FRAME MODELER 2D

An Open-Source Platform for the 2-Dimensional Modeling and Seismic Analysis of Buildings

Version 1.2309
Manual

Prepared by

Dr. Ahmed Elkady

National Infrastructure Laboratory (N|I|L), University of Southampton,

United Kingdom

September, 2023

UNIVERSITY OF
Southampton

Table of Contents

Disclaimer

Citation

Elkady, A. (2021). " Open-Source Platform for the 2-Dimensional Modeling and Seismic Analysis of Buildings." *SoftwareX, (under review)*.

Download and Support Links

GitHub repository: https://github.com/amaelkady/FM-2D

Registration for software updates: https://forms.gle/qLNwgGFS5vbCxmHB7

Video tutorials on YouTube: https://www.youtube.com/playlist?list=PLz_XdUL-6Y_m10fBEMmzPKqI00wdzbDUN

1. Installation

The step-by-step installation procedure of the software, in Windows, is discussed in this section. Note that FM-2D v1.2010 executable is only made available for Windows machines, however, the software can run directly on a Mac machine using the MATLAB source code. Figure 1.1 shows all the files/folder that are downloaded by the user through the FM-2D's GitHub repository. These are as follows:

FM2D_Installer.exe this is the main installation file.
FM2D Manual.pdf this is FM-2D's manual and installation guide.
Supporting Documents this folder includes all the pre-formatted and template files that can be modified by the user and imported into the different modules in FM-2D.
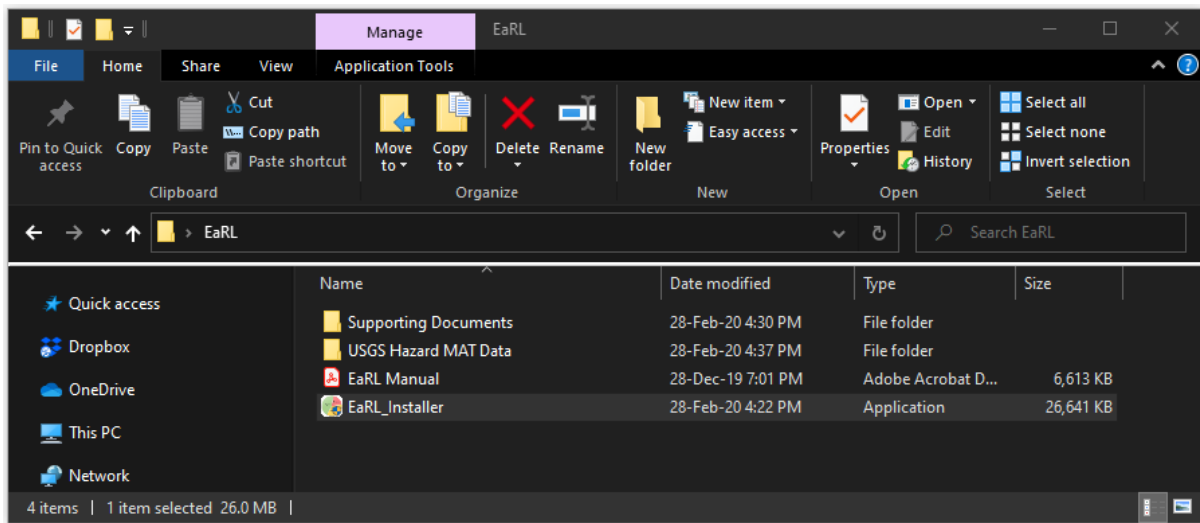
Figure 1.1. Downloaded software files and folders

The software can be installed by running the *FM2D_Installer.exe* executable file. As demonstrated in Figure 1.2, the installation process is similar to any software. The user is prompted to different windows to approve the installation process and set the installation directory. By default, the installation directory is "*C:\Program Files\University of Southampton\FM_2D*". Note that if MATLAB runtime was not already installed on the local PC, the user will be prompted to download the latest runtime environment (about 700Mb). An internet connection is required in this case.

Once installed, the executable file FM_2D.exe, as well as other files, can be found in the default installation directory under "*.....\University of Southampton\FM_2D\application\FM_2D.exe*" as shown in Figure 1.3. Since FM-2D runs analysis using OpenSEES, users shall independently install Tcl/Tk and OpenSEES on their local PC. OpenSEES installation guide can be found at https://opensees.berkeley.edu/OpenSees/user/download.php. After installation, the user must make sure the executable *OpenSEES.exe*, which was already generated during FM-2D installation procedure, is working properly from the installation directory. If not, then please re-check your OpenSEES installation procedure and note that in Windows machines the OpenSEES "lib" folder would typically need to be located within the "*...\University of Southampton\FM_2D*" directory. Finally, it is important to note that FM-2D should be opened by running the *FM_2D.exe* file from the installation directory using administrator privileges.

Otherwise, internal processes that require reading and writing files, in the installation directory, may not work properly.

The users are highly recommended to register their information online using the following form link: https://forms.gle/CkfSBmjhqATJZQJj6; so that they are able to receive e-mails notification regarding future releases and/or bug-fixes.

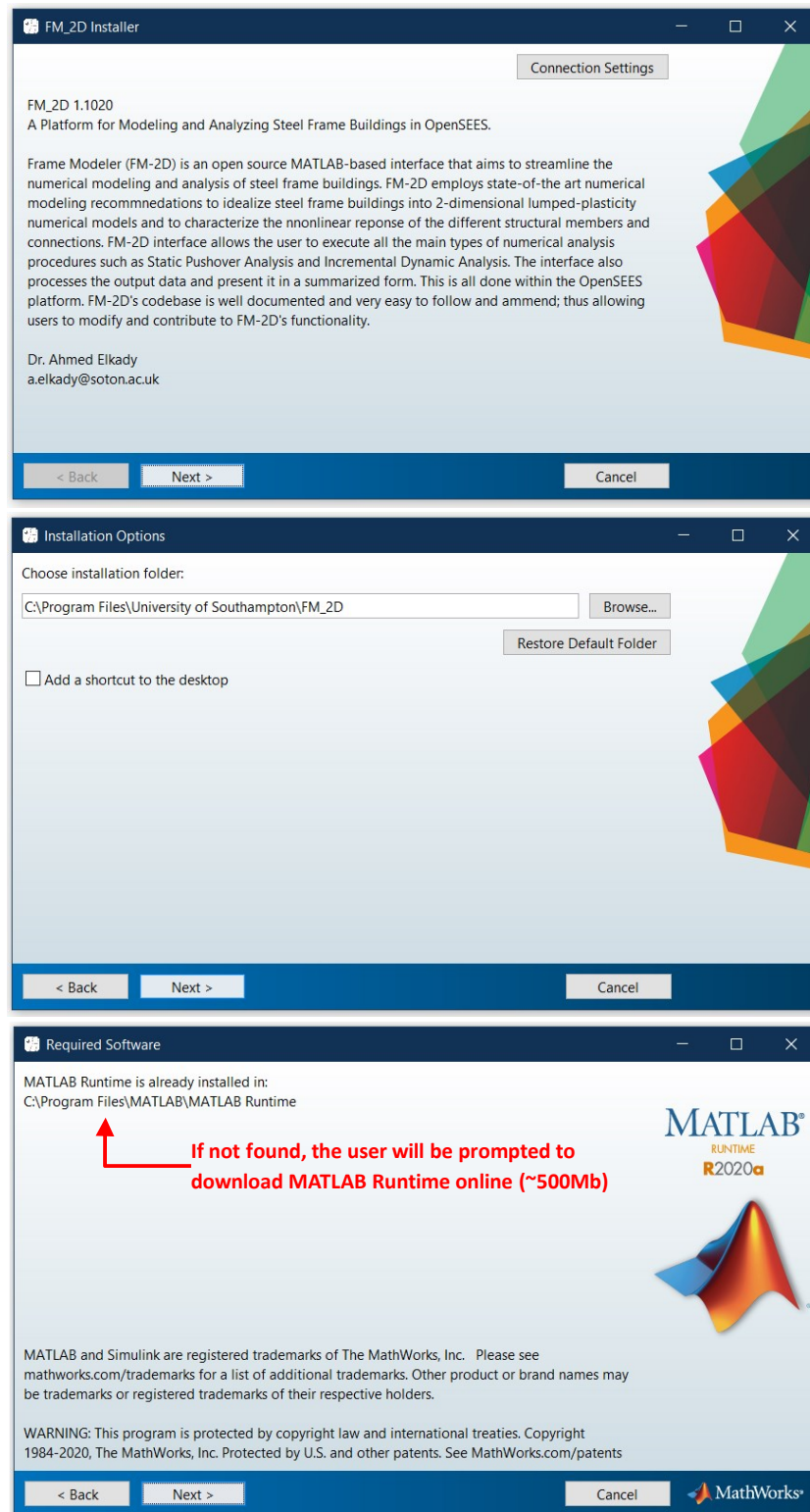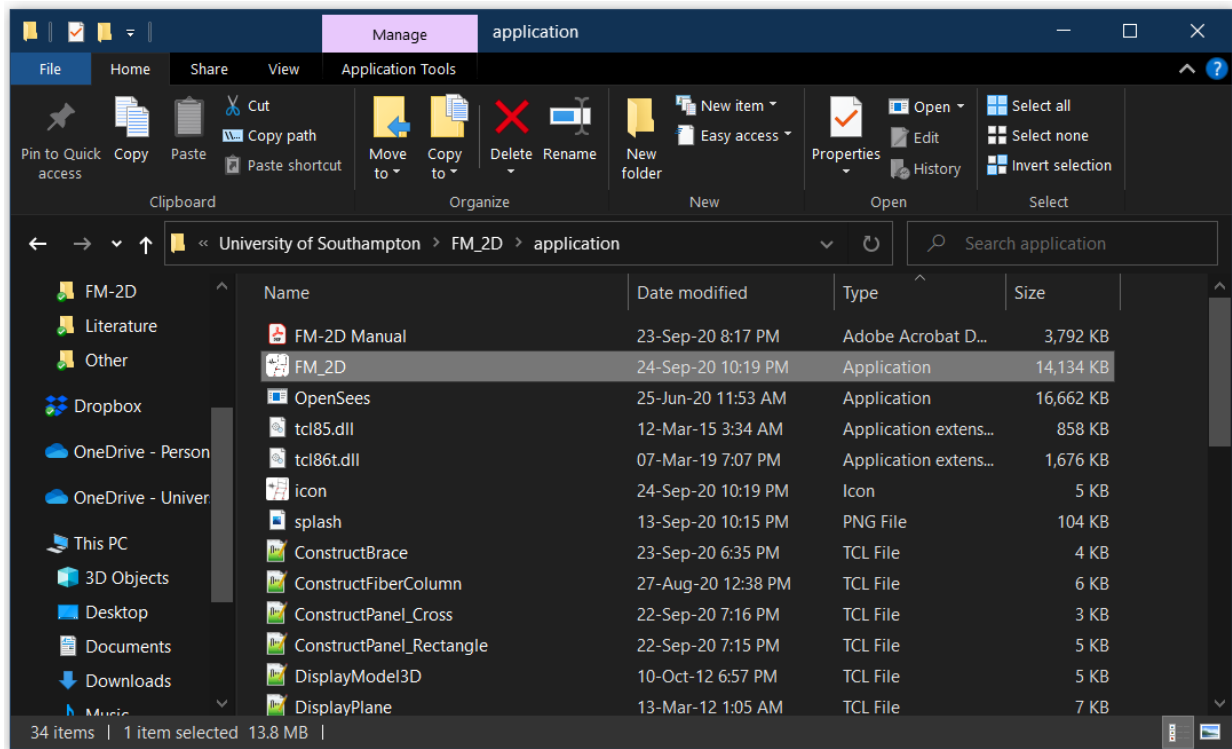Figure 1.2. Installation steps

Figure 1.3. Installed files and folders in the default installation directory

2. Introduction

The performance-based earthquake engineering (PBEE) framework (SEAOC 1995; Cornell and Krawinkler 2000; FEMA 2000; Porter 2003; Deierlein 2004; Krawinkler and Miranda 2004; Moehle and Deierlein 2004; Aslani and Miranda 2005; Yang et al. 2009; FEMA 2012a; Hamburger et al. 2012) has been increasingly used in designing and retrofitting buildings, particularly within the past decade. At its core, this framework relies on the availability of robust response data for a building under a given earthquake scenario in order to be able to reliably assess/quantify structural performance, in terms of damage and economic losses, beyond the scope of conventional building codes. Structural response data includes global engineering demand parameters (EDP) such as the story-drift ratios (*SDR*), the residual drift rations (*RDR*) and peak floor accelerations (*PFA*). Local EDP includes for instance connection and member rotational demands. Obtaining structural response data is arguably the most demanding and time consuming step of the PBEE framework. This step involves 1) numerical modeling the building in a structural analysis platform, 2) determine the nonlinear hysteretic response parameters of the different connections and members, 3) defining the loads and masses, 4) conducting static and dynamic analysis procedures using a number of ground motion records, and finally, 5) reading and processing response data. This is laborious step can take weeks and sometime months to complete. Most importantly, this step requires a skilled and experienced user who knows the specifics of state-of-the-art numerical modeling guidelines and simulation techniques. While existing software packages (Ramirez and Miranda 2009; FEMA 2012b; SP3 2017; Elkady et al. 2018), for economic loss assessments, provide assistance with most of the PBEE framework steps (such as the damage fragility data of structural component and contents and the associated consequences), the burden of obtaining structural response data remain entirely on the user. Some software provides tools to generate structural response data based on simplified methods. As such, the accuracy of the generated data can be low and the associated uncertainty in the corresponding computed economic losses can be large.

To that end, an open-source MATLAB-based (2019) standalone software is developed and presented herein. The software is named Frame Modeler-2D (FM-2D) and it aims to streamline and speed the numerical modeling and analysis of steel frame buildings. Using the Open System for Earthquake Engineering Simulation Platform (OpenSEES) (Mckenna 1997; Mazzoni et al. 2006), FM-2D builds robust 2-dimensional (2D) numerical models of steel frame buildings using state-of-the-art numerical modeling guidelines, conducts different types of nonlinear analysis procedures and processes and reports the structural response data in a comprehensive manner. FM-2D has a well-structured and documented codebase; thus, allowing it be a platform for engineers and researchers to further develop and contribute to its modeling and simulation capabilities.

3. Operational Outline

FM-2D's operational outline is illustrated in Figure 3.1. A project definition follows a sequential three-step procedure. In brief, the procedure involves: 1) selecting the frame type and the basic numerical model parameters, 2) defining the building geometry, member sizes, loads and materials, and 3) defining the analysis type and its parameters. Using the user-defined data, FM-2D constructs and generates *tcl* files of the numerical model, executes the analysis in OpenSEES, processes the analysis data, stores structural response in *txt* files for the user to visualize or to use externally with other software packages. The different graphical user-interface (GUI) modules in FM-2D that are used to define the model, building and analysis data are discussed in details in Section 4. The technical details for generating the 2D numerical models are discussed in Section 7.
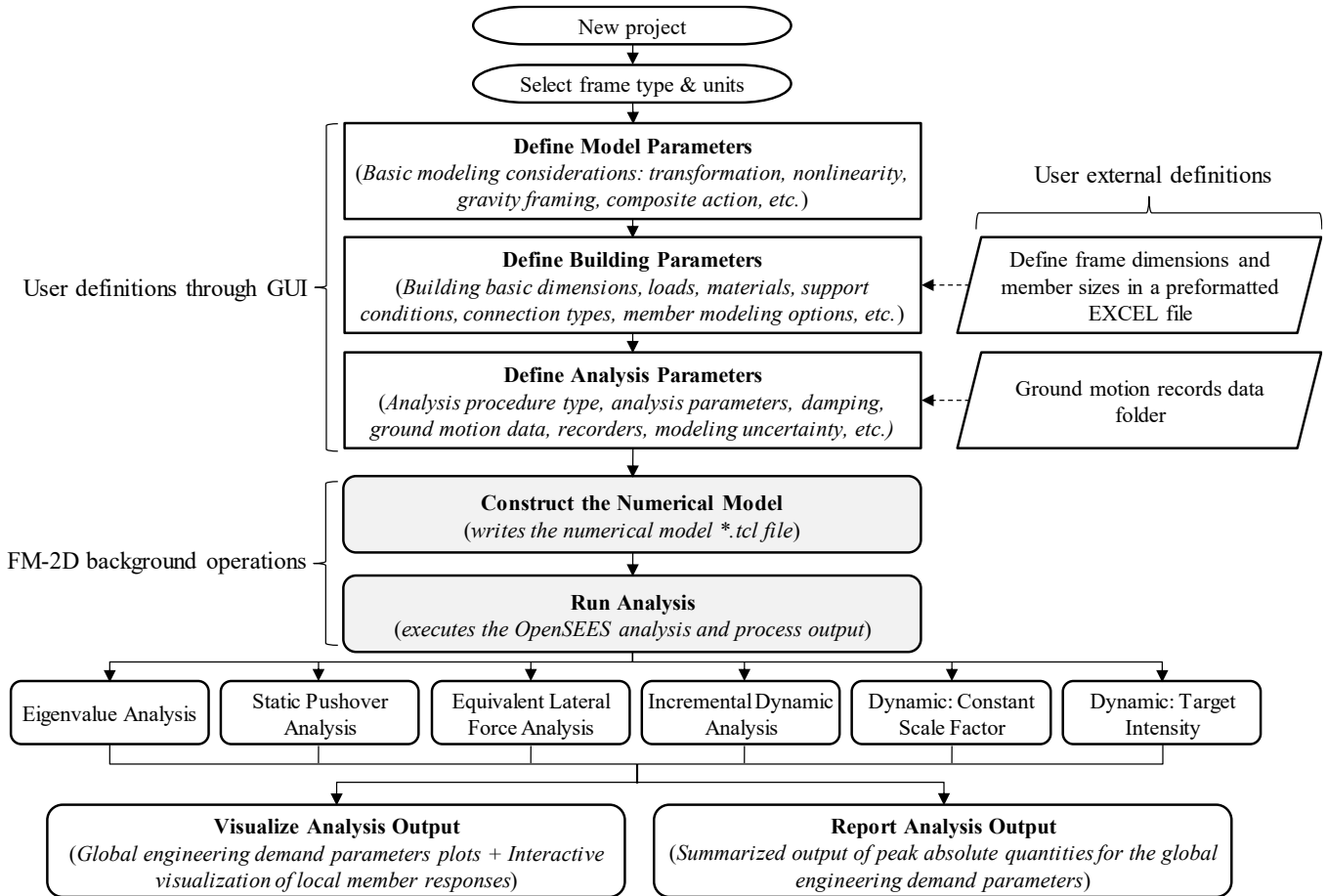
Figure 3.1. FM-2D operational outline

4. Interface Components

4.1 Main Console

Figure 4.1 illustrates the main console of FM-2D, which is divided into three main panels:

Panel A is the project file management panel, which includes the following buttons:

- o "New" create a new project file
- o "Open" open an existing project file
- o "Save As" save an open project under a different name
- o "Scope" shows a summary of the current project definitions

Panel B includes three buttons that define the three main components of any project:

- o "Model Parameters" defines the main numerical modeling parameters
- o "Building Parameters" defines the building geometry and numerical modeling details
- o "Analysis Parameters" defines the type of analysis and its parameters

Panel C includes the execution buttons:

- o "Get period" for probing the building first-mode period, seismic weight and mass.
- o "Build tcl file" for building a standalone *tcl* file of the building model.
- o "Run Model" for running the numerical analysis.
- o "Visualize" for visualizing the numerical analysis results.
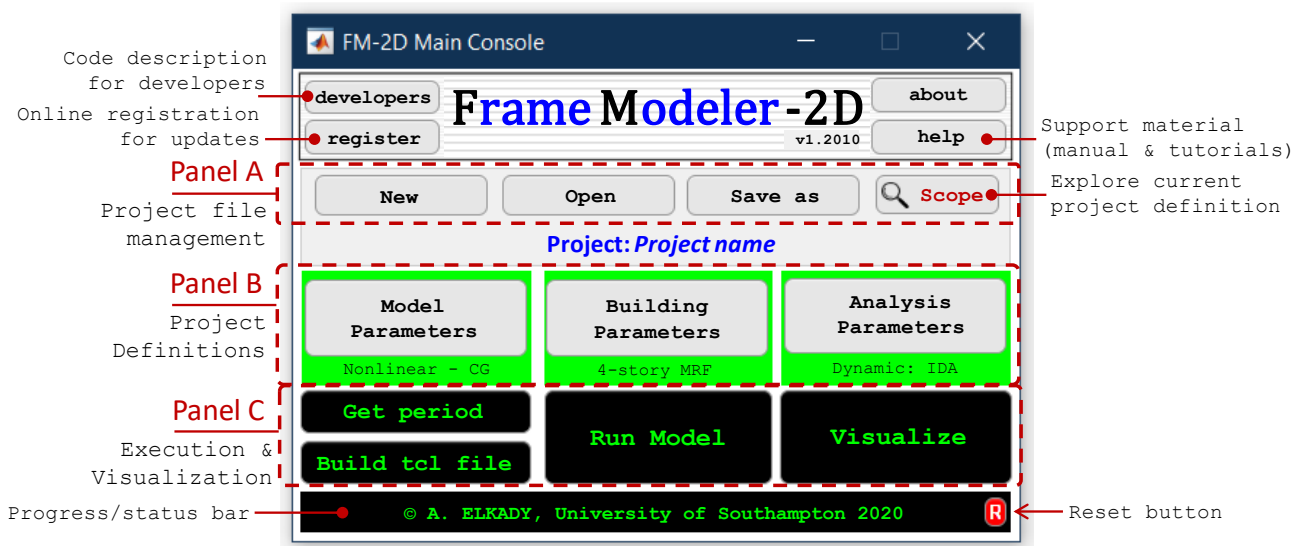
Figure 4.1. FM-2D v1.2010 main console

The FM-2D project is saved as a transferable MATLAB *.mat* file and can be loaded through the toolbox at any time from different machines. When starting a new project, the user is prompted to select the building frame type (MRF or CBF) and the project units ("kN & mm" or "kip & in") as shown in Figure 4.2. Interface for selecting the frame type and project units. The user can also provide an optional description of the project for reference.
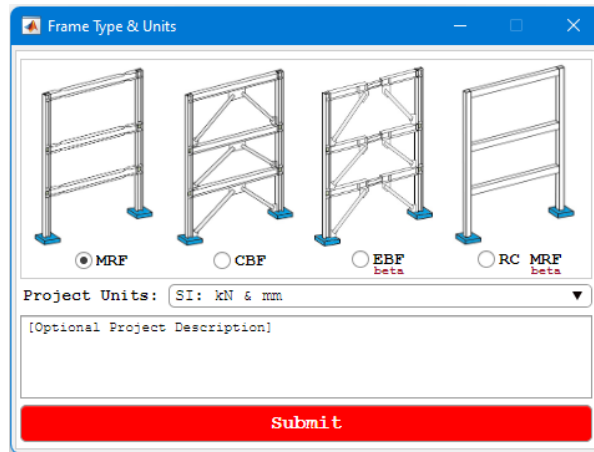
Figure 4.2. Interface for selecting the frame type and project units

4.2   Model Parameters

As demonstrated in Figure 4.3, FM-2D idealizes a given steel frame building, and its response in given orthogonal direction, as a 2-dimensional numerical model. Particularly, the idealized 2D model comprises one of the main lateral-force resisting planar frames in the considered orthogonal direction (noted hereafter simply as MF). In FM-2D v1.2010, this main frame (MF) can be either a moment-resisting frame (MRF) or a concentrically-braced frame (CBF). The rest of the building, that is the gravity framing system (GFS) tributary to the considered MF, is represented by a single-bay fictitious equivalent gravity frame (EGF) as proposed by Gupta and Krawinkler (1999) and discussed in Lignos et al. (2011), Lignos et al. (2013) and Elkady and Lignos (2015). The EGF captures the strength and stiffness of the gravity framing system (if chosen be considered by the user) and most importantly, it is used to represent the gravity P-Delta loads of the building in 2D. The EGF is connected to the MF using rigid or flexible truss floor links, that are meant to transfer the floor inertia forces between the two systems. The 2D numerical model uses mainly the lumped plasticity modeling approach (also known as the point plasticity or concentrated plasticity) to idealize the structural members and connections behaviors. This is discussed in detail in Section 7.
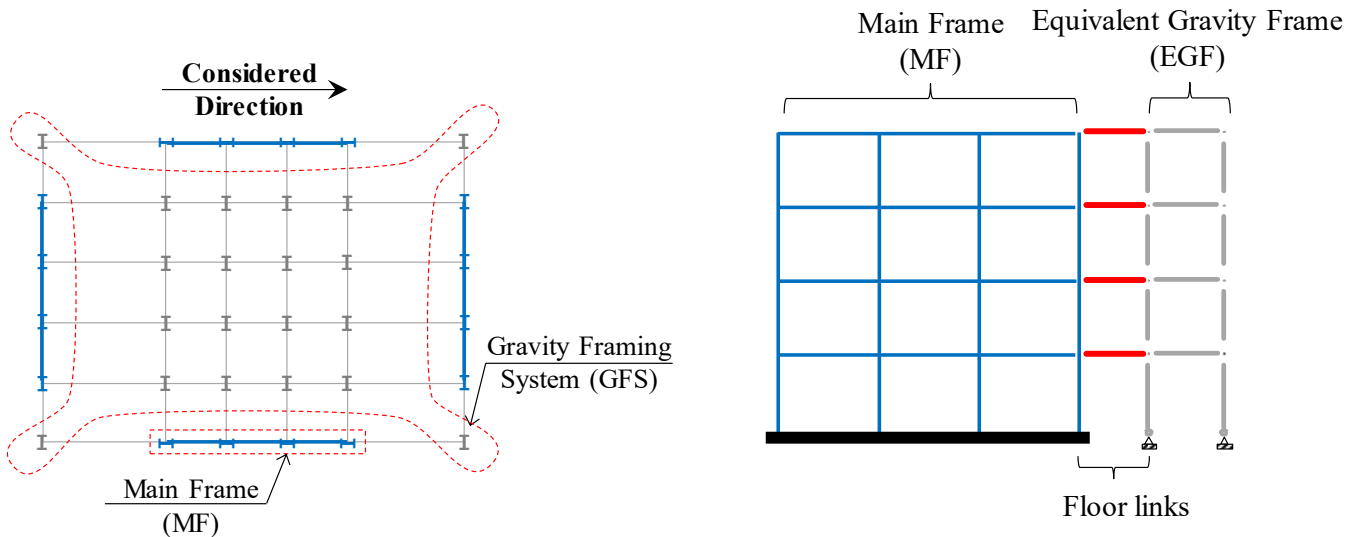
Figure 4.3. Building numerical representation in 2D

The first step in an FM-2D project definition is to specify the "Model Parameters". When the "Model Parameters" button is clicked, the user is prompt to the module shown in Figure 4.4. Model parameters module, regardless of the selected frame type. The module contains two tabs, described as follows:

4.2.1   Modeling Options tab

Geometric Transformation: from the dropdown menu, select the type of geometric transformation that will be assigned to the beam and column elements in your model: Linear, P-Delta or Correlational.   Refer to OpenSEES wiki for more details: https://opensees.berkeley.edu/wiki/index.php/Geometric_Transformation_Command

Numerical Model Type: from the dropdown menu, select whether the numerical model would be Linear or Nonlinear. The Linear option can be used to create an elastic linear model, which can be used for instance to conduct an equivalent lateral force analysis as per ASCE (2016).

Panel Zone Dimension & Deformation: mark the checkbox to consider the column web panel zone –rectangular– dimension and its rotational flexibility (i.e., shear distortion deformation) under shear loads. If the check box is not marked, the panel zone flexibility is not considered, where it will be idealized as a rigid cruciform shape as discussed in detail in section 7.4 (refer to Figure 7.2).

Rigid Floor Diaphragm: mark the checkbox to tie all nodes at each floor level with a diaphragm constraint (i.e., all nodes have equal lateral degree of freedom).
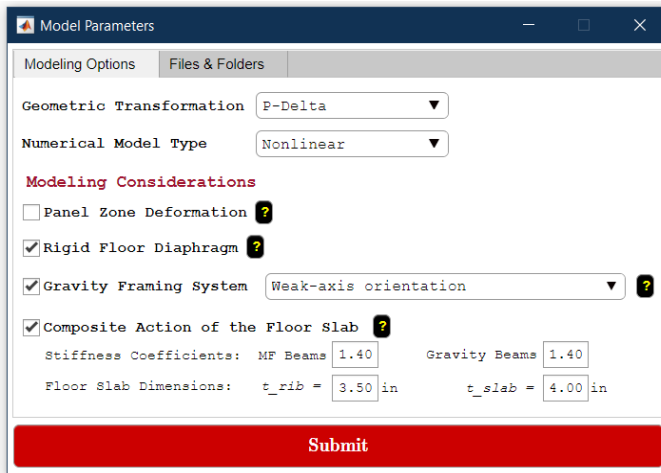
Gravity Framing System: mark the checkbox to consider the strength and stiffness of the gravity framing system's columns, beams and beam-to-column shear connections, as per Elkady and Lignos (2015). When considered, the user shall also specify (using the dropdown menu) the orientation of the gravity columns cross-section axis with respect to the considered orthogonal building direction.

Composite Action of the Floor Slab: mark the checkbox to consider the effects of the composite action provided by the floor composite –concrete on steel deck– slab when modeling the MF and GFS beams and beam-to-column connections, as per the recommendations of Elkady and Lignos (2014, 2015). When considered, the user shall specify the amplification factors for the bare steel elastic stiffness ratio of the MF and gravity beams. By default, these factors are taken equal to 1.4. The user shall also specify the floor slab dimensions including the steel deck rib height (t_{rib}) and the thickness of the concrete slab above the steel deck rib (t_{slab}). By default, those are taken as 3.5 inches and 4.0 inches, respectively.
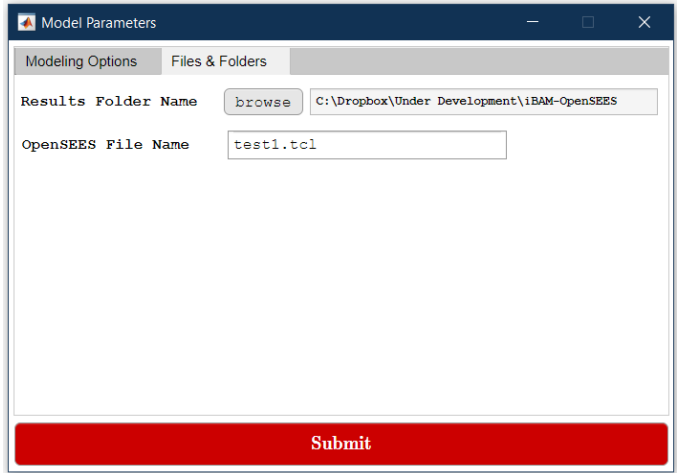
4.2.2 Files & Folders tab

Results Folder Name: using the "browse" button, the user can select a directory to save the analysis results. By default, this directory is taken the same as the project file directory.

OpenSEES File Name: specify in the textbox the name of the numerical model *tcl* file that will be created by FM-2D (e.g., *TestFrame.tcl*). Note that this name should not include any spaces or special characters. By default, this filename is taken the same as the project name minus the spaces (if any).

(a)   (b)

Figure 4.4. Model parameters module

4.3 Building Parameters

The second step is to define the "Building Parameters". When the "Building Parameters" button is clicked, the user is prompt to a module containing four tabs, described as follows:

4.3.1 Building Geometry tab

To aid the subsequent discussion and description of geometric parameters, Figure 4.5 shows the plan layout of a generic building with its basic components annotated. In this figure, the two orthogonal directions are labeled "Direction 1" and "Direction 2". The former is assumed here to be the "considered" direction for the numerical model/analysis to be conducted.

The "Building geometry" tab is shown in Figure 4.6a, based on the CBF option. Note however that this tab is almost similar to that of the MRF option. In this tab, the user shall specify some basic geometric and configuration parameters including:

1) the building dimensions in the plan layout.
2) the floor area tributary to the MF's exterior and interior columns as demonstrated in Figure 4.5.
3) the number of MF in the considered direction (e.g., two MFs in the generic building in Figure 4.5).
4) the MF's number of stories and bays. Note that for CBFs, FM-2D v1020 currently allows only single bay CBFs.
5) the GFS's number of columns and beams. Note that the number of GFS columns should be that in the entire building layout (i.e., 20 columns in the generic building) and 2) the number of GFS beams should be that in the considered orthogonal direction only (i.e., 24 beams in the generic building).
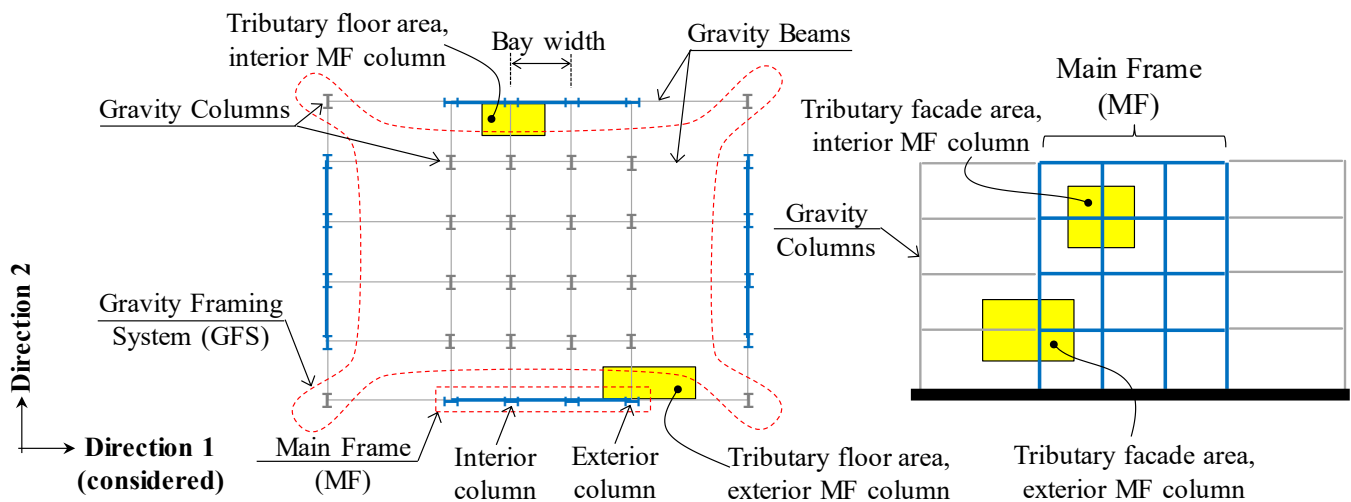

Figure 4.5. Components of a generic regular building plan and elevation layout

Finally, the user shall define building story heights, bay widths and member sizes. This is done in a pre-formatted EXCEL file that is provided within FM-2D supporting documents (see *docs* folder in GitHub). The EXCEL file includes a number of sheets as follows:

For both MRFs and CBFs:

W_bay: specify the width of the MF bays (using the same units of the projects).

H_story: specify the heights of the MF stories (using the same units of the projects) as demonstrated in Figure 4.7a.

MF Columns: specify the cross-sections of the MF columns at each story and axis as demonstrated in Figure 4.7b. Note that user can choose any cross-section of those included in the integrated database, using the same cross-section notation. Also note that if there is a splice at a given story i (i.e., change in column size), the user shall specify –for that story i– the column cross-section size of the lower story i-1. For other section that are not in the integrated section database, the user can define custom I-shaped sections by entering the following "**BU** $d\mathbf{x}b_f\mathbf{x}t_w\mathbf{x}t_f$" where BU stands for built-up, d is the section depth, b_f is the section flange width, t_w is the web thickness, and t_f is the flange thickness. For instance: BU 500x200x12x15. Note that these section dimensions should be in the same units as the defined project.

MF Beams: specify the cross-sections of the MF beams at each floor and bay as demonstrated in Figure 4.7c. Note that user can choose any cross-section of those included in the integrated database, using the same cross-section notation. For other section that are not in the integrated section database, the user can define custom I-shaped sections by entering the following "**BU** $d\mathbf{x}b_f\mathbf{x}t_w\mathbf{x}t_f$" where BU stands for built-up, d is the section depth, b_f is the section flange width, t_w is the web thickness, and t_f is the flange thickness. For instance: BU 500x200x12x15. Note that these section dimensions should be in the same units as the defined project.

GFS Columns: specify the cross-sections of the GFS columns at each story. Note that user can choose any cross-section of those included in the integrated database, using the same cross-section notation. Also note that GFS columns are assumed to be the same at all axes.

GFS Beams: specify the cross-sections of the GFS beams at each floor. Note that user can choose any cross-section of those included in the integrated database, using the same cross-section notation. Also note that GFS beams are assumed to be the same in all bays.

Column Splice: specify the presence of a splice at each story. Specify "0" for no splice and "1" for splice, as demonstrated in Figure 4.7d. The user can also specify the relative location of the splice within the respective story height, with respect to the bottom floor. By default, the splices are allocated at mid-height of the story (i.e., relative location = 0.5). Also note that the splice connection is assumed, in the current release, to be fully-rigid.

Doubler Plates: specify the total thickness of the doubler plates in the column web panel zone region at each floor and axis (if any) as demonstrated in Figure 4.7e.

Fs Profile: specify the force coefficients for the floor links at each floor as demonstrated in Figure 4.7f. These coefficients will be multiplied by the reference force, F_s, specified to under the "Supports & Connections" tab, to obtain the "limiting force" that will be assigned to floor truss link. Note that these

coefficients are set equal to 1.0 by default if the F_s profile is assumed constant. These coefficients need only be specified if a non-uniform F_s profile is selected.

For CBFs only:

Braces: specify the cross-sections of the braces at each story as demonstrated in Figure 4.7g. Note that user can choose any cross-section of those included in the integrated database, using the same cross-section notation.

CGP t_plate: specify the thickness of the corner gusset plate (CGP) at each story as demonstrated in Figure 4.7h.

MGP t_plate: specify the thickness of the mid-span gusset plate (MGP) at each story.

CGP L123: specify the characteristic lengths L_1, L_2 and L_3 of the corner gusset plate (CGP) at each story as demonstrated in Figure 4.7i (refer to Hsiao et al. (2013) and Figure **** details about these characteristic lengths).

MGP L123: specify the characteristic lengths L_1, L_2 and L_3 of the mid-span gusset plate (MGP) at each story.

CGP L_offset: specify the offset length between the brace end and beam-to-column centerlines intersection, for the corner gusset plate (CGP) at each story (refer to Figure 4.8).

MGP L_offset: specify the offset length between the brace end and beam-to-column centerlines intersection, for the mid-span gusset plate (MGP) at each story (refer to Figure 4.8).

CGP L_connection: specify the length of the connection between the brace and the corner gusset plate (CGP) at each story (refer to Figure 4.8) as demonstrated in Figure 4.7j.

MGP L_connection: specify the length of the connection between the brace and the mid-span gusset plate (MGP) at each story (refer to Figure 4.8).

MGP W_plate: specify the width of the mid-span gusset plate (MGP) at each story (refer to Figure 4.8).

L_brace: specify the end-to-end length of the brace at each story (refer to Figure 4.8).

For RC MRFs:

The sections for the RC frame beams and column are defined in the Excel sheet using the following notation (see template Excel sheet *RCMF_2Story.xlsx* for example):

For rectangular cross sections:

    H x *B* x *coverH* x *coverB* x *nBarTop* x *dBarTop* x *nBarBot* x *dBarBot* x *nBarInt* x *dBarInt* x *nBarShear* x *dBarShear* x *s*

For circular cross sections:

D x *cover* x *nBar* x *dBar* x *nBarShear* x *dBarShear* x *s*

where,

H and *B* are the section's depth and width, respectively

D is the section's diameter

coverH and *coverB* are the distances from section boundary to neutral axis of reinforcement in the vertical and horizontal direction, respectively

cover is the distance from section boundary to neutral axis of reinforcement

nBarTop, *nBarBot* and *nBarInt* are the number of reinforcing bars in the top layer, bottom layer and intermediate layer, respectively

dBarTop, *dBarBot* and *dBarInt* are the diameters of the reinforcing bars in the top layer, bottom layer and intermediate layer, respectively

nBarShear are the number of shear reinforcing bars (i.e., legs of the stirrup)

dBarShear is the diameter of the reinforcing bars in shear

s is the shear reinforcement (stirrup) spacing.

\# rho_T   Area ratio of longitudinal bottom reinforcement (tension)

\# rho_C   Area ratio of longitudinal top reinforcement (compression)

\# rho_I Area ratio of longitudinal middle reinforcement

\# rho_sh   Area ratio of transverse reinforcement in the plastic hinge region spacing

(a)   (b)

Figure 4.6. Building parameters module: Building geometry tab for (a) MRF and (b) CBF

(a)

	Height
Story 1	180
Story 2	156
Story 3	156
Story 4	156
Story 5	

W_bay | **H_story** | MF Col...

(b)

	Axis 1	Axis 2	Axis 3	Axis 4	Axis
Story 1	W24X103	W24X103	W24X103	W24X103	
Story 2	W24X103	W24X103	W24X103	W24X103	
Story 3	W24X103	W24X103	W24X103	W24X103	
Story 4	W24X62	W24X62	W24X62	W24X62	
Story 5					

... | **MF Columns** | MF Beams | Column Splice | GFS Columns | ...

(c)

	Bay 1	Bay 2	Bay 3	Bay 4
Floor 2	W21X73	W21X73	W21X73	
Floor 3	W21X73	W21X73	W21X73	
Floor 4	W21X57	W21X57	W21X57	
Floor 5	W21X57	W21X57	W21X57	
Floor 6				

... | MF Columns | **MF Beams** | Column Splice | GFS Columns

(d)

	Splice	Relative Location
Story 1	0	0.5
Story 2	0	0.5
Story 3	1	0.5
Story 4	0	0.5
Story 5		

... | MF Beams | **Column Splice** | GFS Columns

(e)

	Bay 1	Bay 2	Bay 3	Bay 4	Bay 5
Floor 2	0	0.3125	0.3125	0	
Floor 3	0	0.3125	0.3125	0	
Floor 4	0	0.3125	0.3125	0	
Floor 5	0	0.3125	0.3125	0	
Floor 6					

... | **Doubler Plates** | Fs Profile

(f)

	Fs Coeff.
Floor 2	1
Floor 3	1
Floor 4	1
Floor 5	1
Floor 6	

... | Doubler Plates | **Fs Profile**

(g)

	Bay 1	Bay 2
Story 1	HSS12.5X.500	
Story 2	HSS12.5X.500	
Story 3	HSS11.25X.500	
Story 4	HSS9.625X.500	
Story 5	HSS9.625X.375	
Story 6	HSS7.5X.312	
Story 7		

... | **Braces** | CGP t_plate | MGP t_plate | CGP L123

(h)

	Bay 1	
Story 1	0.625	
Story 2	0.625	
Story 3	0.625	
Story 4	0.625	
Story 5	0.5	
Story 6	0.5	
Story 7		

... | Braces | **CGP t_plate** | MGP t_plate

(i)

	L1	L2	L3
Story 1	11.8722	25.579	3.7405
Story 2	6.5786	23.3259	1.4874
Story 3	7.5914	10.9283	30.4097
Story 4	7.6514	18.0122	1.0753
Story 5	7.0529	14.5269	0.4768
Story 6	7.185	13.4558	3.9323
Story 7			

... | CGP t_plate | MGP t_plate | **CGP L123** | MGP L123

(j)

	Bay 1	Bay 2
Story 1	27	
Story 2	27	
Story 3	24	
Story 4	21	
Story 5	26	
Story 6	10	
Story 7		

... | **CGP L_connection** | MGP L_connection

Figure 4.7. Dimensions and member sizes definitions in the pre-formatted EXCEL file

Mid-Span Gusset Plate (MGP)
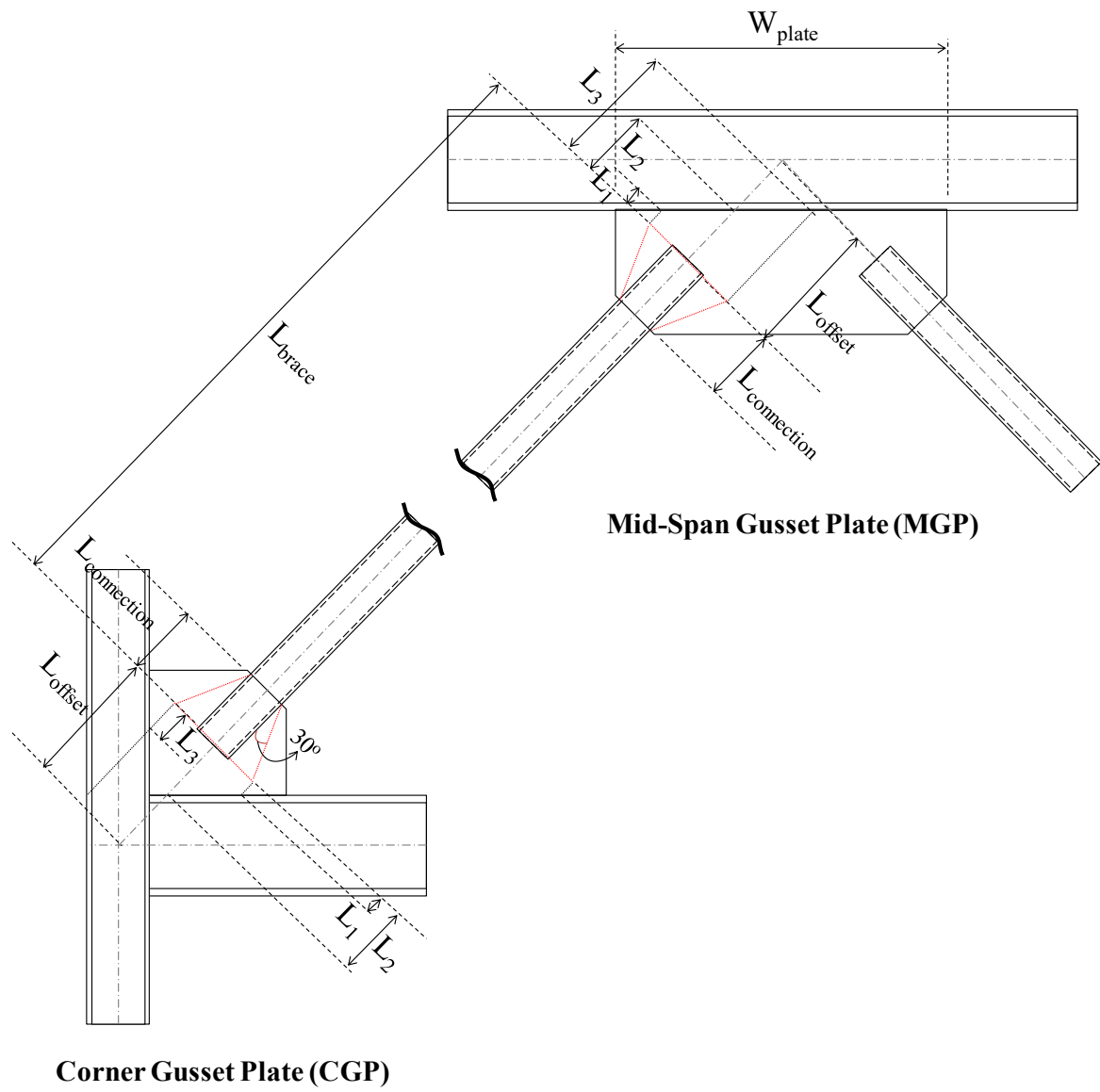
Corner Gusset Plate (CGP)

Figure 4.8. Geometric parameters used to define the CBF brace-to-gusset connection

4.3.2 Support & Connections tab

The "Support & Connections" tab for the MRF and CBF options is shown in Figure 4.9a and b, respectively. In these tabs, the user shall specify the support conditions and connection types. In particular:

MF Column Supports: select, from the dropdown menu, whether the main frame columns have a fixed or a pinned support at the ground floor level.

MF Beam-to-Column Connection: select, from the dropdown menu, the main frame beam-to-column connection type. For MRFs (see Figure 4.9a), two types are available: 1) reduced beam section connection (RBS) and 2) Welded/Bolted Full Section (i.e., other-than-RBS connection as per Lignos and Krawinkler (2011)). In case of the former, the user shall specify the additional geometric parameters *a*, *b* and *c* as demonstrated in Figure 4.9a. Further details about the connections geometries can be found in AISC (2016). For CBFs (Figure 4.9b), three types of connection are available to choose from: 1) Pinned (i.e., zero stiffness and strength) and 2) Shear, or 3) Moment (i.e., fully-rigid) connection. For CBFs, the connection type can be for odd- and even-numbered floors, which can be particularly useful for X-bracing configurations where no gusset plate connections are present at odd-numbered floor levels.

MF to EGF Link Condition: select, from the dropdown menu, the type of the MF-to-EGF floor link whether it is a rigid or flexible link. The floor link is meant to represent the flexibility of the floor diaphragm to main frame connection. In FM-2D, this link is represented with truss members with an infinitely-stiff bilinear axial force-axial deformation response as demonstrated in Figure 4.10. The bilinear response is defined using the break force, F_s, and the post break force stiffness, K_s. Typically, and by default in FM-2D, a rigid link is considered at each floor level; thus, a sufficiently large break force is assumed. For the flexible link option, the user shall specify the values for F_s and K_s. Additionally, the user can specify if the F_s force varies at different floor. In case that it does, the F_s force coefficients need to be specified in the pre-formatted EXCEL as discussed in the previous section.

MF Beam Mid-Span Condition: specify whether or not the mid-span point (at the mid-span gusset plate connection) is allowed to sag following brace buckling.

(a)   (b)

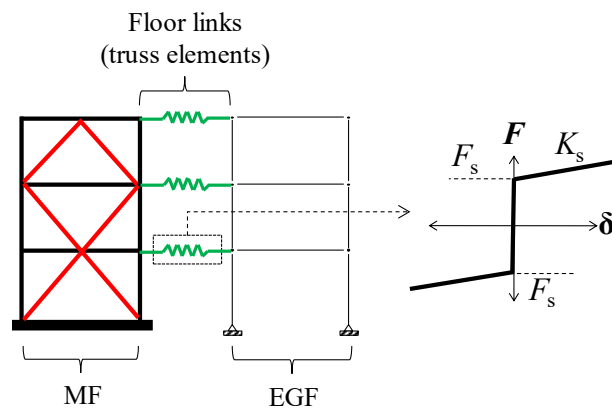Figure 4.9. Building parameters module: Supports and connections tab for (a) MRF and (b) CBF

Figure 4.10. Floor truss links; MF-to-EGF connection

4.3.3   Loads & Material tab

Loads: specify the uniform area loads for the pre-defined load cases: Dead load (DL), Live load (LL), Generic load (GL) and Cladding load (CL). For DL, LL and GL, loads can be specified with respect to the typical floor and roof levels.

Load Combination Coefficients: specify the coefficients of the load combinations that will be used to compute the seismic weights, W_s, and the building mass, M. In particular, for a specific floor level, W_s and M are computed as follows,

$$W_s = C_{w,DL} \cdot DL + C_{w,CL} \cdot CL + C_{w,LL} \cdot LL + C_{w,GL} \cdot GL \tag{4.1}$$

$$M = C_{m,DL} \cdot DL + C_{m,CL} \cdot CL + C_{m,LL} \cdot LL + C_{m,GL} \cdot GL \tag{4.2}$$

In which, C_w and C_m parameters are the loading combination coefficients for the seismic weight and mass computation, respectively. In the 2D model, the gravity loads and masses are computed and applied at nodal point at each floor level as demonstrated in Figure 4.12. In particular, the MF's columns are assigned loads and masses corresponding to their tributary area; L_t and m_t, respectively which are computed as follows:

$$L_t = A_{trib,Column} \cdot W_s \tag{4.3}$$

$$m_t = A_{trib,Column} \cdot M \tag{4.4}$$

Note here that $A_{trib,Column}$ refers to the MF column's tributary "floor" area for DL, LL and GL while it refers to the tributary "façade" area for CL, as demonstrated in Figure 4.5. The rest of the building load and mass (L_g and m_g, respectively) is assigned to the EGF grid nodes as demonstrated in Figure 4.12. These are computed as follows:

$$L_g = (A_{trib,MF} - A_{trib,Column}) \cdot W_s \tag{4.5}$$

$$m_g = (A_{trib,MF} - A_{trib,Column}) \cdot M \tag{4.6}$$

Where, $A_{trib,MF}$ refers to the "floor" or "façade" area tributary to one MF in the considered direction. For the example building shown in Figure 4.12, this tributary floor area would be equal to half of the floor area (that is the total floor area divided by two MFs that are available in the considered direction)

Figure 4.11. Building parameters module: Loads and material tab
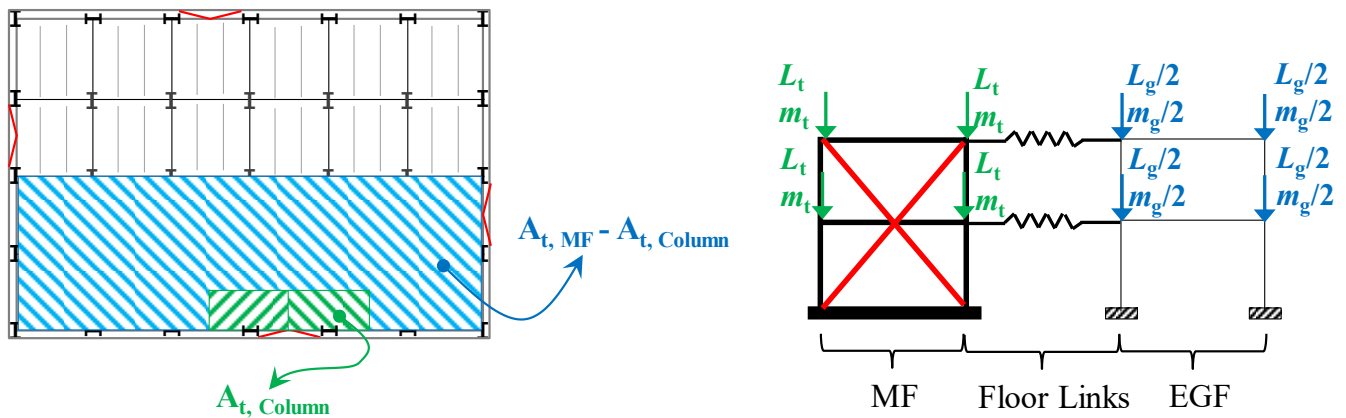

Figure 4.12. Illustration of mass and gravity load computation and allocation in the numerical model

Material: specify the material properties in the same unit as that of the project. This includes the modulus of elasticity (E), the yield stress (f_y) and Poisson's ratio (μ). Using the dropdown menu, the user can select a steel type/grade from an integrated material library. This material library and associated properties is based on extensive material calibrations done by de Castro e Sousa and Lignos (2018); de Castro e Sousa et al. (2020) against the Voce-Chaboche constitutive plasticity model (von Mises 1913; Voce 1948; Chaboche 1989). Check the following OpenSEES wiki link for more details: (https://opensees.berkeley.edu/wiki/index.php/UVCuniaxial_(Updated_Voce-Chaboche)).

4.3.4 Member Modeling tab

The "Member Modeling" tab for the MRF and CBF options is shown in Figure 4.13a and b, respectively. Modeling parameters to be defined are as follows:

MF Column Elements: specify an option for modeling the elastic MF column elements. Three options are available: 1) Elastic Beam-Column where an *elasticBeamColumn* element type is used (which considers only flexural deformations), 2) Fiber D-based where a displacement-based beam-column (*dispBeamColumn*) is used with a fiber-based cross-section and 3) Fiber F-based where a force-based beam-column (*forceBeamColumn*) is used with a fiber-based cross-section.

Fiber-Based Elements: specify the modeling details for fiber-based elements, including: 1) the number of segments along the element length, 2) the number of integration points per segment, and 3) the global mid-length geometric imperfection (i.e, amplitude of sin wave-shaped imperfection) as a multiple of the element length.

MF Column P-M Interaction: specify the method that will be used to obtain the axial load level P that will in turn be used to reduce the main frame's columns' plastic flexural strength, M_y. Two methods are available:

1) Compute P based on the gravity load (i.e., the seismic weight) multiplied by an optional scale factor. In particular, for the column at story i in an n-story building, P is computed using Equation 4.7 in which P_t is the sum of column tributary loads in the floors above the column.

$$P = \text{scale factor} \cdot P_t = \text{scale factor} \cdot \sum_{floor=i+1}^{floor=n+1} L_t \tag{4.7}$$

2) Compute P based on the maximum compressive axial load level to be experienced by the column during a pushover analysis, $P_{PO,max}$, as follows:

$$P = P_t + (P_{PO,max} - P_t)/2 \tag{4.8}$$

Note that method #2 will require FM-2D to run a preliminary Pushover analysis prior to the main analysis.
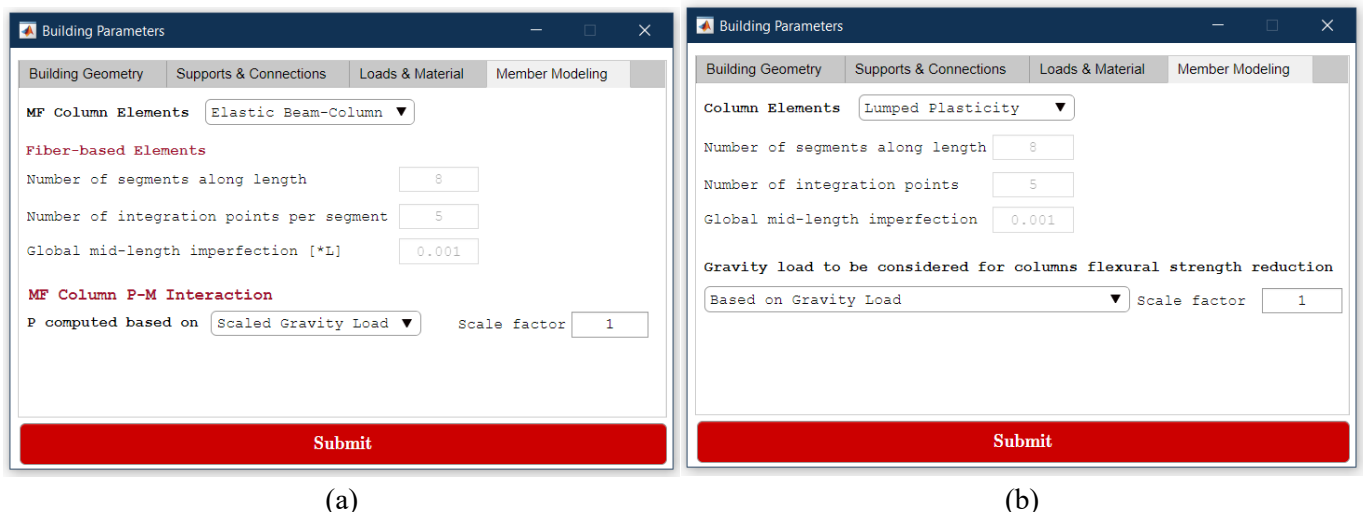

| (a) | (b) |

Figure 4.13. Building parameters module: Member modeling tab for (a) MRF and (b) CBF

4.4 Analysis Parameters

The "Analysis Parameters" module is shown in Figure 4.14. This module contains six tabs as follow:

4.4.1 Run Options tab

Analysis Type: In FM-2D v1.2010, a total of six analysis types are available:

Eigenvalue Analysis: Eigenvalue (modal) analysis to obtain the fundamental periods of the building and the corresponding mode shapes.

Static Pushover: Static linear/nonlinear pushover analysis using a pre-specified mode pattern.

IDA: Incremental dynamic analysis (IDA) up to collapse (Vamvatsikos and Cornell 2002).

Dynamic: Target Scale Factor: Linear/Nonlinear response-history analysis by scaling a set of ground motion records using a pre-specified constant scale factor.

Dynamic: Target Intensity: Linear/Nonlinear response-history analysis by scaling a set of ground motion records to a pre-specified target seismic intensity at a pre-specified period.

ELF: Linear/Nonlinear static analysis using a pre-specified equivalent lateral force profile.

Show Animation: select this option to display the structural deformation profile during analysis. If selected, the user has the option to specify the deformation profile scale factor as well as the size of the animation window in pixels. It is not recommended to use this option when running IDA for large buildings, as it can increase the run time.

Show OpenSEES Status: select this option to display OpenSEES feedback during analysis. This will be displayed within the Windows Command Shell that will open when FM-2D is running. This option can be very useful in tracking possible problems with the *tcl* code.

Show Scope: select this option to display a live plot of the maximum SDR and PFA recorded during the analysis.

Maximum Run Time: specify the maximum run time for a pushover analysis or a single ground motion record analysis. This option is useful for terminating infinitely-iterative loops that may occur during simulations due to convergence problems. A value of 5 to 10 minutes is recommended in most cases.

Include Modeling Uncertainty: in the case of response-history analysis, select this option if you need to include the numerical modeling uncertainty to your analysis. In that case, you need to specify the logarithmic standard deviation associated with the different numerical modeling parameters such as those of the employed deterioration component models or the material and damping parameters, as shown in Figure 4.15. Note that the current version 1.2010 assumes that all parameters are log-normally distributed and uncorrelated. As part of the uncertainty consideration, a Monte Carlo scheme is employed where a number of numerical model realizations are generated and analyzed for each ground motion record.
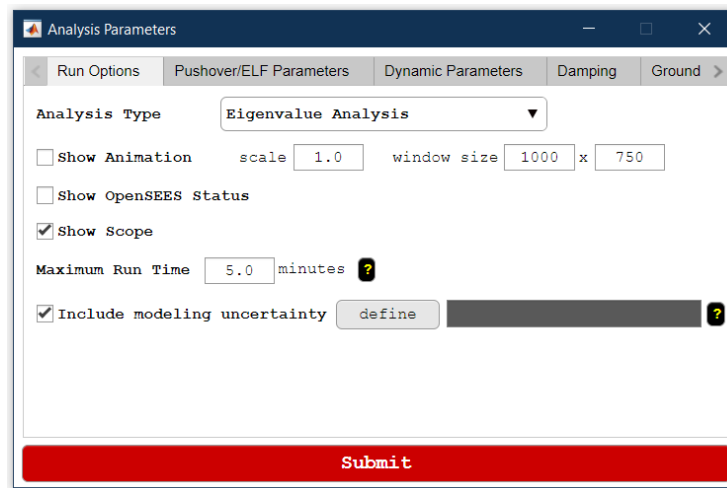
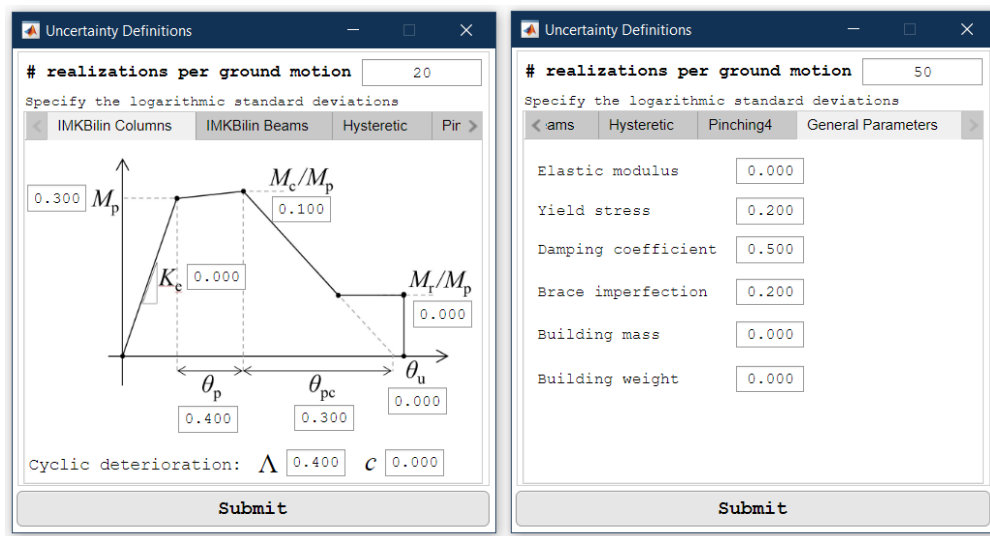Figure 4.14. Analysis parameters module: Run options tab

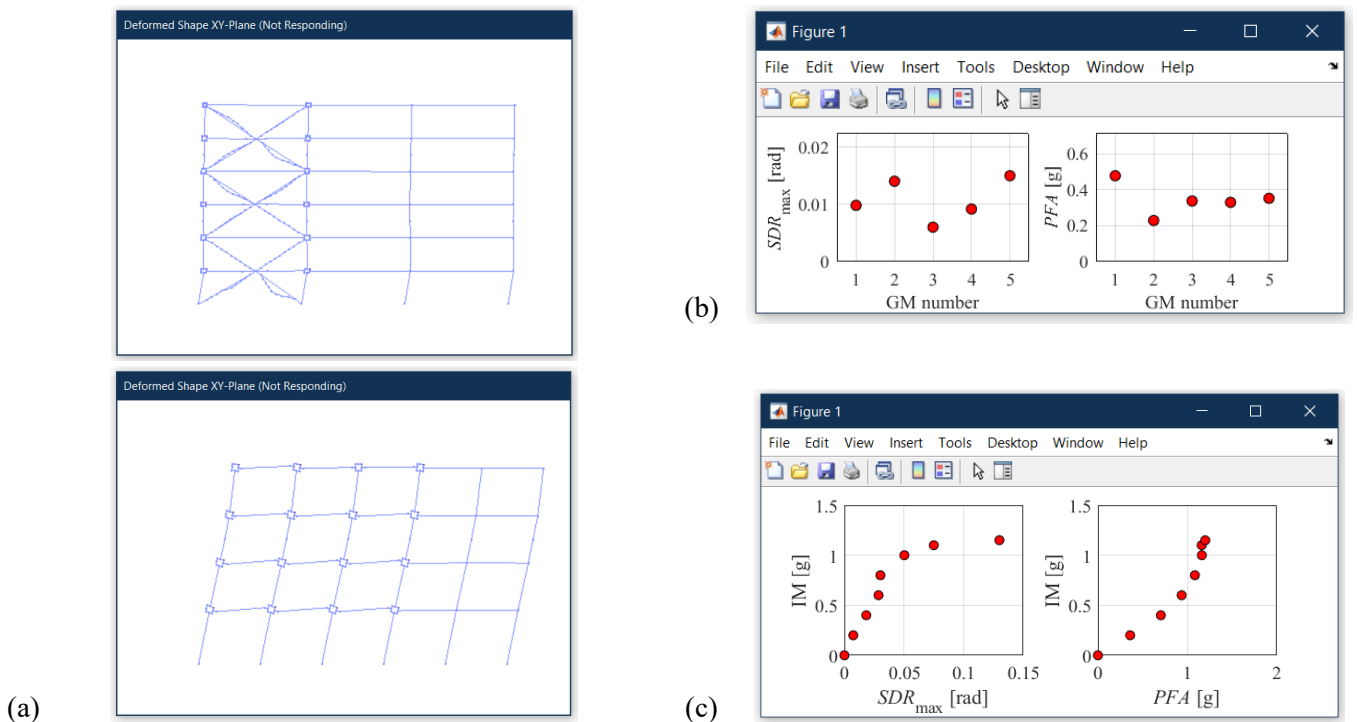

Figure 4.15. Module for modeling uncertainty definition

Figure 4.16. (a) Deformed shape window; (b) scope figure for dynamic analysis; (c) scope figure for IDA

4.4.2 Pushover/ELF Parameters tab

Pushover Pattern based on Mode: specify the Eigen mode number for which the modal shape will be used as a displacement pattern for pushover analysis.

Target Roof Drift Level: specify the target drift of the roof (top floor) as a percentage of the total building height.

ELF Profile: browse for a *.txt* file containing a single column of data representing the values of the equivalent lateral forces to be applied at each floor. Note that for a building with n number of stories, the n number of forces need to be specified in the text file representing all building floors except for the ground floor.
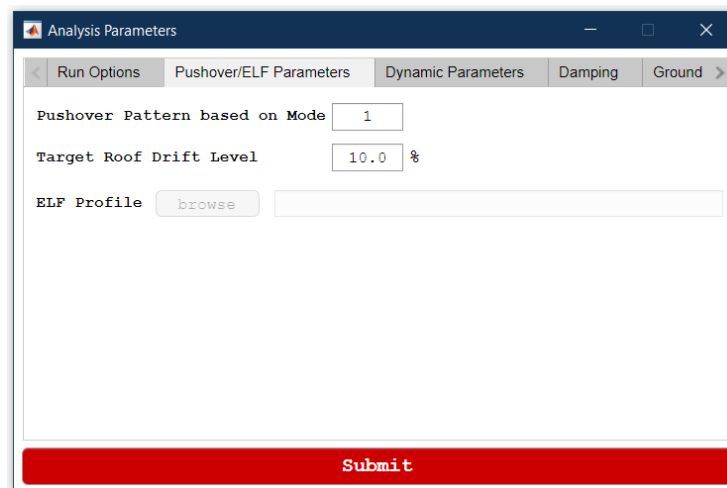

Figure 4.17. Analysis parameters module: Pushover/ELF parameters tab

4.4.3 Dynamic Parameters tab

Intensity Measure: for any type of dynamic analysis, select the intensity measure type to be used in ground motion scaling. Two types of acceleration-based intensity measures are available in the current release: 1) the spectral acceleration at the building first-node period, $Sa(T_1)$ and 2) the spectral acceleration ordinates average (geometric mean) within a given range of periods Sa_{avg} (Eads et al. 2015).

Scaling Period: specify the reference period that will be used to scale ground motions records.

Scaling Factor: specify the constant scaling factor that will be used to scale ground motions records.

Target Intensity: specify the target seismic intensity, in units of g, for which the ground motions records' intensity measure will be scaled to.
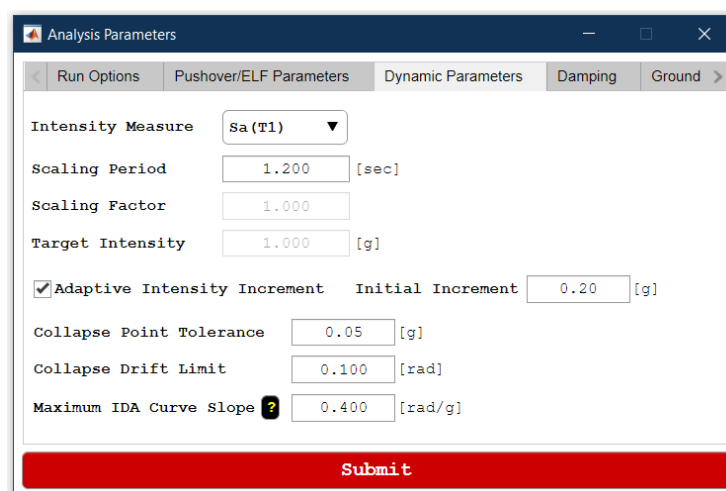
Adaptive Intensity Increment: for IDA, select this option to let FM-2D change the seismic intensity increment (step) depending on how close the analysis is to the collapse point. This option can help save run time for IDA analysis compared to using a constant increment.

Initial/Constant Increment: for IDA, specify the initial intensity increment if adaptive intensity increment is selected. If not, then specify the constant intensity increment.

Collapse Point Tolerance: for IDA, specify the tolerance (accuracy) to be used when tracking the collapse point; that is the error for the identified collapse intensity will be within +/- the tolerance.

Collapse Drift Limit: for dynamic analysis, specify an additional collapse-identification criteria based on the story-drift ratio limit as which collapse can be assumed to have occurred. In general, deteriorating systems will collapse under increasing P-Delta forces before reaching this SDR limit.

Maximum IDA Curve Slope: for dynamic analysis, specify an additional collapse-identification criteria based on the maximum slope of the SDR-versus-Intensity IDA curve, beyond which collapse can be assumed to have occurred. A value of 0.4rad/g is recommended (corresponding to an increase of 0.04 radians per an intensity increment of 0.1g).

Figure 4.18. Analysis parameters module: Dynamic parameters tab

4.4.4 Damping tab

Raleigh Periods: for dynamic analysis, specify the periods used in the Raleigh damping model.

Damping Coefficient: for dynamic analysis, specify the damping coefficient as a percentage.

Figure 4.19. Analysis parameters module: Damping tab

4.4.5 Ground Motions tab

Ground Motion Folder: for dynamic analysis, browse for the folder containing the ground motion records. This folder should include individual ASCII text files for ground motion record. The text file should include the acceleration data in units of *g*. The data can be organized in a single or multiple column format. The folder should also include a file named *AllGMinfo.txt*. This file should include two columns of data: the record name (also the acceleration file name) and the time step. Note that once the ground motion folder is selected, the ground motion data will be processed and stored internally in the project's MATLAB file. In that sense, the GM folder need not be present when the project file is transferred to another machine.

Consider GM Records: for dynamic analysis, specify the range of records to be considered. This option is meant to allow the user to run a subset of the imported records.

Free Vibration Time: for dynamic analysis, specify the amount of free vibration time to be added at the end of each ground motion record. A value of 10 seconds is typically used to allow the structure to come to rest and to trace residual drifts.

Analysis Time Step: for dynamic analysis, specify the size of the time step for the transient analysis as a function of the ground motion record time step. A value of 0.25 to 0.5 is commonly used.
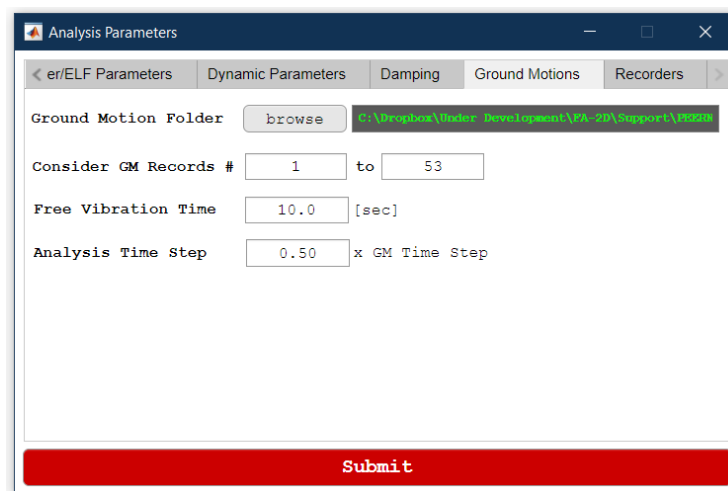

Figure 4.20. Analysis parameters module: Ground motions tab

4.4.6 Recorders tab

From this tab, the user can select the local and global EDPs to be saved. By default, the SDR, PFA and Eigen modes are saved. Using the checkboxes shown in Figure 4.21, other recorders can be activated. The description of each EDP and the name notation of the generated output files are summarized in Table 4.1.

Table 4.1. Recorder options and their definitions

Recorder Option	Description	Output Filename*
Time	Analysis time vector	Time.out
Floor Displacement	Floor absolute displacement	Disp[F]_MF.out
Story Drift Ratio	Story drift ratio	SDR[S]_MF.out
Floor Acceleration	Relative floor acceleration	RFA[F]_MF.out
Floor Velocity	Relative floor velocity	RFV[F]_MF.out
Column Element Forces	Axial, shear and moment forces for elastic column elements	Column[S][A].out
Beam Element Forces	Axial, shear and moment forces for elastic beam elements	Beam[F][B].out
Column Spring M/R	Moment and rotation of the column springs	ColSpring[S][A][L]_F.out ColSpring[S][A][L]_D.out
Beam Spring M/R	Moment and rotation of the beam springs	BeamSpring[F][B][L]_F.out BeamSpring[F][B][L]_D.out
Panel Zone Spring M/R	Moment and rotation of the panel zone springs	PZ[F][A]_M.out PZ[F][A]_R.out
Brace F/D	Axial force and deformation of the brace elements	Brace[S][B]_F.out Brace[S][B]_D.out
Floor Link F/D	Axial force and deformation of the floor links truss elements	FloorLink[F]_F.out FloorLink[F]_D.out
Support Reaction	Reaction forces at ground supports	Support[A].out
Eigen Modes	Eigen mode periods	EigenPeriod.out
Eigen Vectors	Eigen mode vectors	EigenVectorsMode[M].out

* [F] Floor number   [S] Story number   [B] Bay number   [A] Axis number   [M] Mode number
   [L] Spring location: For Beams: Top (T) or Bottom (B) spring within a story
                         For columns: Left (L) or   Right (R) spring within a bay
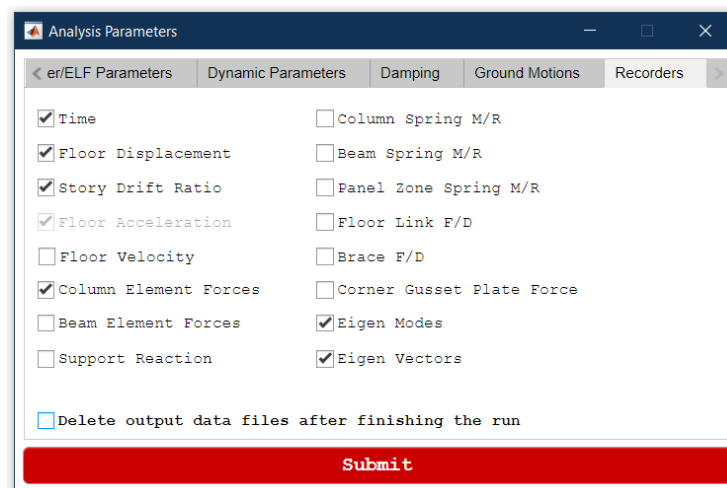
Figure 4.21. Analysis parameters module: Recorders tab

5. Visualization Options

The visualization module, shown in Figure 5.1, provides the user with a variety of options to visualize the analysis results. The visualizations options differ based on the frame type, the analysis procedure and the recorded EDPs. The plot options are described below.
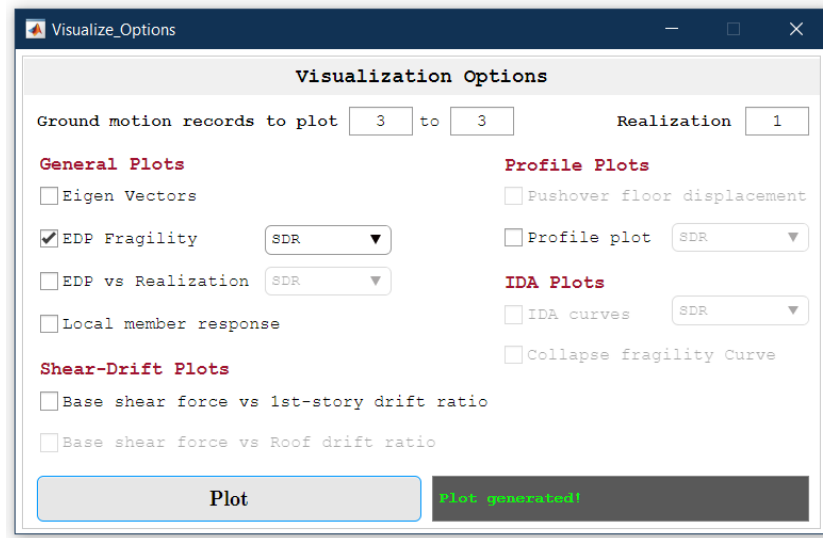
Figure 5.1. Visualizations options

Eigen Vectors: Plots the profile of the Eigen vector for the different mode periods as shown in Figure 5.2.
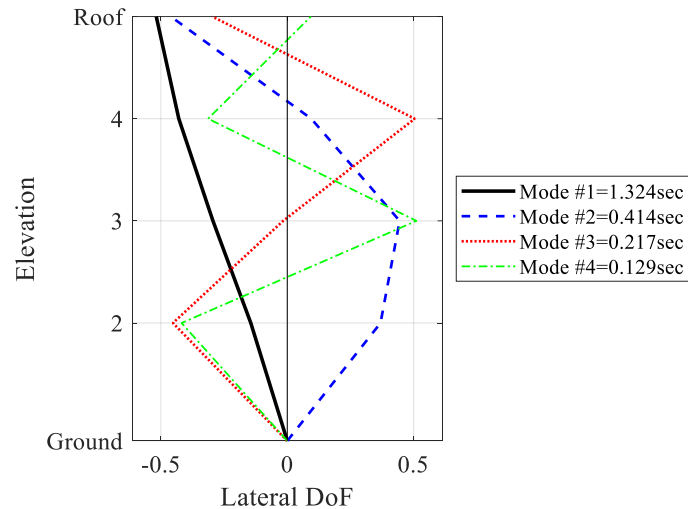
Figure 5.2. Eigen vectors plot

Shear-Drift Response: Plots the base shear force, normalized by the seismic weight, versus the first-story drift ratio or the roof drift ratio. Figure 5.3a and Figure 5.3b show a sample plots of the base shear-drift response based on dynamic and pushover analysis, respectively. Note that in the case of dynamic analysis, the user can specify, from the top of the visualization module, the range of ground motion records to be processed and plotted.

(a)   (b)
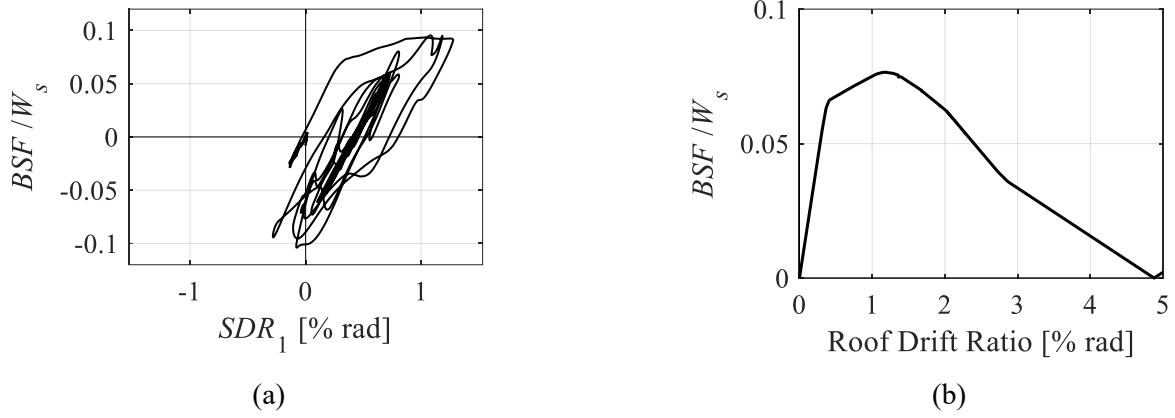
Figure 5.3. Normalized base shear force plots: (a) versus first-story drift ratio based on dynamic analysis; (b) versus roof drift ratio based on pushover analysis

IDA Plots: In the case of IDA, two additional plots can be generated. The first plot is the IDA curves; which is a plot of the seismic intensity versus a specific EDP as shown in Figure 5.4a for the *SDR*. IDA curves are generated for *SDR*, *RDR*, *PFA* and *PFV*, unless the recorder was not activated for any of those EDPs. The second plot is the collapse fragility curve as shown in Figure 5.4b. This curve is a lognormal cumulative distribution function (CDF) that is fitted to the empirical probability distribution of the identified collapse intensities for the different ground motion records.

(a)   (b)
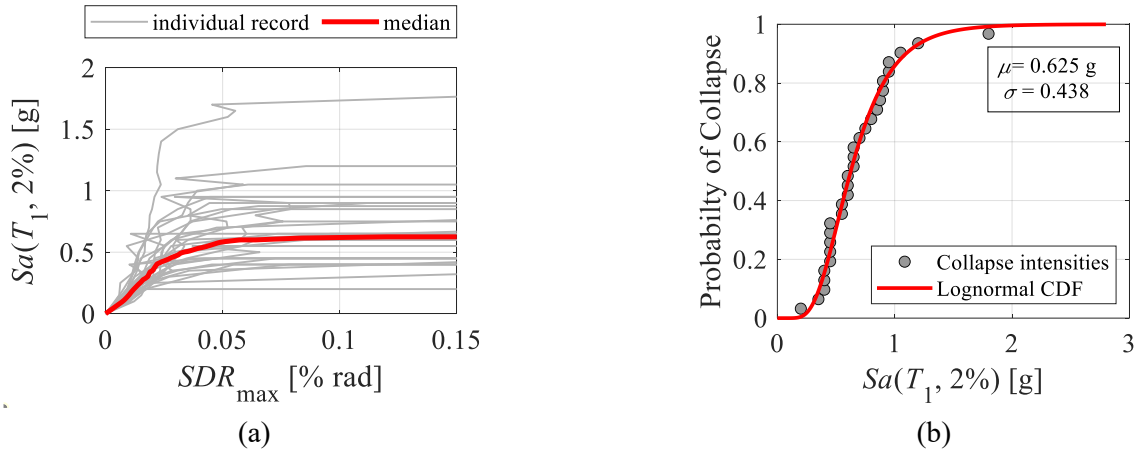
Figure 5.4. IDA plots: (a) IDA curve for maximum *SDR*; (b) collapse fragility curve

EDP Profiles: Plots the profile of the different EDPs along the building height. Figure 5.5 shows the different profile plots generated by FM-2D for *SDR*, *RDR*, *PFA*, *PFV* and Displacement profile. As shown in these figures, in the case of multiple ground motion records (more than 3), the median profile is computed and superimposed.

Local Member Response: This will open an interactive interface as shown in Figure 5.6. From this interface, the user can explore the responses of the frame elastic elements and nonlinear springs that were selected to be recorded.
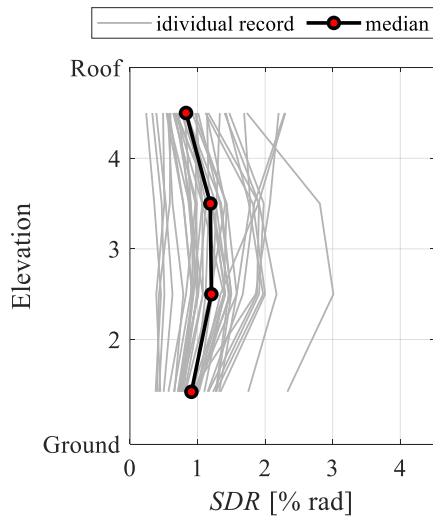
(a) Story-drift ratio
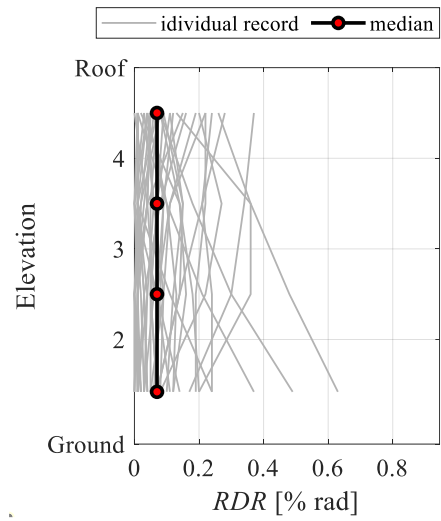
(b) Residual story-drift ratio
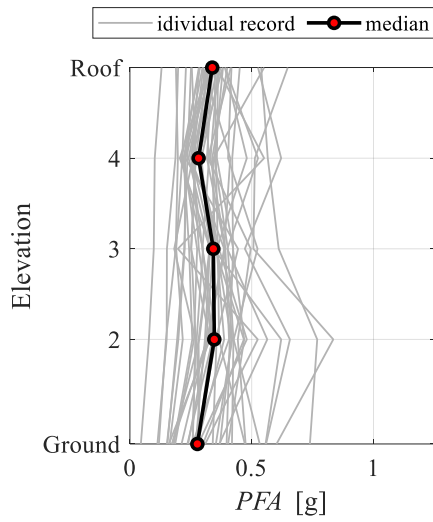
(c) Peak floor accelerations
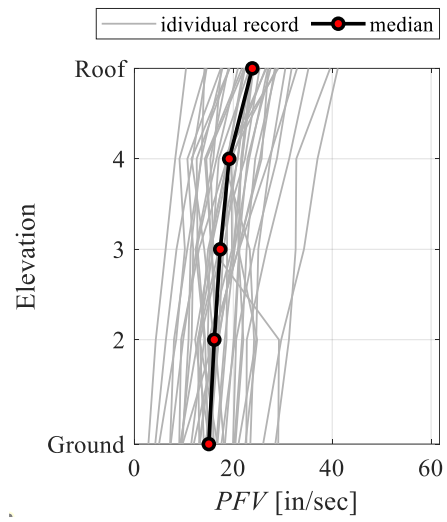
(d) Peak floor velocity
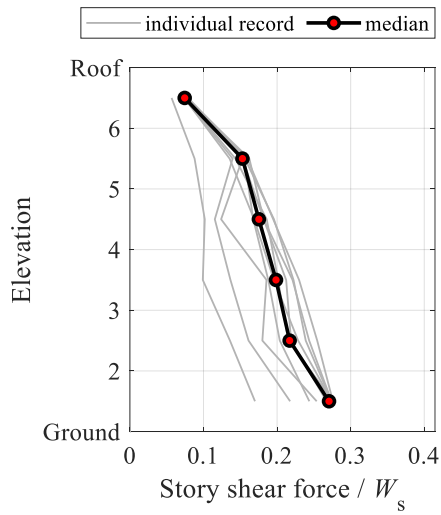
(e) Story shear force
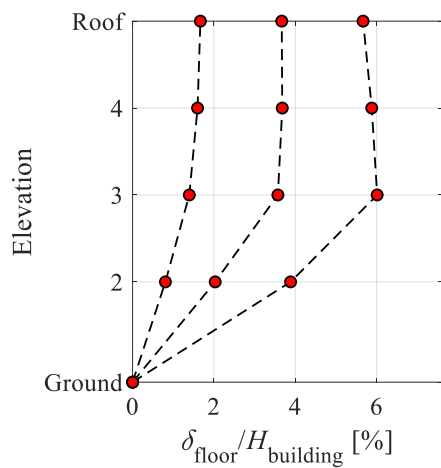
(f) Displacement profile for Pushover/ELF analysis

Figure 5.5. EDP profile plots
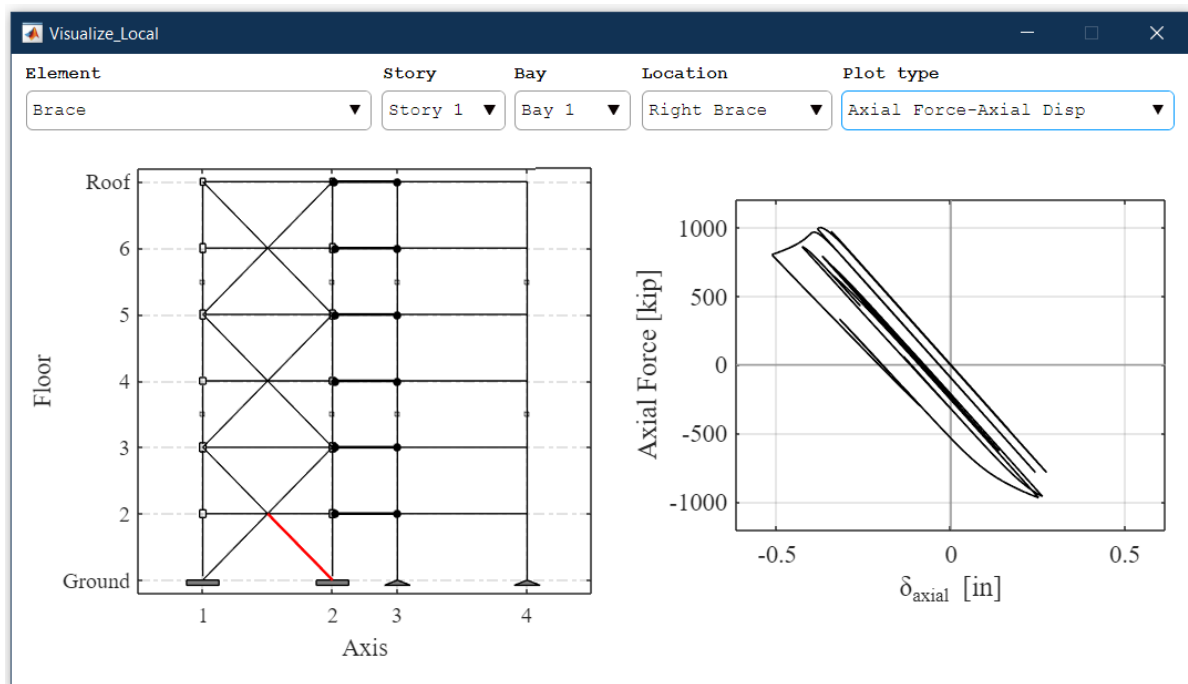
Figure 5.6. Interactive interface for visualizing local member response

6. Project Inspector

The user can verify the defined project components and parameters by using the "scope" button located in the project file management panel (see Figure 4.1). This button will open a window that provides a summary of the current project definition as shown in Figure 6.1.

Figure 6.1. Project summary window

7. Modeling Methodology

FM-2D employs state-of-the-art numerical modeling recommendations when constructing the numerical models. A brief description of the main modeling assumptions are provided below.

7.1 Main Frame Steel Columns

The main frame columns are modeled as elastic elements with nonlinear rotational springs at their end (i.e., lumped plasticity approach). The springs are assigned the modified Ibarra-Medina-Krawinkler (IMK) phenomenological deterioration model (Ibarra et al. 2005; Lignos and Krawinkler 2011). In particular, the recently updated *IMKbilin* model is used (refer to https://github.com/amaelkady/Modified-IMK-Models-for-OpenSEES). The moment and rotation parameters of this model's characteristic monotonic backbone curve (see Figure 7.1a) as well as the cyclic deterioration parameters, are defined based on the latest recommendations by Lignos et al. (2019).

7.2 Steel Moment Beam-to-Column Connections

Similar to the columns, the flexibility of the main frame beam-to-column moment connections are modeled as nonlinear rotational springs. The springs are assigned the modified Ibarra-Medina-Krawinkler (IMK) phenomenological deterioration model (Ibarra et al. 2005; Lignos and Krawinkler 2011). In particular, the recently updated *IMKbilin* model is used (refer to https://github.com/amaelkady/Modified-IMK-Models-for-OpenSEES). The moment and rotation parameters of this model's characteristic monotonic backbone curve (see Figure 7.1a) as well as the cyclic deterioration parameters, are defined based on the recommendations by Lignos and Krawinkler (2011); Lignos et al. (2019).

7.3 RC Beams and Columns

RC beams and columns are idealized as linear elastic beam-column elements with zero-length nonlinear rotational springs at the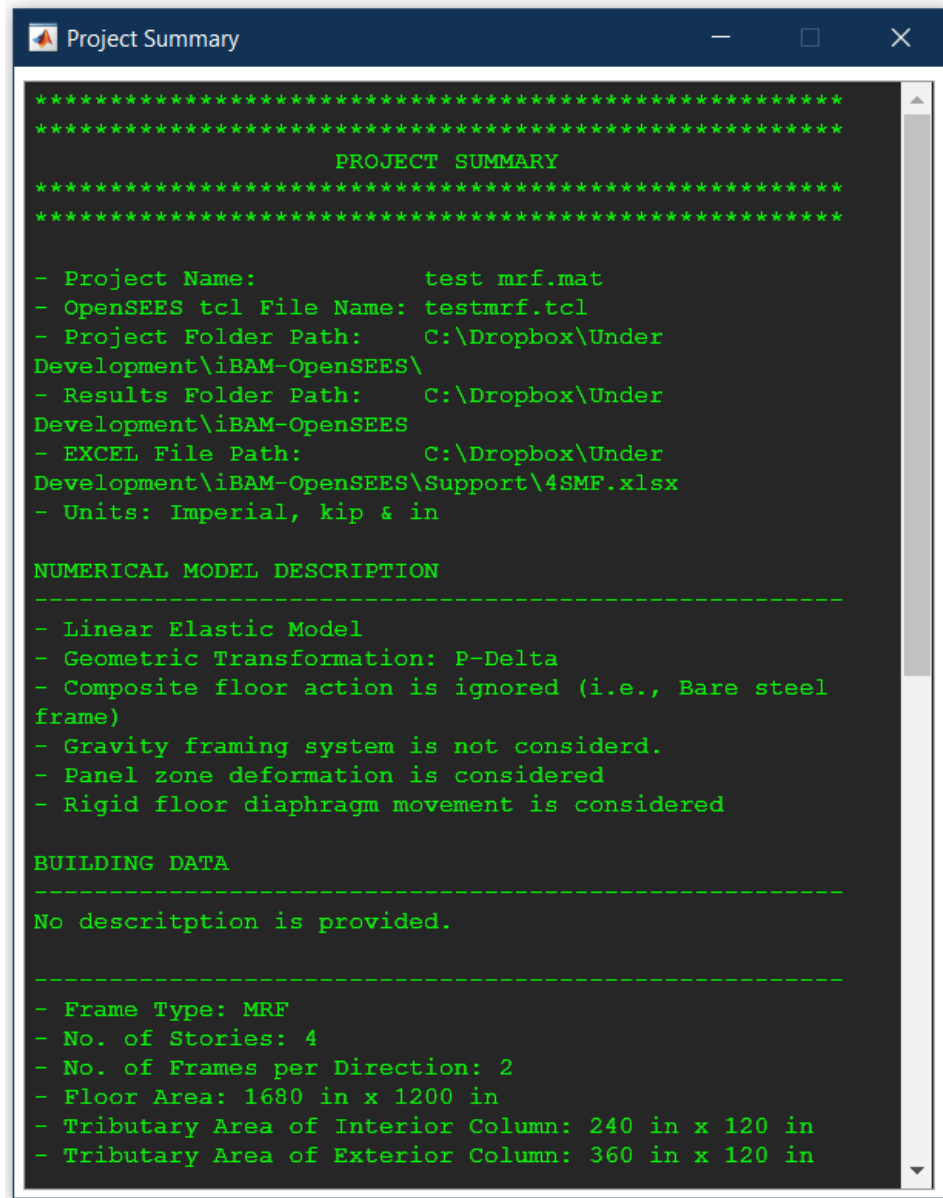ir ends. The elastic elements can be modeled as either fiber-based elements or elastic beam-columns. The rotational springs are assigned the modified Ibarra-Medina-Krawinkler (IMK) deterioration model with a peak-oriented hysteretic response (Ibarra et al. 2005). The parameters of the backbone curve of this model are computed as per the empirical regression equations developed by Haselton et al. (2008). Note that these models do not account for undesirable shear failure that is expected in older non-ductile RC buildings.

7.4 Steel Column Panel Zone

If the column panel zone deformation is considered (see Section 4.2.1: Modeling Options), is explicitly modeled using the "parallelogram" model as discussed in Gupta and Krawinkler (1999) as shown in Figure 7.2a. A rotational spring allocated at the corner of the idealized panel zone is used to capture the moment-shear distortion response. The rotational spring is assigned the *Hysteretic* material model (https://opensees.berkeley.edu/wiki/index.php/Hysteretic_Material). Referring to Figure 7.1b, the three points defining this model backbone curve (that is the panel zone stiffness, strength and shear distortion parameters) are

defined as per the latest recommendations by Skiadopoulos et al. (2020) that were developed as an update to the historical Krawinkler (1978) model. If the composite action is considered (see Section 4.2.1: Modeling Options), the bare steel strength parameters of the panel zone are modified as per Elkady and Lignos (2014).

7.5   Braces and Gusset Plates

The steel brace model is based on the force formulation proposed by Spacone et al. (1996). The brace component is divided into two elements that are offset (initial camber) in the mid-length of the brace to trigger global (flexural) buckling. A displacement-based formulation is used where the brace is discretized by *n* segments (8 segments by default). The displacement-based beam-column element is assigned a corotational transformation to account for large displacements due to global buckling. A fiber-based approach is employed to represent the brace cross-section. The cross-section fiber elements are assigned a uniaxial engineering stress-strain relationship that is based on the Menegotto-Pinto *Steel02* model (Menegotto and Pinto 1973). Depending on the cross-section type (rectangular or circular Hollow Structural Section (HSS) or wide-flange) a different discretization process and set of material parameters are employed based on the recommendations by Karamanci and Lignos (2014) based on extensive calibrations with experimental data. Fracture resulting from low-cycle fatigue is simulated by masking the *Steel02* material with the *Fatigue* material model (Uriz et al. 2008). This material model is based on a linear strain accumulation rule in accordance with the Coffin-Manson relationship (Manson 1965). In summary, this modeling approach explicitly simulates the post-buckling brace behavior and the interaction between the second-order bending moment and the axial force, while capturing the spread of plasticity along the brace depends on the steel brace cross-section and the onset of ductile fracture.

The flexibility and flexural yielding of the gusset plates due to out-of-plane brace bending are explicitly simulated using a nonlinear out-of-plane rotational spring placed at the ends of each brace as proposed by Hsiao et al. (2013).

7.6   Gravity Framing

If the gravity framing system is considered (see Section 4.2.1: Modeling Options), the equivalent gravity frame is used to represent it as discussed in Gupta and Krawinkler (1999). In particular, at each story, the flexural strength and stiffness properties of the columns as part of the equivalent gravity frame are set equal to half of the summation of those of the interior gravity framing. Similarly, the equivalent gravity beams per floor are assigned a flexural strength and stiffness equal to half of the summation of those of the gravity beams in the considered orthogonal direction.

The gravity beam-to-column connections are assumed as conventional shear-tab connections (a single steel plate fillet welded to the supporting column with a single column of structural bolts). A rotational spring is used to simulate the moment-rotation response of these connections. The spring is assigned the *Pinching4* deterioration model which simulates a deteriorating pinched load-deformation response. The model's backbone curve is defined based on four points as shown in Figure 7.1c. The four points (i.e., moment and rotation parameters) for bare steel and composite shear-tab connections are defined based on the recommendation by Elkady and Lignos (2015). The Pinching4 model is masked with a *MinMax* model to superimpose the ultimate rotations at failure (assumed to be 8% and 12% radians

for bare and composite connections, respectively). Furthermore, the user has the option to consider the sudden increase in the connection's stiffness and strength due to beam binding on the column flange when the gap between the two closes (refer to Elkady and Lignos (2015)). This will involve adding another spring in parallel which is assigned the elastic perfectly-plastic gap model, *ElasticPPGap*.

7.7 Damping

Damping is simulated with the Rayleigh model. The damping coefficient, ζ, is assigned to two mode periods; all of which are defined by the user (see Section 4.4.4: Damping tab). The Rayleigh damping stiffness-proportional term is assigned only to the elastic column and beam elements while the mass-proportional term is assigned to all the frame nodes with masses as suggested by Zareian and Medina (2010). The proposed method of assigning damping to the frame model helps avoiding fictitious damping forces when the frame behaves in the nonlinear range.

(a) *IMKBilin* model   (b) *Hysteretic* model   (c) *Pinching4* model
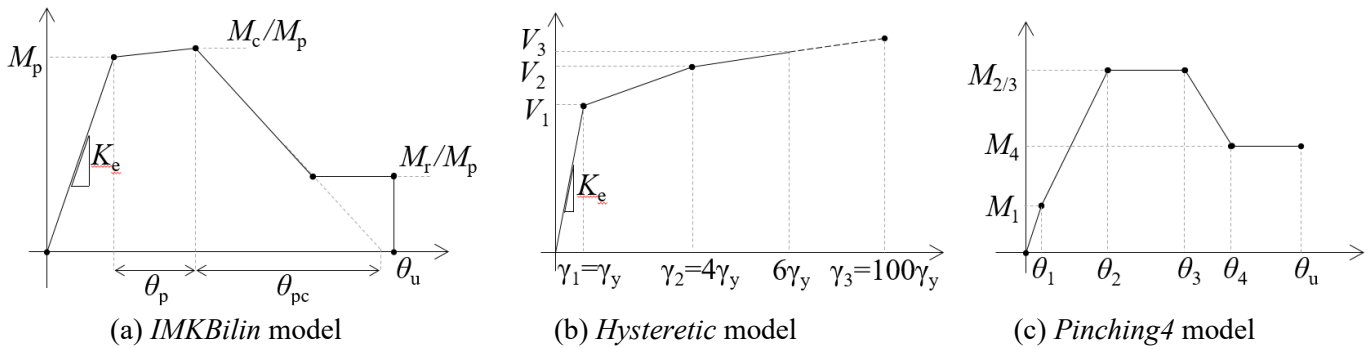
Figure 7.1. The backbone curve parameters for the main deterioration models used in FM-2D

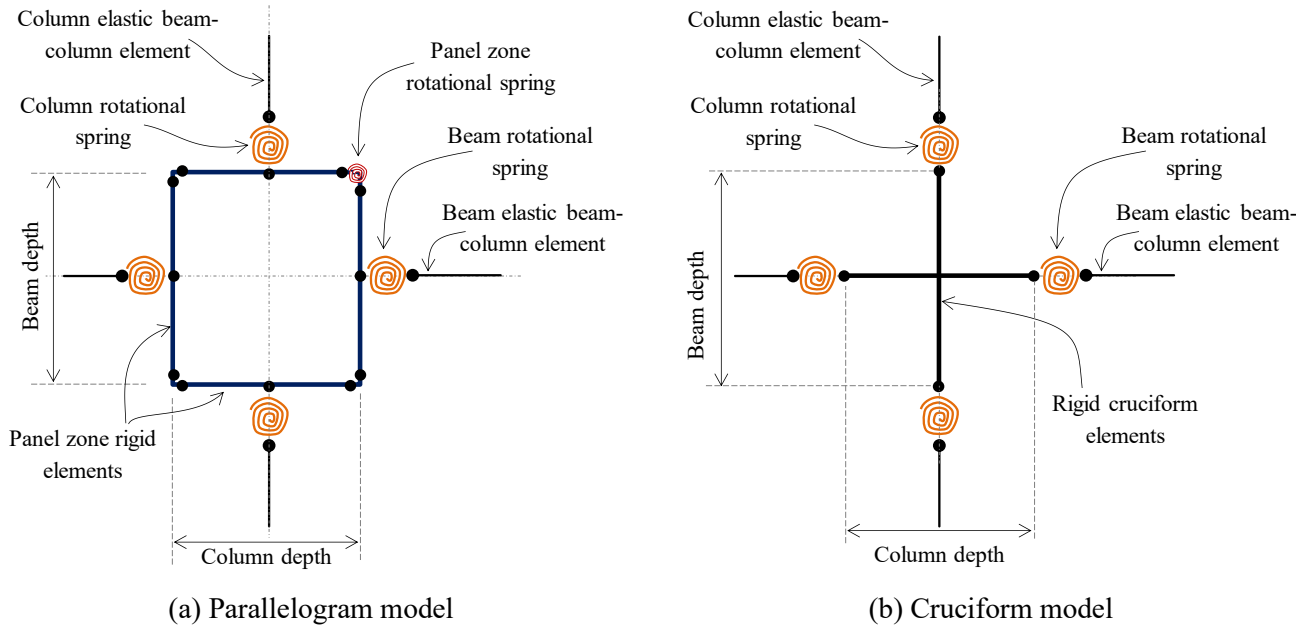(a) Parallelogram model   (b) Cruciform model

Figure 7.2. Types of panel zone idealization when (a) considered and (b) ignored

8. For Developers

FM-2D is fully developed in MATLAB. The entire source code is a combination of MATLAB's *.mlapp* graphical user interface (GUI) files, *.m* script files and *.mat* data files as well as OpenSEES *.tcl* subroutine files. The source code is available in the online GitHub repository inside the folder named *src*. Users who are interested in further developing FM-2D are able to do so through the GitHub platform. In brief, the user should 1) Create an account on GitHub.com, 2) Download and install the GituHub Desktop application, 3) Fork FM-2D's repository (i.e., take a parallel copy of FM-2D's code on the user's local computer) and 4) finally, further developing and expanding the source code. Proposed changes to the code should then be publically shared with other users, under the GPLv3 license. The user can "commit" any changes to the online repository where they will be vetted and validated before being patched to the master code. Credit will be given to any user contributions in newly compiled executables. Consult the GitHub quick start guide here: https://help.github.com/en/github/getting-started-with-github/quickstart.

To help users understand the source code, the MATLAB script code within the *.m* function files is well commented and organized. Furthermore, by clicking the "developers" button in FM-2D's main console, the user is prompted to the interface shown in Figure 8.1. . This interface comprises two tabs:

(a) Code Hierarchy tab, see Figure 9.1a, shows the code's organizational tree. By expanding the branches of this tree list, the user can inspect the dependency and the relation between the different source code GUI and function files. Additionally, by clicking on a specific element (GUI or function name) within this code tree, a full description of this element is displayed in the tip window at the bottom of the interface. The description also includes the names of the output variables generated by a specific function.

(b) Variables tab, see Figure 9.1b, which lists the details of all the variables used within the source code, including their name, type (scalar, vector, matrix, structure, etc.) and the parent module(s) or function(s) where this variable is used and defined. By clicking on a given variable name, a full description of the variable will be displayed in the tip window.

These tools are aimed to help the user comprehend the source code. As such, the user is potentially able to navigate the source code and modify/add specific functions accordingly.
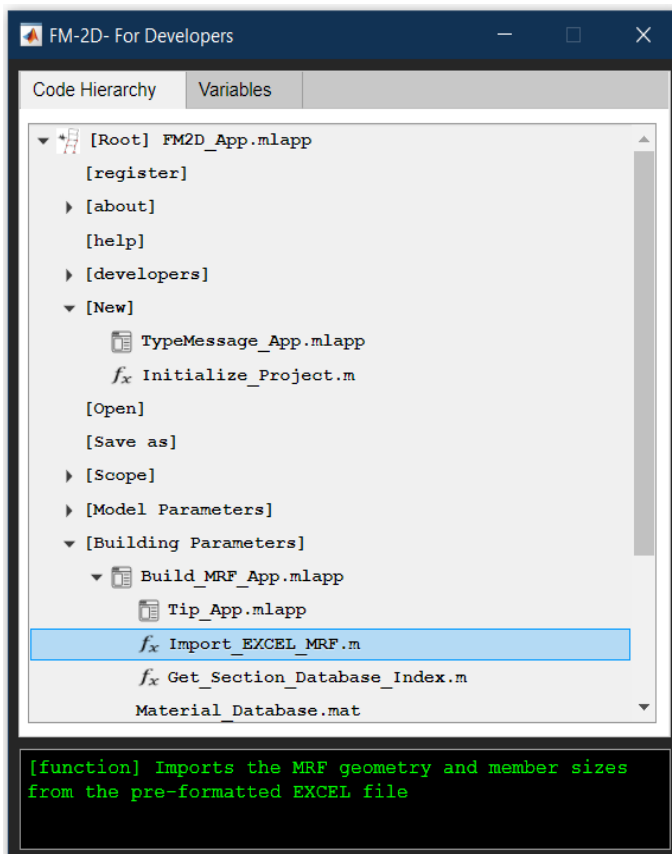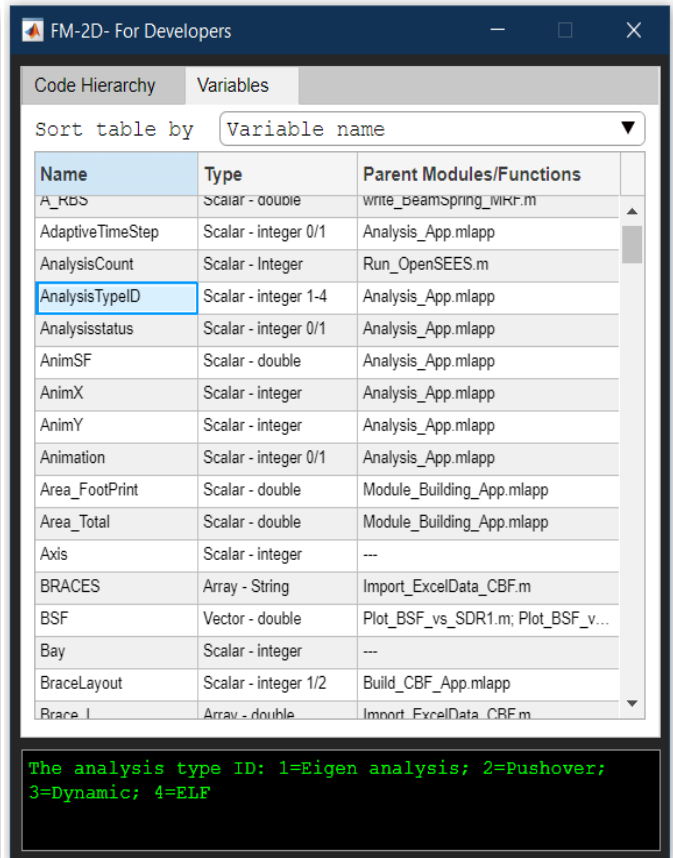
(a)　　　　　　　　　　　　　　　　　　　(b)

Figure 8.1. Developers interface for source code hierarchy and variable definitions

8.1 Source Code File Description

The source code for FM-2D is available inside the *src* folder in the GitHub repository. The source code comprises a number of files that can be categorized into four types as shown in Figure 8.2. Each of which is described below.
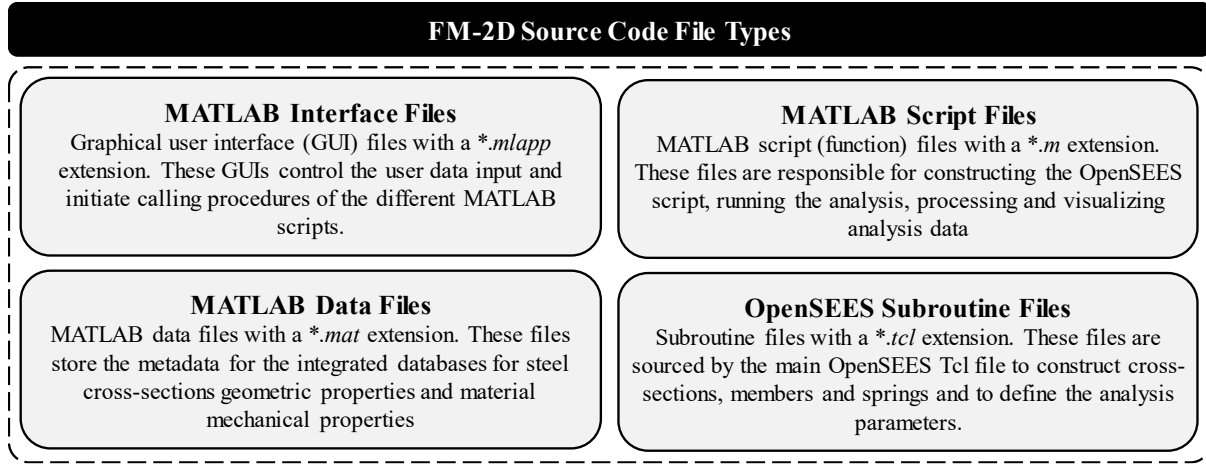
Figure 8.2. Types of source code files

8.1.1 MATLAB interface files

Those are the graphical user interface (GUI) files with a *.mlapp* file extension. These GUIs control the user data input and the subsequent calling procedures of the different subroutines and MATLAB codes. The files are:

FM2D_App.mlapp: Main console interface (see Figure 4.1).

TypeMessage_App.mlapp: Interface for selecting the frame type, the project's units and for providing an optional project description (see Figure 4.2).

Model_App.mlapp: Interface for defining the numerical model parameters such as the geometric transformation and the consideration of composite action and gravity framing system contributions.

Build_MRF_App.mlapp: Interface for defining the MRF building geometry, member sizes, loads and materials.

Build_CBF_App.mlapp: Interface for defining the CBF building geometry, member sizes, loads and materials.

Analysis_App.mlapp: Interface for defining the analysis type and its associated parameters.

ExecutionMessage_App.mlapp: Interface for defining the execution type; whether a new *tcl* file will be built for the run or if the run will be based on the existing *tcl* file.

Visulaize_App.mlapp: Interface for selecting the different visualization/plot options for the analysis results.

Visulaize_Local_App.mlapp: Interface for interactively visualizing the local response of the frame elements and springs.

Project_Summary_App.mlapp: Interface for displaying the project definitions summary when the "scope" button is clicked.

About_App.mlapp: Interface for displaying a brief description of FM-2D and the author contact information.

Tip_App.mlapp: Interface for displaying a tip window when the help "?" button is clicked anywhere in FM-2D.

8.1.2 MATLAB script files

Those are the MATLAB code files with a *.m* file extension. These file are used to build the *tcl* files, running the OpenSEES analysis, processing and visualizing analysis data. This is brief description of these script files:

Execution_Commands.m: Calls the different scripts for constructing the *tcl* files and then executes the OpenSEES analysis

CREATOR_MODEL_MRF.m: executes the commands for generating the main *tcl* file for the steel MRF numerical model, excluding the analysis type code.

CREATOR_MODEL_MRF_RC.m: executes the commands for generating the main *tcl* file for the RC MRF numerical model, excluding the analysis type code.

CREATOR_MODEL_CBF.m: executes the commands for generating the main *tcl* file for the CBF numerical model, excluding the analysis type code.

CREATOR_ANALYSIS.m: generates the code lines in the main *tcl* file for the analysis procedures.

Import_ExcelData_MRF.m: imports the MRF building geometry and member sizes from the pre-formatted EXCEL sheet.

Import_ExcelData_CBF.m: imports the CBF building geometry and member sizes from the pre-formatted EXCEL sheet.

Import_GM_Data.m: imports the ground motion records acceleration and metadata from a specified folder.

write_*.m:** these scripts write specific lines of code to the main *tcl* file.

Plot_*.m:** these scripts plot different aspects of the analysis results.

Run_OpenSEES.m: this is main (root) script for running the different types of analysis, processing the analysis data and saving summary results in text file.

Run_exe.m: runs the OpenSEES executable.

Read_Analysis_Data.m: read the dynamic analysis results from the OpenSEES output files.

Save_Increment_Data.m: save the incremental dynamic repose data in a structured variable *IncdData* such that it can be processed later on.

Save_Results_Dynamic.m: reads the results of dynamic analysis (SDR, RDR and PFA) and saves a summary of the maximum absolute quantities in text files.

Save_Results_IDA.m: reads the results of IDA (SDR, RDR and PFA) that are stored in the *IncrData* variable and saves the IDA data in text files.

Initialize_Project.m: Initialize some of the project parameters when a new project is created.

Get_Mass_and_Period.m: gets the mass, seismic weight and runs an Eigenvalue analysis to obtain the first mode period when the "Get period" button is clicked.

Process_EigenAnalysis.m: processes the OpenSEES Eigenvalue analysis output and stores a summary of the Eigen modes (periods) and vectors in a text file.

Load_SecData.m: loads the appropriate cross-section database, based on the member being modeled, and transforms the units of the metadata.

Get_Section_Database_Index.m: finds and stores the index of the main frame member cross-sections within the database.

Get_Project_Summary.m: collates the project summary text that will be shown when the "Scope" button is clicked.

Get_Col_Axial_Force.m: computes the main frame columns' compressive axial load demand based on the selected method; gravity load or pushover analysis load.

Get_IM.m: computes the value of the intensity measures; $Sa(T_1)$ or Sa_{avg}.

Get_SA_SDoF.m: computes the spectral acceleration of an elastic single degree of freedom system.

Show_Status_Dynamic.m: shows, in the Windows Command Shell, a summary of the ground motion being analyzed, the seismic intensity, scaling factor, etc during dynamic analysis.

Show_Scope_Dynamic.m: shows a live (instantly-updated) plot of the SDR and PFA output during dynamic analysis.

Show_Scope_IDA.m: shows a live (instantly-updated) plot of the SDR and PFA IDA curves during IDA.

8.1.3 MATLAB data files

Those are the MATLAB data files with a *.mat* file extension. These files store the metadata for the structural cross-section and materials properties. The files are:

AISC_WF_Sec_Database.mat: Contains a structured array with all the cross-section geometric properties based on the AISC wide-flange cross-section database. A total of 268 cross-sections are included ranging from W8 to W44. Note that data are stored in imperial units.

HSS_Sec_Database.mat: Contains a structured array with all the cross-section geometric properties based on the AISC hollow structural section cross-section database. A total of 519 circular and rectangular HSS cross-sections are included. Note that data are stored in imperial units.

EURO_I_Sec_Database.mat: Contains a structured array with all the cross-section geometric properties based on the European I-shaped cross-section database. A total of 504 cross-sections are included of type IPE, HEA, HEB, HL, HLZ, HD, UB, UBP and UC. Note that data are stored in imperial units.

Material_Database.mat: Contains a structured array with all the mechanical properties of different steel types/grades as well as the corresponding calibrated parameters of the Voce-Chaboche constitutive plasticity model (von Mises 1913; Voce 1948; Chaboche 1989). The material parameters are based on extensive calibrations conducted by de Castro e Sousa and Lignos (2018); de Castro e Sousa et al. (2020). These parameters are used in FM-2D as part of the recently developed uniaxial UVC material model in OpenSEES (Hartloper et al. 2019a; Hartloper et al. 2019b). Note that data are stored in imperial units.

8.1.4 OpenSEES subroutine files

Those are the OpenSEES subroutine (i.e., procedure) files with a *.tcl* file extension. These file are called (i.e., sourced) by the main numerical model *tcl* file to construct different members and springs and to define the analysis parameters. The files are:

Spring_Elastic.tcl: Constructs a rotational spring with an elastic response.

Spring_Rigid.tcl: Constructs a rotational spring with a large elastic stiffness (i.e., rigid).

Spring_Zero.tcl: Constructs a rotational spring with a zero stiffness (i.e., pin).

Spring_IMK.tcl: Constructs a rotational spring with a "bilinear" modified Ibarra-Medina-Krawinkler deterioration response.

Spring_IMK_RC.tcl: Constructs a rotational spring with a "peak-oriented" modified Ibarra-Medina-Krawinkler deterioration response representing the moment-rotation response of an RC beam-column.

Spring_Pinching.tcl: Constructs a rotational spring with a "pinched" deterioration response.

Spring_PZ.tcl: Constructs a rotational spring for the panel zone with a trilinear hysteretic response.

Spring_Gusset.tcl: Constructs a rotational spring for the panel zone with a bilinear response based on Hsiao et al. (2013).

FiberCHSS.tcl: Constructs a fiber cross-section for a circular hollow structural steel section.

FiberCHSS.tcl: Constructs a fiber cross-section for a rectangular hollow structural steel section.

FiberWF.tcl: Constructs a fiber cross-section for a wide-flange/I-shaped cross-section.

FiberRC_Rectangular.tcl: Constructs a fiber cross-section for a rectangular RC cross-section.

FiberRC_ Circular.tcl: Constructs a fiber cross-section for a circular RC cross-section.

FatigueMat.tcl: Constructs a *Steel02* material model masked with a *Fatigue* material to simulate ductile fracture.

ConstructPanel_Rectangle.tcl: Constructs the elements of the panel zone rectangular (parallelogram) model.

ConstructPanel_Cross.tcl: Constructs the elements of the panel zone cruciform model.

ConstructFiberColumn.tcl: Creates a fiber displacement-based column element.

ConstructBrace.tcl: Creates a fiber displacement-based brace element.

Define_Material_RC.tcl: Creates an RC uniaxial material using the Concrete04 model.

Generate_lognrmrand.tcl: Generates a log-normally distributed random variable for a specified mean and standard deviation.

LibAnalysisStaticParameters.tcl: Initiates the static analysis solver parameters.

SolutionAlgorithm.tcl: Defined the different convergence criteria parameters for static displacement-driven analysis.

SolutionAlgorithmSub.tcl: Iterates between the convergence criteria parameters defined in SolutionAlgorithm.tcl to find convergence in static displacement-driven analysis.

DynamicAnalysisCollapseSolverX.tcl: Runs dynamic (transient) analysis while alternating between solver algorithms and convergence test criteria to reach a solution.

SDRlimitTester.tcl: Reads the story-drift ratio in each story at each time increment during the analysis and checks if the collapse SDR criteria was reached.

DisplayModel3D.tcl: Displays the node numbers, deformed or mode shape in all 3 planes.

DisplayPlane.tcl: Sets up the display parameters for specified view plane and display.

8.2 Model Node and Element Notation

Figure 8.3 to Figure 8.5 show the idealized MRF and CBF models along with the node and element notation that is being used in FM-2D. This notation need to be respected by developers and users when modifying/adding to the *tcl* code, in order to avoid any potential overlap in material, element, section and node IDs. In addition to these figures, the exact formulas used to generate the ID for the different OpenSEES objects are defined below:

8.2.1 Nodes

Grid and support nodes: $(10 \cdot F + A) \cdot 10$

Beam end nodes: $100 \cdot F + 10 \cdot A + i$ where, $i = \{2,4\}$

Column end nodes: $100 \cdot S + 10 \cdot A + i$ where, $i = \{1,3\}$

Column splice nodes: $100{,}000 + 1000 \cdot S + 100 \cdot A + i$ where, $i = \{71,72\}$

Beam plastic spring nodes: $1000 \cdot F + 100 \cdot A + i$ where, $i = \{20, 40\}$

Panel zone nodes: $400{,}000 + 1000 \cdot F + 100 \cdot A + i$ where, $i = \{1,2,3,4,5,6,7,8,9,10,88,99\}$

MGP rigid offset nodes: $200{,}000 + 1000 \cdot F + 100 \cdot B + i$

where, $i = \{1,2,3,4,5,6,7,12,13,14,15,16,17\}$ for X-Bracing layout

where, $i = \{1,2,3,4,5,12,13,14,15\}$ for Chevron layout

CGP rigid offset nodes: $100{,}000 + 1000 \cdot F + 100 \cdot A + i$

where, $i = \{40,41,50,51\}$ for X-Bracing layout

where, $i = \{40,41\}$ for Chevron layout

Brace segmental nodes: $100{,}000 + 1000 \cdot F + 100 \cdot A + 41 + i$ where, $i = 1$ to n_{segment}

$200{,}000 + 1000 \cdot F + 100 \cdot B + 16 + i$ where, $i = 1$ to n_{segment}

$200{,}000 + 1000 \cdot F + 100 \cdot B + 17 + i$ where, $i = 1$ to n_{segment}

8.2.2 Elements

Column elastic elements: $600{,}000 + 1000 \cdot S + 100 \cdot A + 41 + i$ where, $i = \{0,1,2\}$

Beam elastic elements: $500{,}000 + 1000 \cdot F + 100 \cdot B + i$ where, $i = \{0,1,2\}$

Beam elastic offset elements: $500{,}000 + 1000 \cdot F + 100 \cdot B + i$ where, $i = \{2,4\}$

Panel zone elements: $7{,}000{,}000 + 1000 \cdot F + 100 \cdot A + i$ where, $i = \{1,2,3,4,5,6,7,8\}$

Floor link elements: $1000 + F$

| Brace elements: | $8{,}100{,}000 + 1000 \cdot S + 100 \cdot B$ | |
	$8{,}200{,}000 + 1000 \cdot S + 100 \cdot B$	
Brace ghost elements:	$4{,}100{,}000 + 1000 \cdot S + 100 \cdot B$	
	$4{,}200{,}000 + 1000 \cdot S + 100 \cdot B$	
MGP rigid elements:	$700{,}000 + 1000 \cdot S + 100 \cdot B + i$	where, $i = \{22,33,44,55,66,77\}$
CGP rigid elements:	$700{,}000 + 1000 \cdot S + 100 \cdot A + i$	where, $i = \{11,99\}$

8.2.3 Springs

Column springs:	$900{,}000 + 1000 \cdot F + 100 \cdot A + i$	where, $i = \{1,3\}$
Beam springs:	$900{,}000 + 1000 \cdot F + 100 \cdot A + i$	where, $i = \{2,4\}$
Panel zone spring:	$900{,}000 + 1000 \cdot F + 100 \cdot A$	
Beam MGP springs:	$900{,}000 + 1000 \cdot F + 100 \cdot A + i$	where, $i = \{22,55\}$
Column splice spring:	$900{,}000 + 1000 \cdot S + 100 \cdot A + 7$	
MGP springs:	$900{,}000 + 1000 \cdot F + 100 \cdot B + i$	where, $i = \{33,44,66,77\}$
CGP springs:	$900{,}000 + 1000 \cdot F + 100 \cdot A + i$	where, $i = \{11,99\}$

8.2.4 Material

Flexible (weak) material:	9
Rigid material:	99
UVC material:	666
Fatigue material:	100 $\rightarrow$ 300
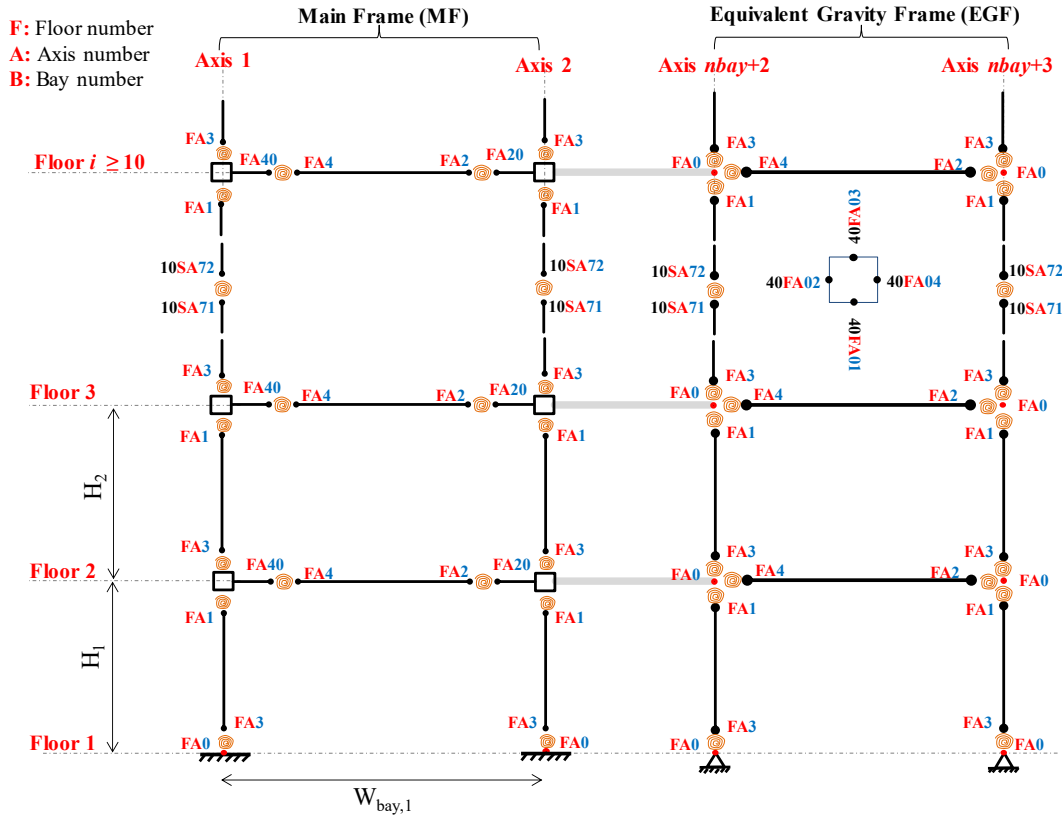
(a) Node notation
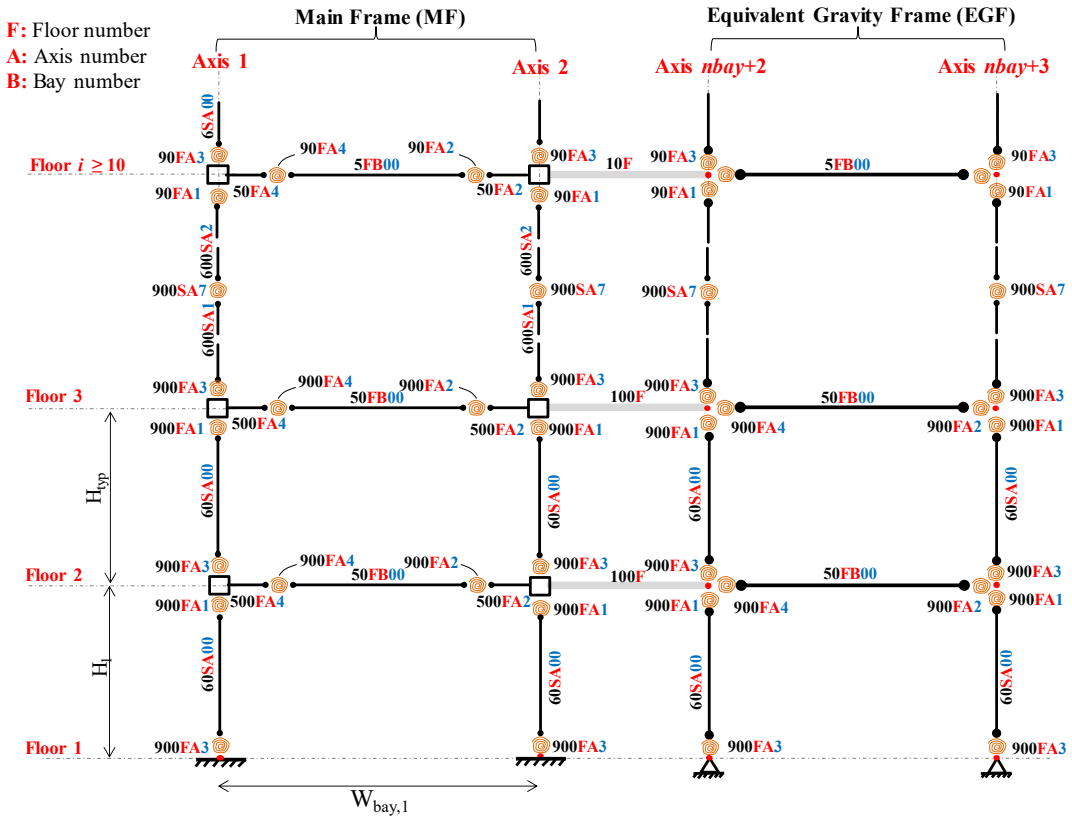
(b) Element notation

Figure 8.3. The node and element notation for the idealized moment-resisting frame model

Parallelogram Model

Cruciform Model
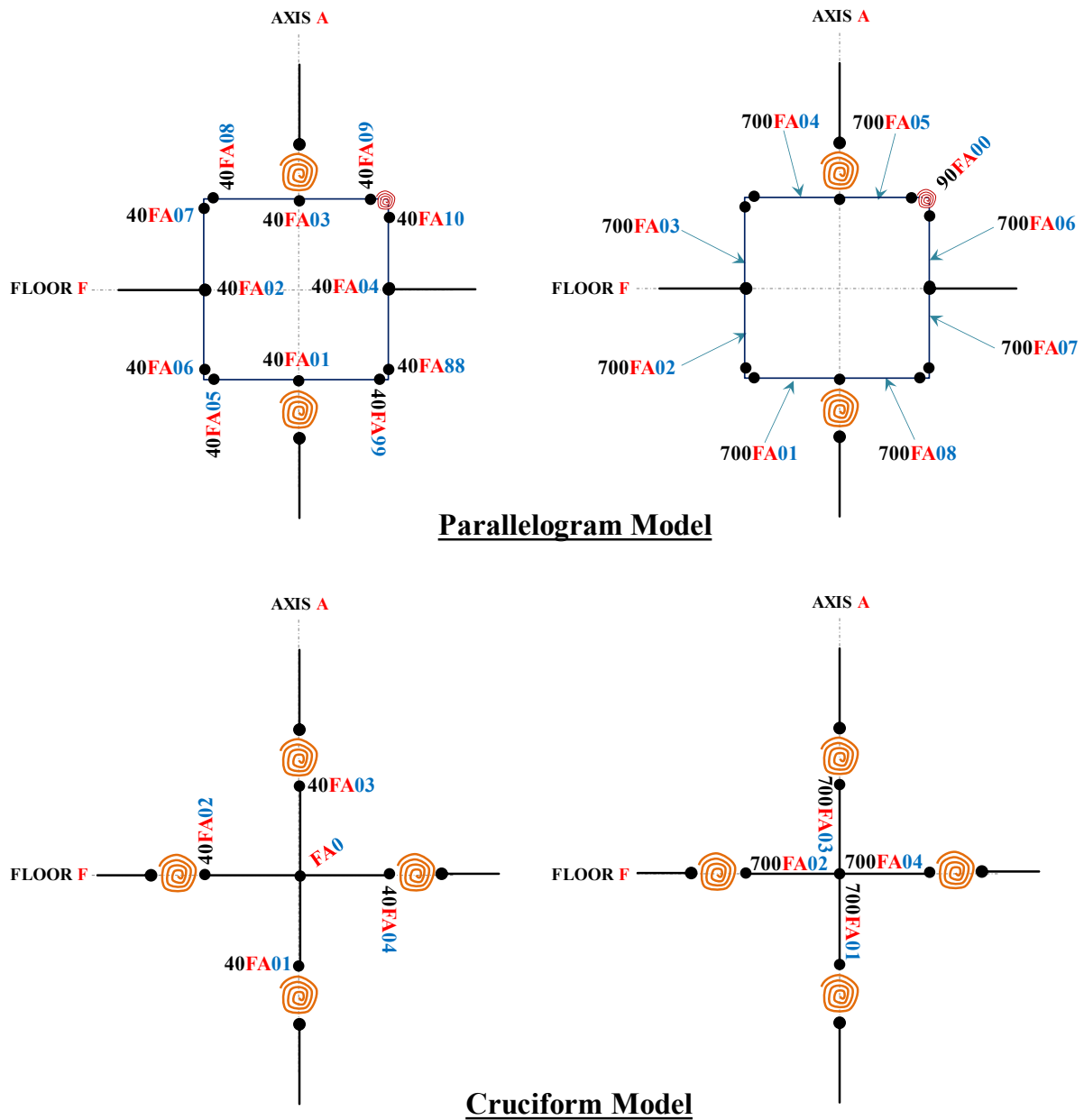
(a) Node notation   (b) Element notation

Figure 8.4. The node and element notation for the idealized panel zone model
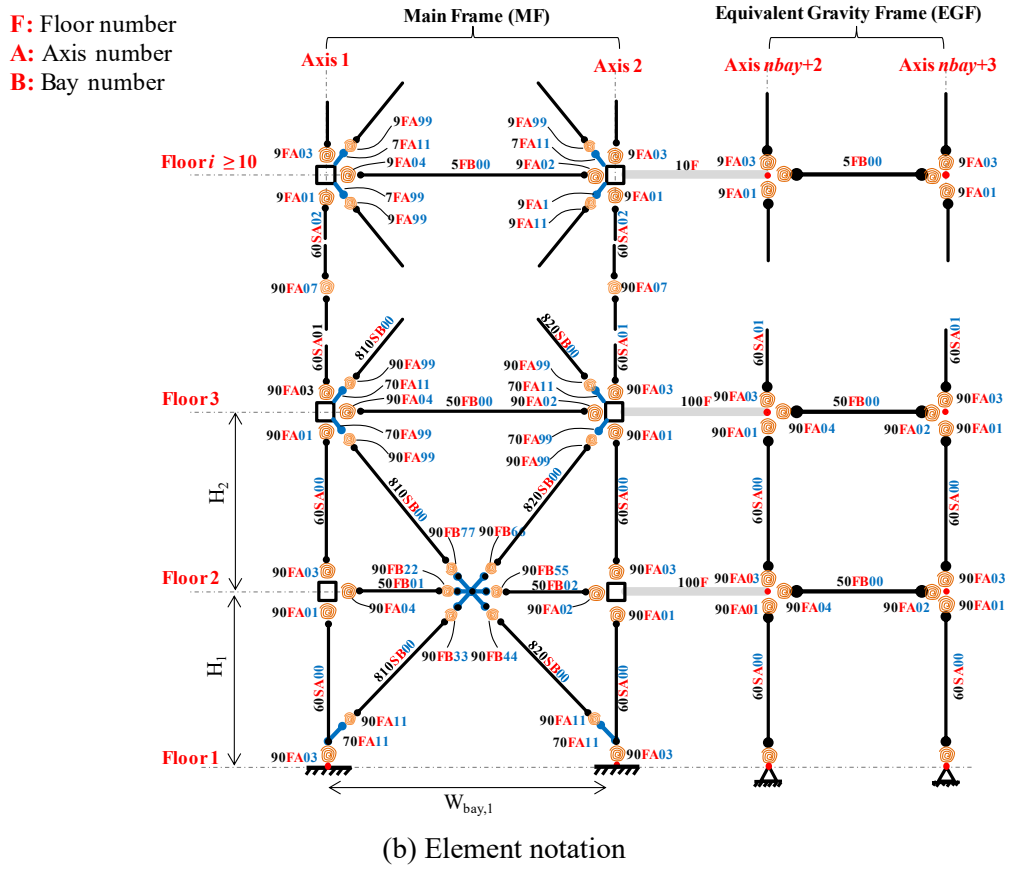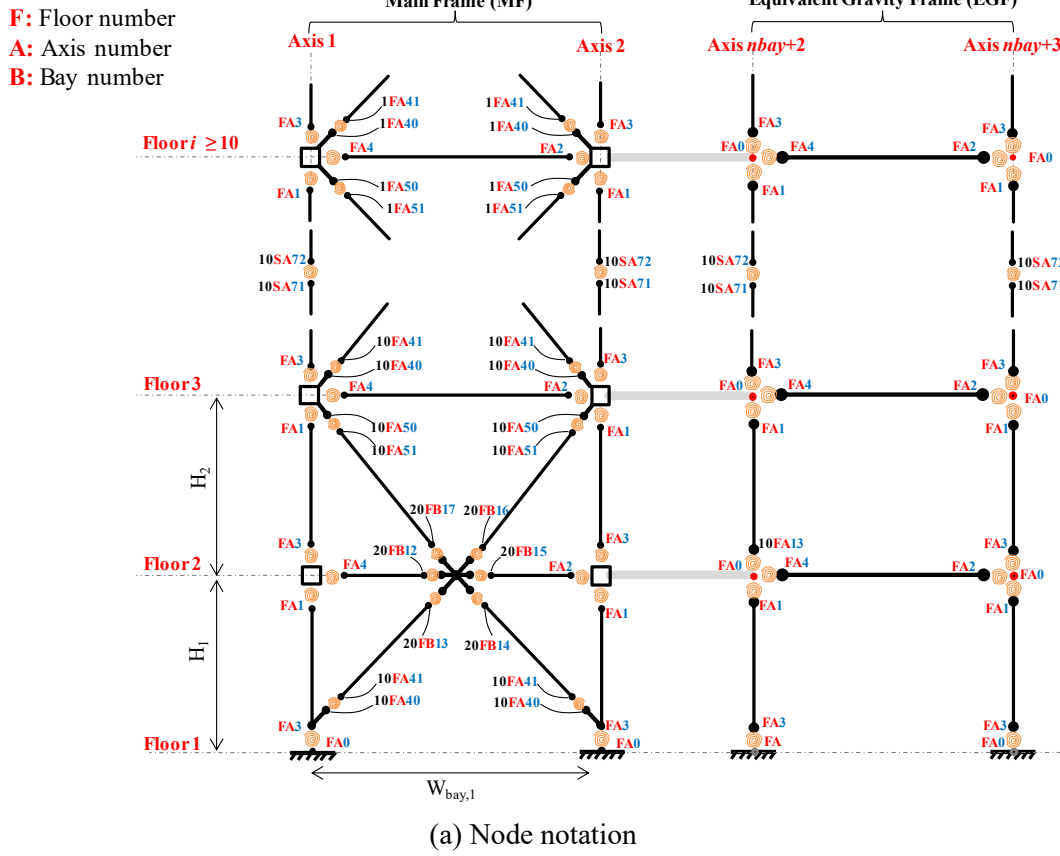
(a) Node notation

(b) Element notation

Figure 8.5. The node and element notation for the idealized concentrically-braced frame model
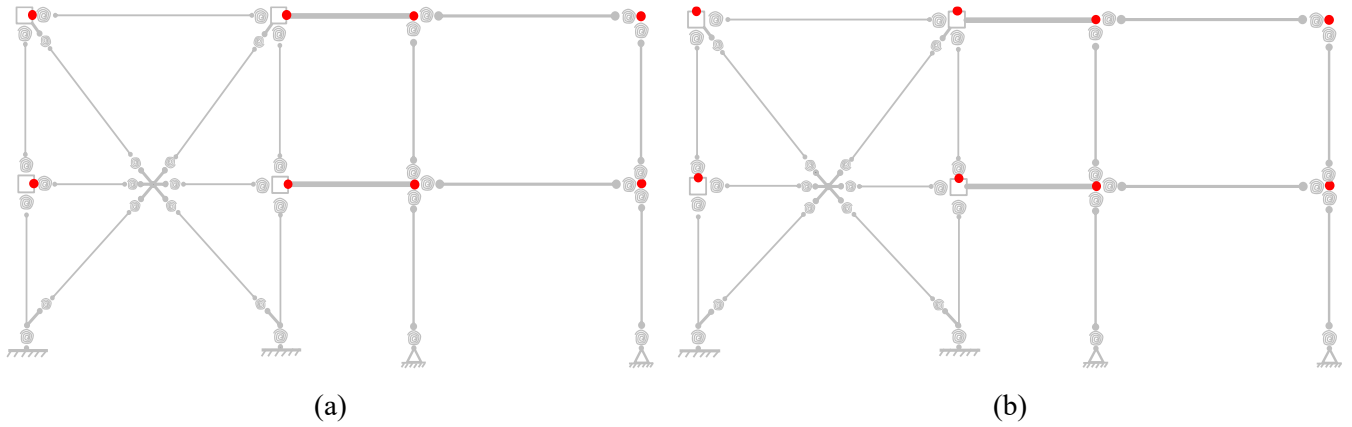
Figure 8.6. Nodes used for (a) mass and (b) gravity load application

9. References

Matlab (2019). *version 9.6.0 (R2019a)*, The MathWorks Inc., Natick, Massachusetts, USA.

AISC (2016). "Prequalified connections for special and intermediate steel moment frames for seismic applications." *ANSI/AISC 358-10*, Chicago, IL.

ASCE (2016). "Minimum design loads ans associated criteria for buildings and other structures." *ASCE/SEI 7-16*, Reston, VA.

Aslani, H., and Miranda, E. (2005). "Probabilistic earthquake loss estimation and loss disaggregation in buildings." *Report No. 157*Stanford, CA.

Chaboche, J.-L. (1989). "Constitutive equations for cyclic plasticity and cyclic viscoplasticity." *International journal of plasticity*, 5(3), 247-302, DOI: 10.1016/0749-6419(89)90015-6.

Cornell, C. A., and Krawinkler, H. (2000). "Progress and challenges in seismic performance assessment." *PEER Center News*, 3(2), 1-3, DOI:

de Castro e Sousa, A., Suzuki, Y., and Lignos, D. (2020). "Consistency in solving the inverse problem of the voce-chaboche constitutive model for plastic straining." *Journal of Engineering Mechanics*, 146(9), 04020097, DOI: 10.1061/(ASCE)EM.1943-7889.0001839.

de Castro e Sousa, A. A., and Lignos, D. G. (2018). "On the inverse problem of classic nonlinear plasticity models." *Report no. 231968*, Ecole Polytechnique Federale de Lausanne (EPFL), Lausanne, Switzerland.

Deierlein, G. G. (2004). "Overview of a comprehensive framework for earthquake performance assessment." *PEER Report 2004/05*, University of California, Berkeley, Berkeley, CA, USA, 15-26.

Eads, L., Miranda, E., and Lignos, D. G. (2015). "Average spectral acceleration as an intensity measure for collapse risk assessment." *Earthquake Engineering & Structural Dynamics*, 44(12), 2057-2073, DOI: 10.1002/eqe.2575.

Elkady, A., Hwang, S. H., and Lignos, D. G. (2018). "Earl: Toolbox for earthquake risk and loss assessment of building assets." *Proc., 16th European Conference on Earthquake Engineering*, Thessaloniki, Greece.

Elkady, A., and Lignos, D. G. (2014). "Modeling of the composite action in fully restrained beam-to-column connections: implications in the seismic design and collapse capacity of steel special moment frames." *Earthquake Engineering & Structural Dynamics*, 43(13), 1935-1954, DOI: 10.1002/eqe.2430.

Elkady, A., and Lignos, D. G. (2015). "Effect of gravity framing on the overstrength and collapse capacity of steel frame buildings with perimeter special moment frames." *Earthquake Engineering & Structural Dynamics*, 44(8), 1289–1307, DOI: 10.1002/eqe.2519.

FEMA (2000). "Prestandard and commentary for the seismic rehabilitation of buildings." *Report FEMA-356*, Federal Emergency Management Agency, Washington, DC.

FEMA (2012a). "Seismic performance assessment of buildings." *Report FEMA P-58-1*, Federal Emergency Management Agency, Washington, DC.

FEMA (2012b). "Seismic performance assessment of buildings - volume 3 - performance assessment calculation tool (pact), version 2.9.65." *Report P-58-3.1*, Federal Emergency Management Agency, Washington, DC, USA, Washington, DC, USA.

Gupta, A., and Krawinkler, H. (1999). "Seismic demands for the performance evaluation of steel moment resisting frame structures." *Report No. 132*, The John A. Blume Earthquake Engineering Center, Stanford University, CA.

Hamburger, R. O., Rojahn, C., Heintz, J. A., and Mahoney, M. G. (2012). "Fema p58: Next-generation building seismic performance assessment methodology." *Proc., 15th World Conference on Earthquake Engineering*, Lisbon, Portugal.

Hartloper, A. R., De Castro e Sousa, A., and Lignos, D. (2019a). "Sensitivity of simulated steel column instabilities to plasticity model assumptions." *Proc., 12th Canadian Conference on Earthquake Engineering (12CCEE)*, Quebec City, Canada.

Hartloper, A. R., de Castro e Sousa, A. A., and Lignos, D. G. (2019b). "A nonlinear isotropic/kinematic hardening model for materials with discontinuous yielding." *Report No. 271062*, Ecole Polytechnique Federale de Lausanne (EPFL), Lausanne, Switzerland.

Haselton, C. B., Liel, A. B., Lange, S. T., and Deierlein, G. G. (2008). "Beam-column element model calibrated for predicting flexural response leading to global collapse of rc frame buildings." *Report No. 2007/03*, Pacific Earthquake Engineering Research Center, Berkeley, CA, USA.

Hsiao, P. C., Lehman, D. E., and Roeder, C. W. (2013). "A model to simulate special concentrically braced frames beyond brace fracture." *Earthquake Engineering & Structural Dynamics*, 42(2), 183-200, DOI: 10.1002/eqe.2202.

Ibarra, L. F., Medina, R. A., and Krawinkler, H. (2005). "Hysteretic models that incorporate strength and stiffness deterioration." *Earthquake Engineering & Structural Dynamics*, 34(12), 1489-1511, DOI: 10.1002/eqe.495.

Karamanci, E., and Lignos, D. G. (2014). "Computational approach for collapse assessment of concentrically braced frames in seismic regions." *Journal of Structural Engineering*, 140(SPECIAL ISSUE: Computational Simulation in Structural Engineering), DOI: 10.1061/(ASCE)ST.1943-541X.0001011.

Krawinkler, H. (1978). "Shear in beam-column joints in seismic design of steel frames." *Engineering Journal*, 15(3), 82-91, DOI:

Krawinkler, H., and Miranda, E. (2004). "Performance-based earthquake engineering." *Earthquake Engineering: From Engineering Seismology to Performance-Based Engineering*, 9, 9-1-9-59, DOI:

Lignos, D. G., Hartloper, A., Elkady, A., Deierlein, G. G., and Hamburger, R. (2019). "Proposed updates to the asce 41 nonlinear modeling parameters for wide-flange steel columns in support of performance-based seismic engineering." *Journal of Structural Engineering*, 145(9), 04019083, DOI: 10.1061/(ASCE)ST.1943-541X.0002353.

Lignos, D. G., and Krawinkler, H. (2011). "Deterioration modeling of steel components in support of collapse prediction of steel moment frames under earthquake loading." *Journal of Structural Engineering*, 137(11), 1291-1302, DOI: 10.1061/(ASCE)ST.1943-541X.0000376.

Lignos, D. G., Putman, C., and Krawinkler, H. (2013). "Seismic assessment of steel moment frames using simplified nonlinear models." *Computational Methods in Earthquake Engineering*, 30, 91-109, DOI: 10.1007/978-94-007-6573-3_5.

Lignos, D. G., Putman, C., Zareian, F., and Krawinkler, H. (2011). "Seismic evaluation of steel moment frames and shear walls using nonlinear static analysis procedures." *Proc., ASCE Structures Congress*, 2204-2215.

Manson, S. S. (1965). "Fatigue: A complex subject—some simple approximations." *Experimental mechanics*, 5(4), 193-226, DOI:

Mazzoni, S., McKenna, F., Scott, M. H., and Fenves, G. L. (2006). "Opensees command language manual." *Pacific Earthquake Engineering Research (PEER) Center*, 264, DOI:

Mckenna, F. T. (1997). "Object-oriented finite element programming: Frameworks for analysis, algorithms and parallel computing." Ph.D., University of California.

Menegotto, M., and Pinto, P. E. (1973). "Method of analysis for cyclically loaded rc frames including changes in geometry and non-elastic behaviour of elements under combined normal force and bending." *Proc., IABSE Congress Reports of the Working Commission*.

Moehle, J., and Deierlein, G. G. (2004). "A framework methodology for performance-based earthquake engineering." *Proc., 13th World Conference on Earthquake Engineering*, Vancouver, Canada.

Porter, K. A. (2003). "An overview of peer's performance-based earthquake engineering methodology." *Proc., 9th International Conference on Applications of Statistics and Probability in Civil Engineering*, San Francisco, USA.

Ramirez, C. M., and Miranda, E. (2009). "Building-specific loss estimation methods and tools for simplified performance-based earthquake engineering." *Report No. 171*, The John A. Blume Earthquake Engineering Center, Stanford University, Stanford, CA.

SEAOC (1995). "Performance based seismic engineering of buildings." Structural Engineers Association of California (SEAOC), Sacramento, California.

Skiadopoulos, A., Elkady, A., and Lignos, D. G. (2020). "Proposed panel zone model for seismic design of steel moment-resisting frames." *Journal of Structural Engineering*, DOI: 10.1061/(ASCE)ST.1943-541X.0002935.

Seismic performance prediction program (2017). *version 2.6*, Haselton Baker Risk Group, LLC. Chico, CA, USA.

Uriz, P., Filippou, F. C., and Mahin, S. A. (2008). "Model for cyclic inelastic buckling of steel braces." *Journal of structural engineering*, 134(4), 619-628, DOI: 10.1061/(ASCE)0733-9445(2008)134:4(619).

Vamvatsikos, D., and Cornell, C. A. (2002). "Incremental dynamic analysis." *Earthquake Engineering & Structural Dynamics*, 31(3), 491-514, DOI: 10.1002/eqe.141.

Voce, E. (1948). "The relationship between dtress and strain for homogeneous deformation." *Journal of the Institute of Metals*, 74, 537-562, DOI:

von Mises, R. (1913). "Mechanik der festen körper im plastisch-deformablen zustand." *Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen, Mathematisch-Physikalische Klasse*, 1, 582-592, DOI:

Yang, T. Y., Moehle, J., Stojadinovic, B., and Der Kiureghian, A. (2009). "Seismic performance evaluation of facilities: Methodology and implementation." *Journal of Structural Engineering*, 135(10), 1146-1154, DOI: 10.1061/(ASCE)0733-9445(2009)135:10(1146).

Zareian, F., and Medina, R. A. (2010). "A practical method for proper modeling of structural damping in inelastic plane structural systems." *Computers & structures*, 88(1), 45-53, DOI: 10.1016/j.compstruc.2009.08.001.