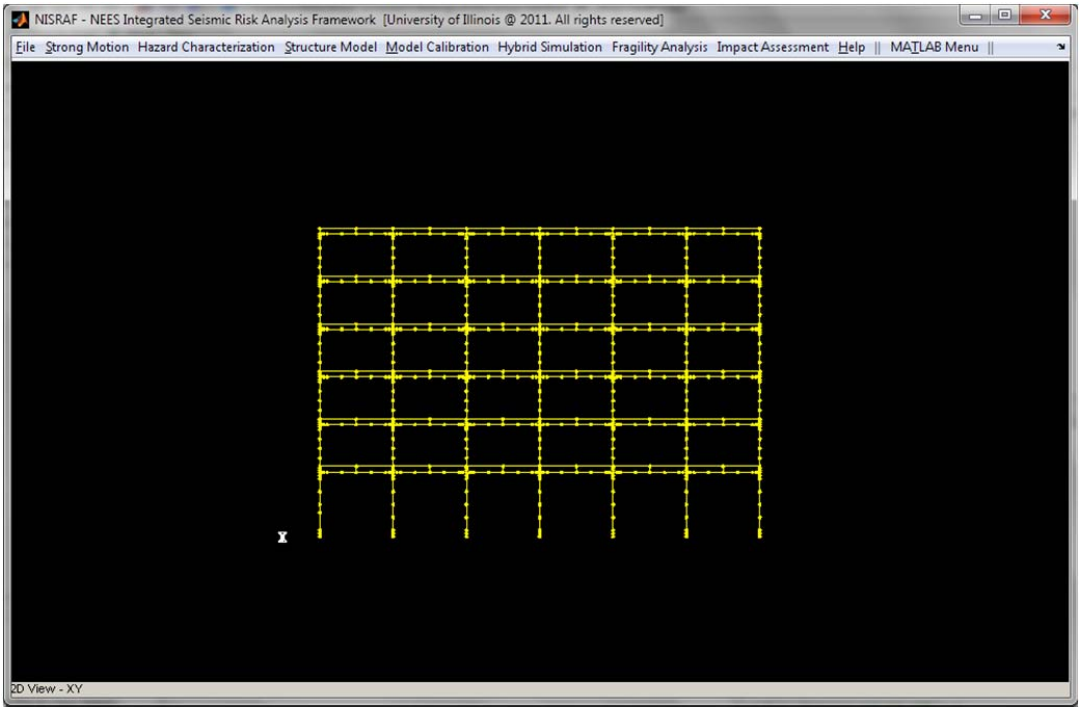


Figure 4-8 2-D FE model of Burbank building in NISRAF

4.5 Model Calibration

With FE model created in Structural Model, Model Calibration is performed to tune the FE model. Two procedures, namely, system identification and model updating, were executed in this step.

4.5.1 System Identification

Input channels and output channels were defined first. Based on the design drawings, exterior and interior columns are firstly supported on steel girders and reinforced concrete girders, respectively, and both of them are in turn supported on a pair of 32 feet long and 30 inches diameter reinforced concrete piles. Therefore, it is reasonable to consider that

all columns are fixed. Hence, the records from the ground floor were treated as the input motions, while other records were considered as the responses of the structure. Consequently, channel 8 and 9 were defined as input, while channels 2 to 7 were output channels, and, hence, the dimension of impulse function matrices was 2 by 6. Note that channels 4 and 5 were not working properly during the Northridge earthquake of 1994. Therefore, data from these two channels were not available and only four output channels were available. The dimension of impulse response function matrices was 2 by 4 for the Northridge earthquake.

The ERA method was then performed for the Northridge earthquake record. The stabilization diagrams and the identified mode shapes were shown in Figure 4-9. The first and second bending modes were then identified as 0.72 Hz and 2.14 Hz, respectively. The associated damping ratios were 3.37% and 6.71% (Table 4-3).

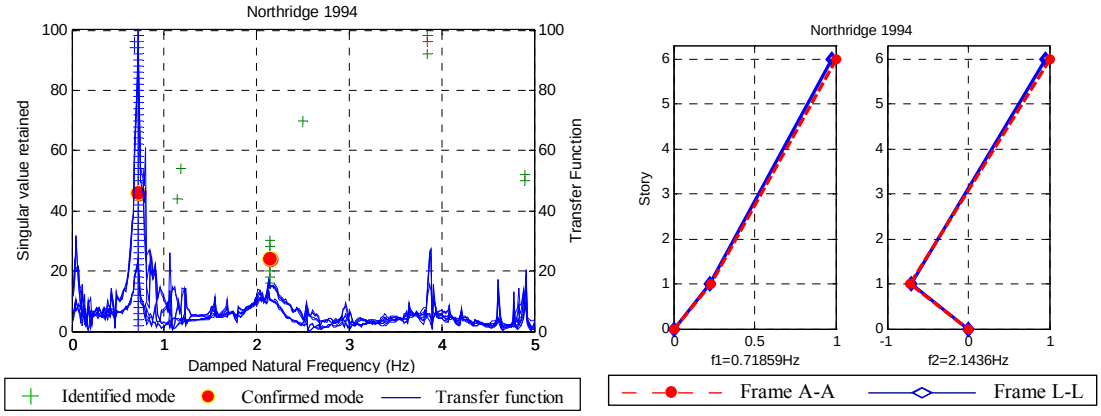


Figure 4-9 Stabilization diagrams and identified mode shapes

Table 4-3 Frequency and δ of identified with ERA method

Mode	f (Hz)	δ(%)
1	0.719	3.373
2	2.144	6.715

4.5.2 Model Updating

With the identified natural frequencies and mode shapes, dynamic FE model updating was then performed to improve the FE model of the Burbank building. Selection of candidate parameters to be updated was the first step in model updating. The selected parameters for the Burbank building were shown in Table 4-4. To keep the physical

meaning of each parameter, lower and upper bounds were applied based on the degree of uncertainties. For example, the effective widths were calculated based on AISC specification, which was likely to be very conservative. In addition, the deflection of the slab defined the contribution of the slab to the composite beam, thus affecting the effective width. Therefore, the effective width of slab had large uncertainty, thus a relatively larger range of variation (±50%) was applied.

Table 4-4 Selected parameters for model updating and updated results

Selected Parameters	Description	Initial Value	Bound (%)	Updated Value	Change (%)
Es (N/mm^2)	Young's modulus of steel	2.10E+05	±5	2.21 E+05	5.00
Mass1 (1000kg)	Lumped mass at 2^{nd} floor	45.65	±5	43.37	-4.99
Mass2 (1000kg)	Lumped mass at 3^{rd}-5^{th} floor	36.53	±5	38.36	5.01
Mass3 (1000kg)	Lumped mass at top floor	54.84	±5	52.1	-5.00
WS1 (mm)	Effective width of concrete slab at 2^{nd}-5^{th} floor	762	±50	1143	50.00
WS2 (mm)	Effective width of concrete slab at top floor	914.4	±50	1371	49.93

The optimization problem defined previously was solved by the Nelder-Mead method. The results listed in Table 4-5 show that the errors between the identified and updated model reduced to 1% and 5.78% for the first and second natural frequency, respectively. Meanwhile, the second mode shape was improved, which gave a value of 0.981 for the MAC. With this refined finite element model, hybrid simulation was conducted to yield a seismic response prediction with higher accuracy.

Table 4-5 Comparison of frequency and mode shape between the original and updated

Mode	Original FE model			Updated FE model		
	frequency (Hz)		*MAC*	frequency (Hz)		*MAC*
	value	error (%)		value	error (%)	
1	0.688	-4.312	0.999	0.712	-1.001	0.999
2	1.956	-8.769	0.975	2.020	-5.784	0.981

4.6 Hybrid Simulation & Fragility Analysis

The calibrated Burbank building model after **Model Calibration** and ground motions from **Hazard Characterization** were used to perform the hybrid simulation and to derive fragility curves in NISRAF.

4.6.1 Performance Limit State

Three performance limit states are specified in this step, namely, the immediate occupancy (IO), the life safety (LS) and the collapse prevention (CP). Interstory drift angles (ISDAs) 0.7%, 2.5% and 5% are assigned to IO, LS and CP performance level, respectively (FEMA, 2000b).

4.6.2 Seismic Input

Ground motions representative of the local hazard characterization are essential in order to capture the realistic structural response. In addition, various ground motions should be considered to avoid excessive scaling on them. Excessive scaling is unrealistic and unreasonable particularly when motion has higher earthquake intensity. Based on the above considerations, the site specific synthetic ground motions with various hazard levels, generated for the Burbank sits, were selected as the earthquake demand in this example. To avoid excessive scaling, records related to 10%, 5% and 2% probability of exceedance in 50 years hazard level are used to derive fragility curves for immediate occupancy, life safety and collapse prevention performance limit state, respectively.

4.6.3 Hybrid Simulation

The calibrated Burbank building model and ground motions from hazard characterization analysis were used to verify the extension of the hybrid simulation to fragility analysis as well as the integration of hybrid simulation in earthquake impact assessment. The calibrated 2-D structure model was divided into two sub-structures, namely, the column (the lower part of the left exterior column at the first floor) and the frame (the remaining structure). The frame module was simulated using Zeus-NL, while the column module—replaced by a small scale aluminum specimen (Figure 4-10)—was tested in the laboratory.

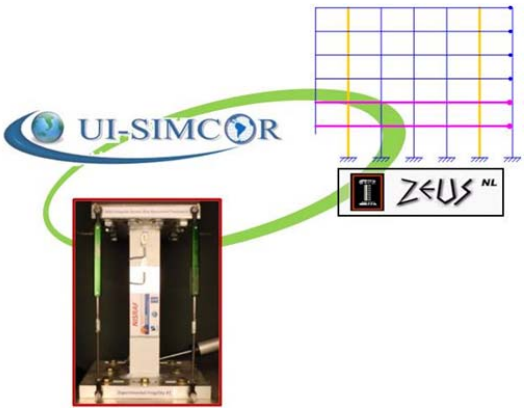


Figure 4-10 Hybrid simulation with two sub-structures (column and frame)

4.6.4 Hybrid Fragility Analysis

Based on the lognormal distribution assumption, mean value of seismic intensity from testing along with dispersions from literature are used to derive the hybrid fragility curves. In this study, mean value of PGA from hybrid simulation tests and dispersions from literature (FEMA, 2000a; Cornell et al., 2002; Yun and Foutch, 2000) were used to derive the fragility curves for this 6-story steel building in Burbank. In the following section, mean PGA values from hybrid simulation tests are presented first, followed by discussions on the dispersions found in literature.

4.6.4.1 Mean PGA Values from Hybrid Simulation

Hybrid simulation results under different synthetic ground motions (10%, 5% and 2% probability of exceedance in 50 years for immediate occupancy, life safety and collapse prevention performance levels, respectively) were used to derive the mean PGA value for each performance level. Step-by-step procedure to derive mean PGA value is given below:

- *Step 1:* 10% probability of exceedance in 50 years ground motion is selected as seismic input for hybrid simulation to derive mean PGA value for immediate occupancy limit state.
- *Step 2:* Interstory drift angle (ISDA) is calculated based on testing results. Comparison of ISDA between the calculated one and the target one (0.7% ISDA for immediate occupancy performance limit state, for example) is then made.
- *Step 3:* Hybrid simulation is resumed (replaced with new specimen if nonlinear behavior occurs in previous test) with seismic input multiplied by a scale factor (calculated based the difference in *Step 2*), if the difference exceeds criterion (±5% difference, for example).
- *Step 4:* Iterations from Step1 to *Step 3* continues till the criterion is met.
- *Step 5:* Once the calculated ISDA matches the defined ISDA, PGA value of current (scaled) record is assigned as the mean PGA value for immediate occupancy performance limit state.

The above procedure is an example of how to drive the fragility curve for IO limit state, while similar procedures were applied to derive curves for LS and CP limit states using 5% and 2% probability of exceedance in 50 years ground motions, respectively.

Figure 4-11 shows the number of hybrid simulation tests used to derive the mean PGA values. Table 4-6 lists the target ISDA (ISDA, interstory drift angle, are defined in previous section for this study) as well as the mean PGA values from hybrid simulation tests.

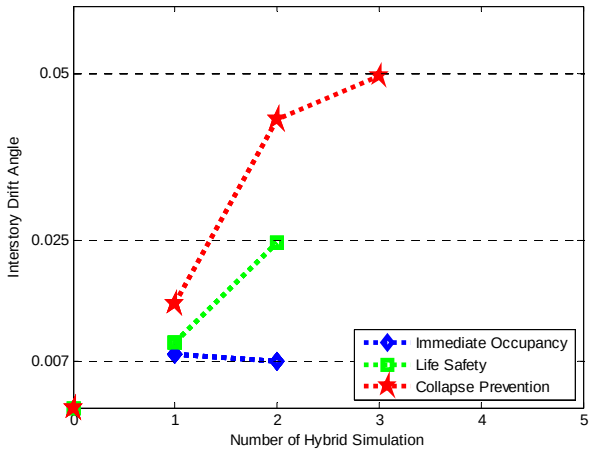


Figure 4-11 Number of hybrid simulation tests to derive fragility curves

Table 4-6 Interstory drift angle (target ISDA) and PGA from hybrid simulation tests

Performance Level	Immediate Occupancy	Life Safety	Collapse Prevention
Interstory drift angle (%)	0.7	2.5	5.0
Mean PGA (g)	0.545	1.627	2.777

4.6.4.2 Dispersions from Literature

Dispersion, a statistics vocabulary, represents the uncertainty term in fragility relationships. Due to the limited number of tests in the hybrid fragility analysis, it is unreasonable and also unrealistic to regress dispersion based on few testing results. Therefore, dispersions of the proposed hybrid fragility analysis are found from the literature.

FEMA 350 (FEMA, 2000a), the recommended seismic design criteria, is specially developed for new steel moment frame buildings. In FEMA 350, as well as in the literature (Cornell et al., 2002; Yun and Foutch, 2002), a method used to evaluate seismic

behavior of steel moment frame buildings is proposed. Within this method, uncertainties for different building height, beam-connection type, analysis procedure (linear or nonlinear, static or dynamic), and local and global failures under different performance levels (IO and CP) are tabulated (Table A-3 in FEMA 350) or illustrated in the content. Table 4-7 lists dispersions which will be utilized to derive hybrid fragility curves.

Table 4-7 Mean PGA value and dispersions for mid-rise steel building fragility curves

Performance Level	Immediate Occupancy	Life Safety	Collapse Prevention
Dispersion	0.311	0.328	0.346

4.6.4.3 Hybrid Fragility Curves

Finally, based on mean PGA value and dispersion, and following lognormal distribution assumption, fragility curves were generated and are shown in Figure 4-12.

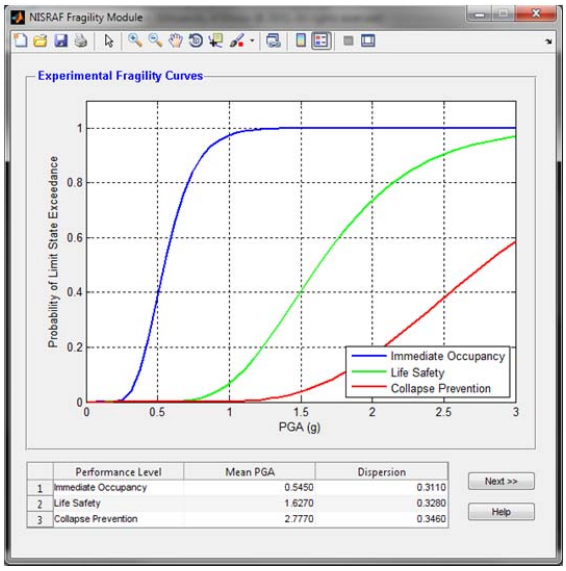


Figure 4-12 Hybrid fragility curves for mid-rise steel moment resisting frame building in Burbank

4.7 Impact Assessment

Finally, with the generated compatible hazard map and fragility curves, MAEviz under NISRAF was conducted to perform earthquake impact assessment (Figure 4-13). Only 15% probability for damage occurred in the immediate occupancy limit state. The results met with the post-earthquake report made by Applied Technology Council (ATC, 2001), which reported slight damage observed to this building from the Northridge earthquake.

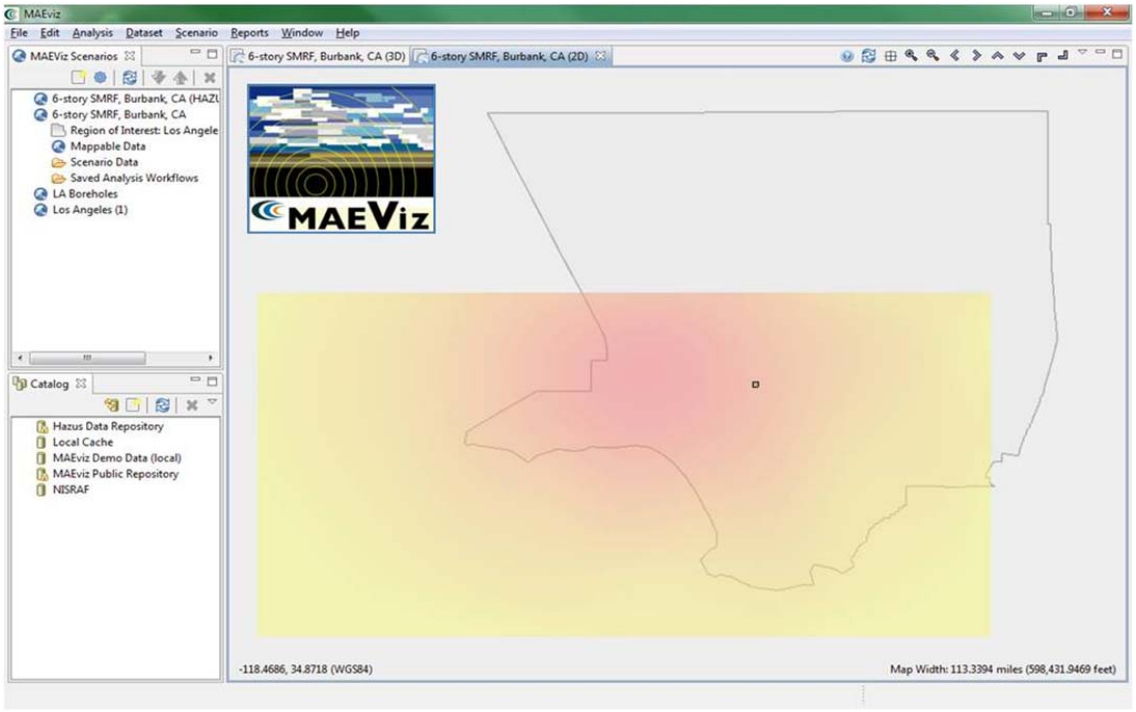


Figure 4-13 Impact assessment for Burbank building in MAEviz

5. Reference

Anderson, J. C. and Bertero, V. V., 1991. Seismic performance of an instrumented six-story steel building, *Report UCB/EERC-91/11*, University of California, Berkeley, CA

Applied Technology Council, 2001. *Database on the performance of structures near strong-motion recordings: 1994 Northridge, California, earthquake*, Report No. ATC-38, Redwood City, CA

Campbell, K. W. and Bozorgnia, Y., 2008. "NGA Ground Motion Model for the Geometric Mean Horizontal Component of PGA, PGV, PGD and 5% Damped Linear Elastic Response Spectra for Periods Ranging from 0.01 to 10 s," *Earthquake Spectra*, 24(1): 139-172

Cornell, C. A., Jalayer, F., Hamburger, R. O., and Foutch, D. A., 2002. "Probabilistic Basis for 2000 SAC Federal Emergency Management Agency Steel Moment Frame Guidelines,'' *Journal of Structural Engineering*, 128(4): 526-533.

Department of Conservation, Division of Mines and Geology, 1998. *Seismic hazard zone report for the Burbank 7.5-minute quadrangle, Los Angeles County, California,* Seismic Hazard Zone Report 016

Federal Emergency Management Agency, 2000a. *Recommended Seismic Design Criteria for New Steel Moment-Frame Buildings*, Report No. FEMA-350, Washington D.C.

Federal Emergency Management Agency, 2000b. *Prestandard and Commentary for the Seismic Rehabilitation of Building*, Report No. FEMA-356, Washington D.C.

Fumal, T. E., Gibbs, J. F., and Roth, E. F., 1979. *In-situ measurements of seismic velocity at 19 locations in the Los Angeles, California region*, SMIP geotechnical report No. 131, U.S. Geological Survey

Gasparini, D. A. and Vanmarcke, E. H., 1976. *Simulated Earthquake Motions Compatible with Prescribed Response Spectra*, Evaluation of Seismic Safety of Buildings Report No.2, Massachusetts Institute of Technology

Hashash, Y., Groholski, D.R., Phillips, C.A., and Park, D., 2009. *DEEPSOIL V3.5beta, User Manual and Tutorial,* Department of Civil and Environmental Engineering, University of Illinois at Urbana-Champaign, Urbana, IL

Li, J., Lin, S. -L., Zong, X., Spencer, B. F., Elnashai, A. S. and Agrawal, A. K., 2009. "An Integrated Earthquake Impact Assessment Framework," *ANCER Annual Meeting*, August 13-14, Urbana, IL

Lin, S.-L. 2010. *An Integrated Earthquake Impact Assessment System*, Ph.D. dissertation, Civil Engineering, University of Illinois at Urbana-Champaign, Urbana, IL

Park, K. S., Kwon, O. S., Spencer, B. F., Elnashai, A. S., 2007. *Tutorial for Beta version of SimBuild, Pre- and Post-processor for UI-SimCor*, Department of Civil and Environmental Engineering, University of Illinois at Urbana-Champaign, Urbana, Illinois.

Yun, S. Y., and Foutch D. A., 2000. *Performance prediction and evaluation of low ductility steel moment frames for seismic loads*, SAC Background Document SAC/BD-00/26, Richmond, CA: SAC Joint Venture.