INTERACTIVE INTERFACE FOR DYNAMIC ANALYSIS PROCEDURES (II-DAP)

Version 1.0

Manual and Examples

(IIDAP YouTube Channel: video tutorials)

Prepared by

Dr. Ahmed Elkady

Prof. Dr. Dimitrios G. Lignos

Resilient Steel Structures Laboratory (RESSLAB)

School of Architecture, Civil and Environmental Engineering (ENAC)

Swiss Federal Institute of Technology in Lausanne (EPFL), Switzerland

January, 2018

Table of Contents

1. Introduction

The Interactive Interface for Dynamic Analysis Procedures (IIDAP) is a standalone MATLAB-based program that performs various dynamic analysis procedures for deteriorating and non-deteriorating single degree-of-freedom (SDoF) systems. These procedures include linear and nonlinear response history analysis (RHA), incremental dynamic analysis (IDA), response spectrum analysis (RSA), and collapse spectrum analysis (CSA). The program employs a range of component deterioration hysteretic models that can adequately capture all the important deterioration modes of structural components and/or systems. IIDAP also incorporates a seismic hazard module that can be used to compute the mean annual frequency of collapse (λ_c) in support of probabilistic performance-based earthquake engineering. Several interactive options are available to visualize the results, making IIDAP a useful tool for practitioners, academics, and researchers.

This report summarizes the main modules and features of IIDAP. The theory and procedures for each analysis type are explained. Finally, illustrative examples are presented to show the visualization capabilities of the program.

2. IIDAP Main Console and Components

Figure 1 shows the main console of IIDAP. The console is divided into three main panels. Panel A is the project file management panel. Panel B is the project definition panel, which includes four buttons that define the four main components of any project: 1) the SDoF system parameters, 2) the system force-deformation response, 3) the input ground motion, and 4) the analysis type. For a new project, these four components shall be defined in this hierarchical order; from top to bottom. Panel B includes three buttons for 1) running the analysis, 2) visualizing the results, and 3) saving the results.
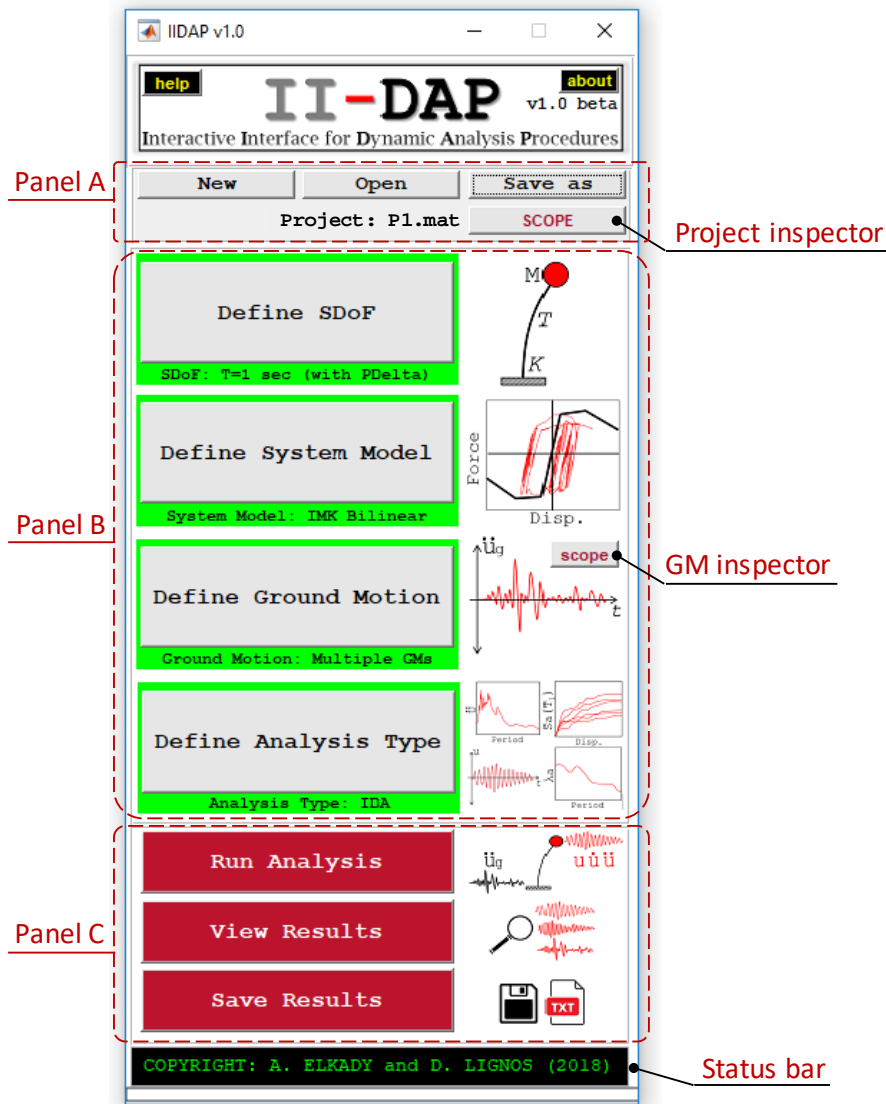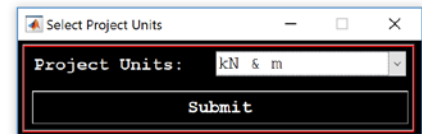

Figure 1. IIDAP v1.0 main console

2.1 Project File Module

Panel A -of the main console- is the project file management panel, which creates a new project or opens an existing project. This panel includes four buttons:

- "New": This button is used to start a new project. When clicking this button, the user is asked to specify the name of the project as well as the saving directory (Figure 2a). The saved project file has a *.mat extension. This is then followed by a dialogue box to specify the project units as shown in Figure 2b.
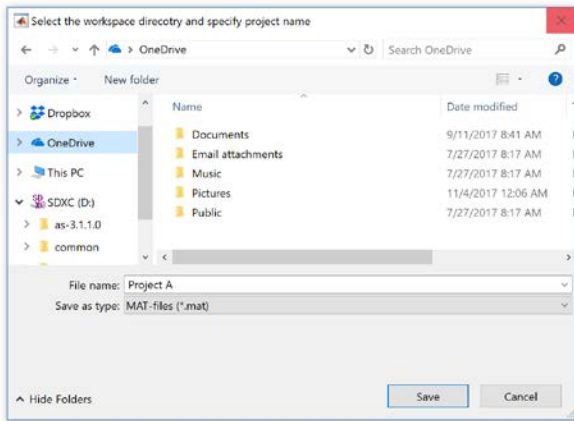
(a) create new project dialogue   (b) project units dialogue

Figure 2. Creating a new project file

- "Open": This button is used to open an existing project. When clicking this button, the user is asked to select the existing *.mat project file.

- "Save as": This button is used to save a copy of the current project under a new name and/or a new directory.

- "Scope": This button can be used at any point during the project definition to check the current project components and parameters. This button will open a window that shows a summary of the current project definition as shown in Figure 3.

```
****************************************************
****************************************************
                  PROJECT SUMMARY
****************************************************
****************************************************


SDoF PROPERTIES
----------------------------------------------------
Period: 1 sec
Mass: 10 kN.sec2/m
Stiffness: 394.784 kN/m
Damping Coefficient: 2%
P-Delta Inclusion: YES
Stability Coefficient: 0.025
----------------------------------------------------


SYSTEM TYPE
----------------------------------------------------
Bi-Linear System Model
Ke: 394.784 kN/m
Kp: 20 kN/m
Ultimate disp.: 1 m
----------------------------------------------------


GROUND MOTION
----------------------------------------------------
Ground Motion Type: Multiple Ground Motion Records
Number of Ground Motions: 44
Scale Factor: 1
Free Vibration Time: 0
----------------------------------------------------


ANALYSIS TYPE
----------------------------------------------------
Analysis Type: Incremental Dynamic Analysis
IDA Step Increment: 0.1 g
IDA Step Tolerance: 0.02 g
Collapse Disp. Limit: 1 m
Collapse Acc. Limit: 5 g
----------------------------------------------------
```
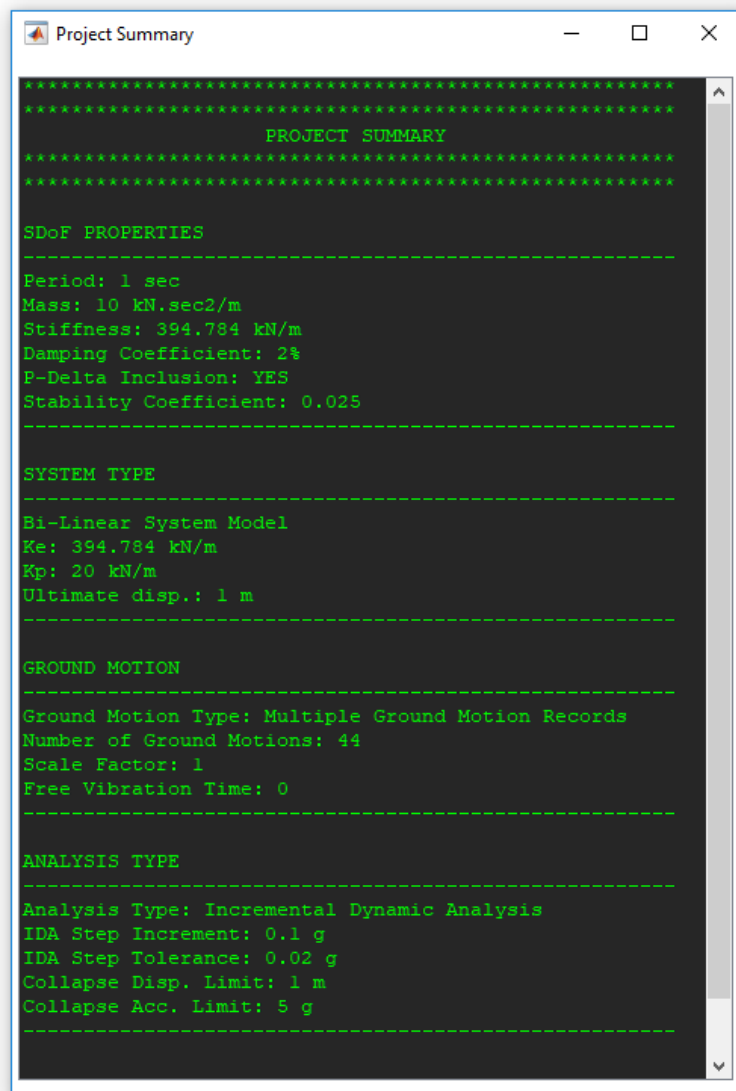
Figure 3. Project summary window

2.2 SDoF Module

The SDoF module is shown in Figure 4. This module is used to define the main SDoF parameters: a) the period, T, b) the mass, M, and c) the lateral "elastic" stiffness, K. From the popup menu at the top, the user can choose to define any two of those three parameters. The third parameter is then calculated using Equations 1a-c. The user shall specify the damping coefficient, ζ. This is then used to calculate the damping ratio, C, using Equation 2, in which ω is the circular frequency of the employed system.

Figure 4. The SDoF module

$$T = 2\pi\sqrt{\frac{M}{K}}$$

(1a)

$$M = K \cdot \left(\frac{T}{2\pi}\right)^2$$

(1b)

$$K = \frac{M}{\left(T/2\pi\right)^2} \qquad (1c)$$

$$C = 2 \cdot \zeta \cdot \omega \cdot M, \quad \omega = 2\pi / T \qquad (2)$$

From the same module, the user can specify whether to include or exclude the system P-Delta effects. To include P-Delta, the user has two options: a) define the system's gravity load, P and height, H, or b) specify the stability coefficient, θ. For option "a", the stability coefficient is internally computed by Equation 3. The stability coefficient is then used to modify (rotate) the defined system model backbone curve (see Section 2.2) due to P-Delta effects as shown in Figure 5.

$$\theta = \frac{P \cdot \delta}{H \cdot V} = \frac{P}{H \cdot K} \qquad (3)$$
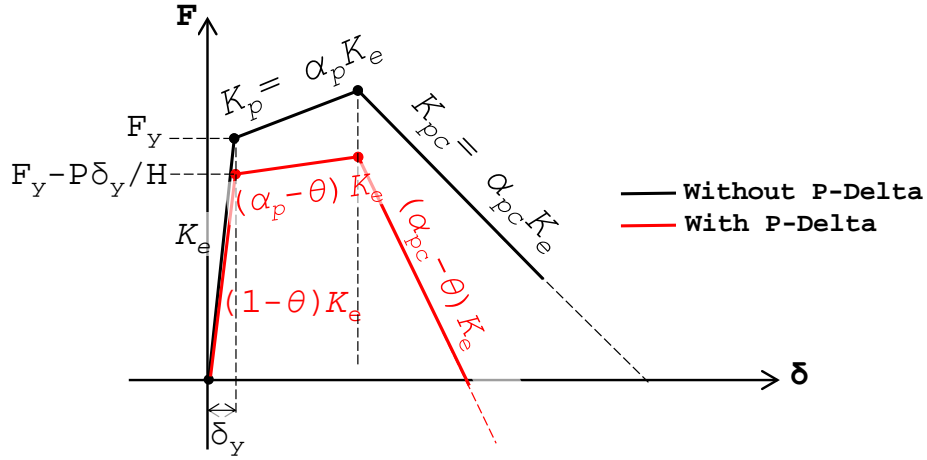

Figure 5. Effect of P-Delta on system backbone curve

2.3 System Model Module

The System Model module is shown in Figure 6. In the current IIDAP version, six system models (i.e., force-deformation hysteretic models) are available. These include: 1) *Linear* elastic non-deteriorating model, 2) *Bi-Linear* non-deteriorating model (symmetric backbone curves for negative and positive loading directions), 3) *IMK-Bilinear* deteriorating asymmetric model

9

with optional smooth-transition between loading branches, 4) *IMK-Pinching* deteriorating asymmetric model with pinched hysteretic behavior, 5) *IMK-Peak Oriented* deteriorating asymmetric model with peak-oriented hysteretic behavior, and 6) *Flag-Shaped* deteriorating model with asymmetric flag-shaped hysteretic behavior.
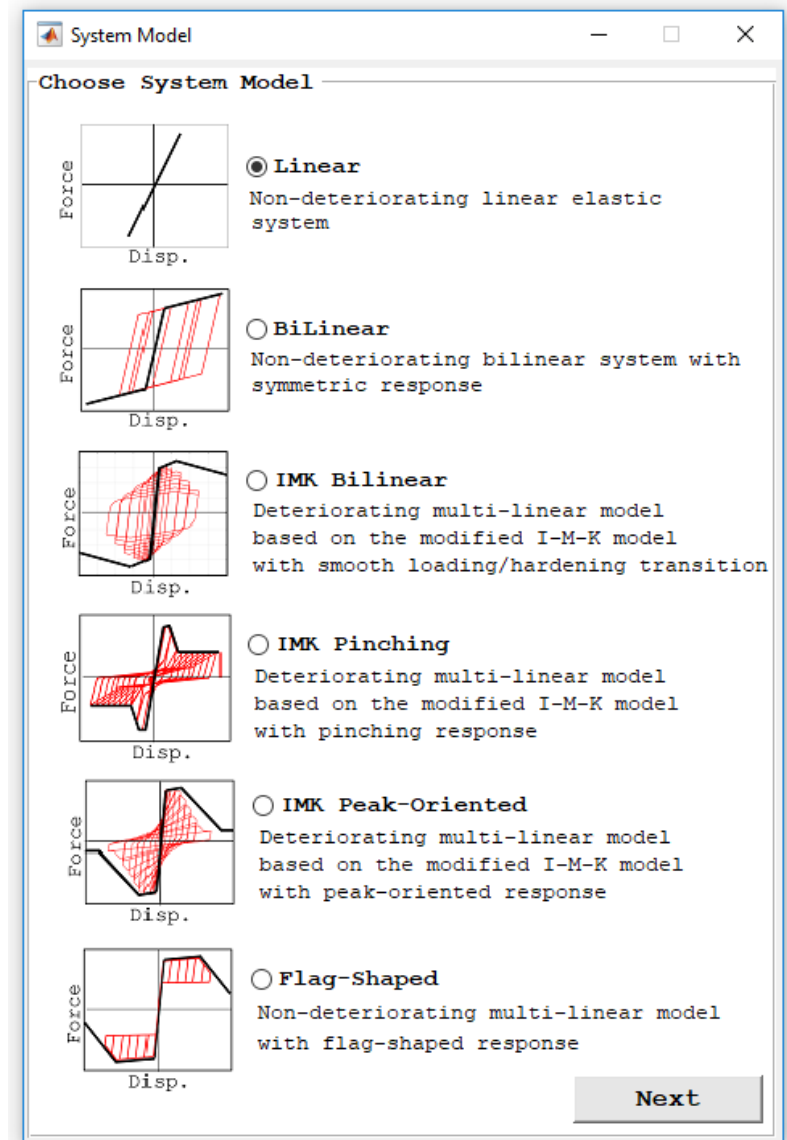

Figure 6. The System Model module

2.3.1 Linear "Elastic" Model

This model is defined only by the elastic stiffness, K_e. However, the user may specify an ultimate force/displacement at which the system loses its lateral load capacity (i.e., force will drop to zero). This is done through the Linear Model module shown in Figure 7. By default,

a relatively large reference value of "999" (inches or meters) is assigned to the system failure displacement. The right hand side of the Linear Model module shows a sample plot of the linear stiffness with and without P-Delta (if applicable).
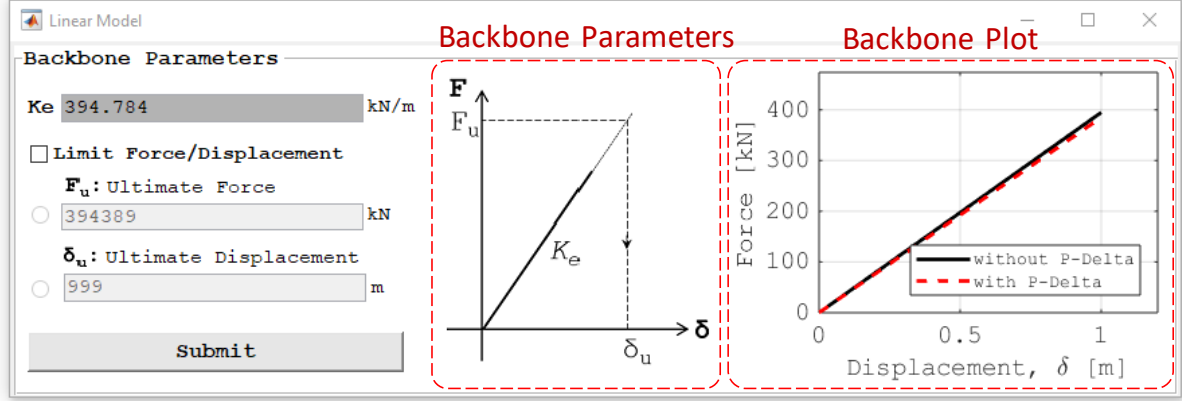

Figure 7. The Linear model module

2.3.2 Bi-Linear Model

The Bi-Linear model module is shown in Figure 8. This model is defined by three main parameters:

1) The elastic stiffness, K_e. This is taken equal to the previously defined SDoF stiffness, K.

2) The yielding point (δ_y, F_y). This point can be defined by either specifying the yield force, F_y, or the yield displacement, δ_y. Herein, the yield force is expressed as a fraction of the system weight $W=Mg$, in which g is the gravity acceleration 386.2 in/sec^2 or 9.81 m/sec^2.

3) The ratio of the post-yield stiffness, K_p, to the elastic stiffness, K_e, (expressed as a percentage).

The user is also given the option to specify an ultimate displacement, δ_u, at which the force drops to zero. The ultimate displacement is expressed as a multiple of the yield displacement, δ_y. By default, $\delta_u = 999$ meters or inches so as it works as a simple bilinear model.

11

In the Bi-Linear model module, the defined backbone curves with and without the P-Delta (if applicable) are plotted automatically on the right hand side of the module (see Figure 8) as the user defines the backbone parameters.
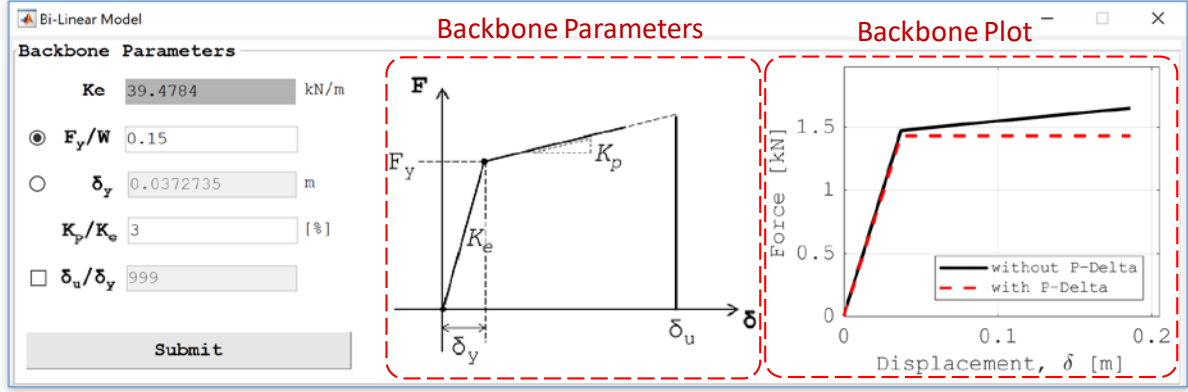

Figure 8. The Bi-Linear model module

2.3.3 IMK-Bilinear Model

The IMK-Bilinear model module is shown in Figure 9. This model is basically the modified Ibarra-Medina-Krawinkler (IMK) deterioration model (Ibarra et al. 2005; Lignos and Krawinkler 2011) with optional control over the smooth transition between the loading branch and the hardening branch. This model in the general case has an asymmetric hysteretic response and considers basic strength, post-capping strength and stiffness deterioration. For a symmetric IMK-Bilinear mode, the user shall define 6 backbone parameters as shown in Panel A in Figure 9:

- δ_p/δ_y pre-capping plastic displacement (normalized by the yield displacement, δ_y)

- δ_{pc}/δ_y post-capping plastic displacement (normalized by δ_y)

- δ_u/δ_y ultimate displacement (normalized by δ_y)

- F_y/W yielding force (normalized by system's weight $W=M{\cdot}g$)

- F_{max}/F_p maximum (capping) force to yield force ratio

- F_{res}/F_p residual force to yield force ratio

12

Additional 8 parameters may be specified to define the amount/rate of cyclic deterioration as shown in Panel B. Three main modes of cyclic deterioration are considered: 1) deterioration in basic strength (i.e., F_y) after each cycle, 2) deterioration in the post-capping strength (i.e., K_{pc} and subsequently F_{max}), and 3) deterioration in the unloading stiffness after each excursion. This is based on the energy dissipated at each cycle/excursion assuming that the system has a pre-defined reference energy capacity, E_t. The reference energy is calculated as using Equation (4), in which Λ is the reference cumulative deformation capacity.

$$E_t = \Lambda \cdot F_y = \lambda \cdot \delta_y \cdot F_y \tag{4}$$

The parameter Λ is calculated as the product of the pre-capping deformation δ_p and the parameter λ. The latter is defined by the user. Note that a higher λ value results in lesser deterioration. The rate of deterioration is controlled by the parameter c where a higher c value typically results in higher deterioration rate in later cycles. Note that a reasonable value of c is between 1.0 and 2.0. The parameters D^+ and D^- define the decrease in rate of cyclic deterioration in the positive and/or negative loading direction. The $D^{+/-}$ values must be larger than or equal to 1.0. When the rate of cyclic deterioration is the same in both loading directions, the $D^+=D^-$.

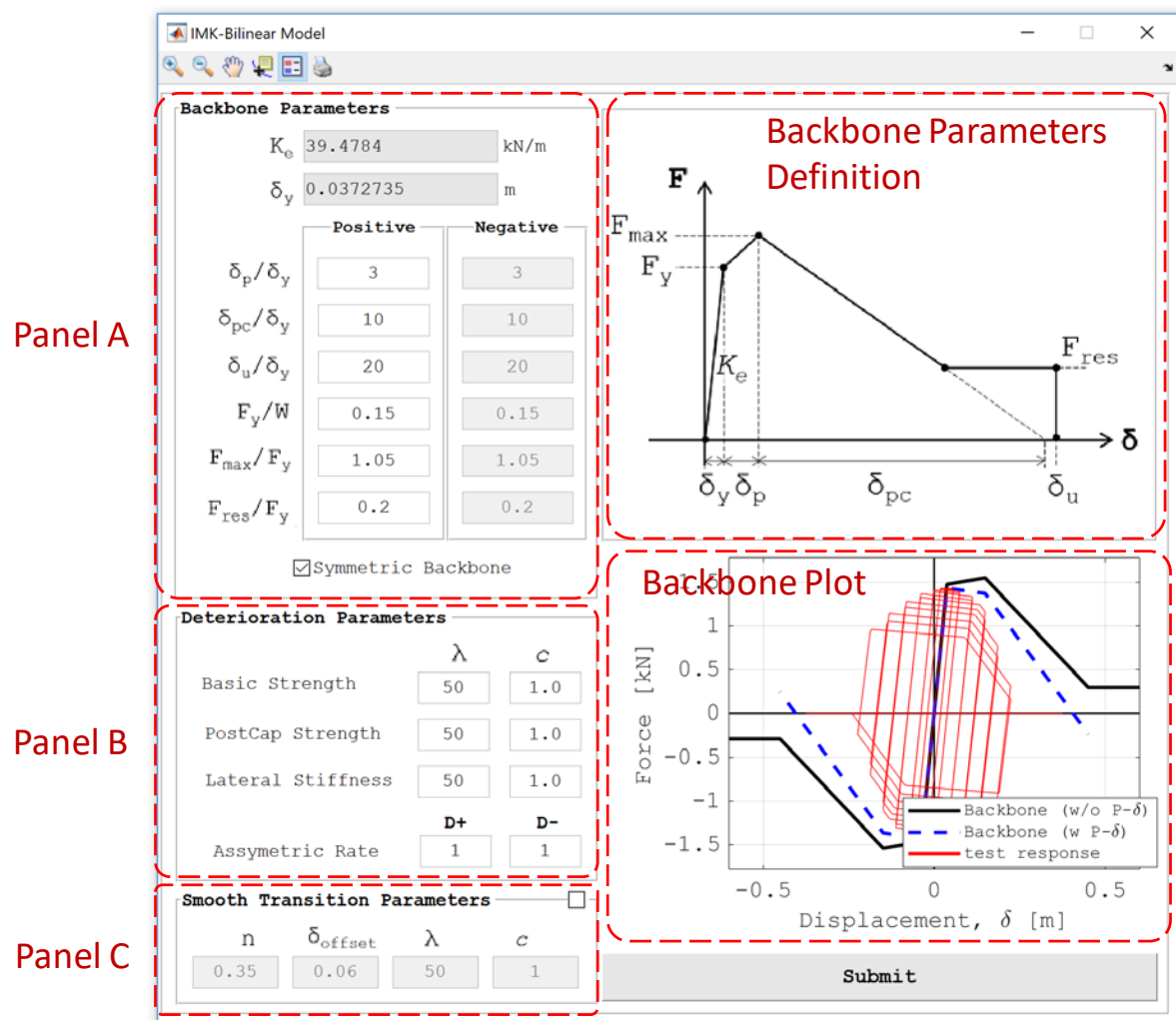Panel C provides the option to include a smooth hysteretic response.

Figure 9. The IMK-Bilinear model module

2.3.4   IMK-Pinching/Peak Oriented Model

The IMK-Pinching Model module is shown in Figure 10. This model is based on the modified Ibarra-Medina-Krawinkler deterioration model with pinching hysteretic response (Ibarra et al. 2005). Hence, it is defined using the same parameters of the IMK-Bilinear model (see Section 2.3.3). Only two additional parameters are used to control the amount of pinching. These parameters are κF and κD, which represent the ratio of the reloading (breaking) point, at the end of pinching branch, to the peak force or displacement up till the current loading excursion, respectively, as illustrated in Figure 11. The same module is used for the IMK-Peak Oriented Model. The only difference in this case is that both κF and κD are set equal to 1.0.
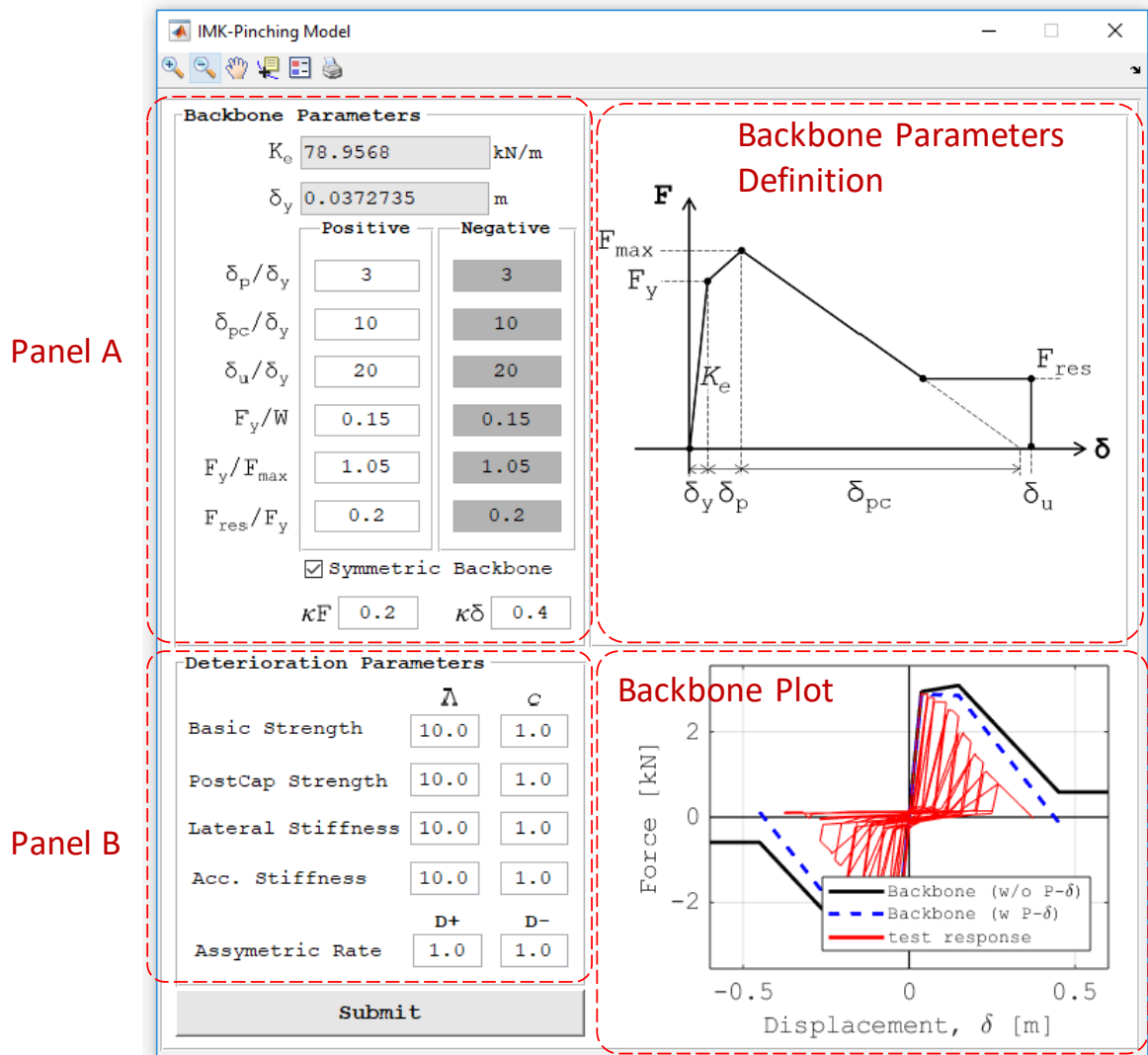

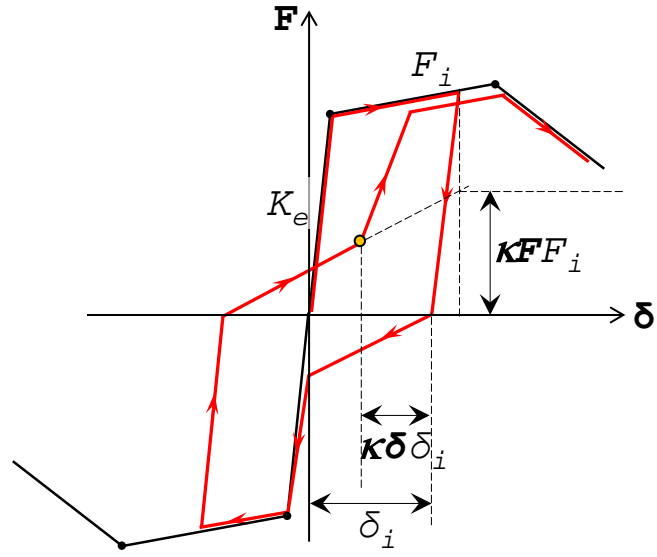Figure 10. The IMK-Pinching model module

15

Figure 11. The IMK-Pinching model module

2.3.5 Flag-Shaped Hysteretic Model

The Flag-Shaped Model module is shown in Figure 12. This model is defined using the same backbone parameters of the IMK-Bilinear model. Two additional parameters are used to define the point at which the system's elastic stiffness is engaged. These parameters are F_{lim} and K_{ul} which represent the limit force at the stiffness break point and the unloading stiffness up to the breaking point, respectively.
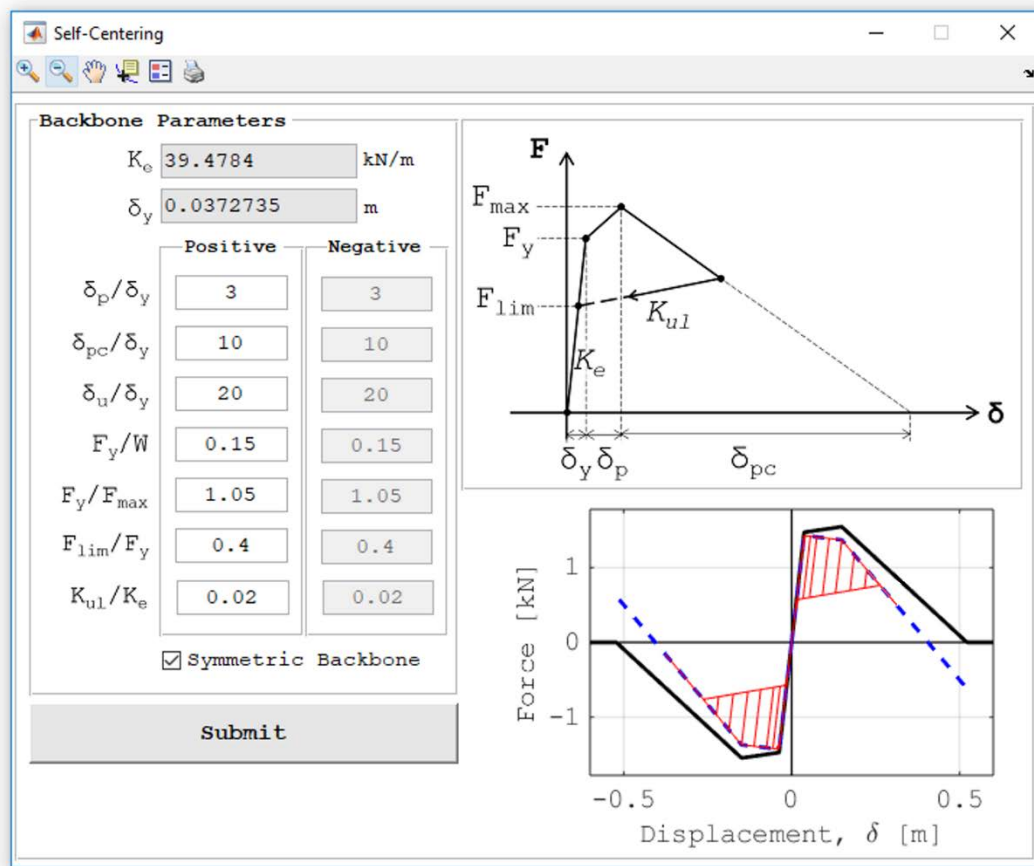

Figure 12. The Flag-Shaped model module

17

2.4 Ground Motion Module

The Ground Motion module is shown in Figure 13. This module is used to define/import the input ground motion(s). For this purpose, the user is offered three options: 1) importing a single ground motion record (see Panel A), 2) importing multiple ground motion records (see panel B), and 3) defining a harmonic ground motion (see Panel C). The procedure to define each option is explained below:

- **Option 1:** Single ground motion record. In this case, the user should browse for the ground motion data file using the browse button. The data file may have file formats such as *.txt, *.csv, *.xls. The data within the file can be formatted in any of those forms:

- **Format 1:** a single column of acceleration data, expressed <u>in gravity acceleration unit, g</u> (see Figure 14a). In this case the user shall specify the time step of the ground motion data.

- **Format 2:** Two columns of data (see Figure 14b). The first column includes the ground motion time <u>in seconds</u> and the second column includes the acceleration <u>in g units</u>. In this case, the ground motion time step is deduced internally by IIDAP from the input data.

- **Format 3:** Acceleration data listed in multiple columns (more than two columns) of data in g units (see Figure 14c). In this case the user shall specify the ground motion time step.

- **Option 2:** Multiple ground motion records. In this case, the user shall browse for the folder containing the multiple ground motions data files. Each of the individual data files can have any of the three formats discussed above in Option 1. In this case, the user shall include in the same ground motion folder an additional text file, named "GM

Data.txt". This file shall contain two columns of data; the first one includes the ground motion record file name (including the file extension). The second column contains the corresponding time step in seconds.

- **Option 3:** the user may define a ground motion in the form of a harmonic wave for basic comprehension of structural dynamics. Four types of harmonic wave are available to choose from using the popup menu (see Panel B). These include sine, cosine, rectangular, and triangular waves. For any wave, the user shall specify the wave amplitude, period, and duration as defined in Figure 15.

Figure 13. The Ground Motion module

In Options 1 and 2, the user could specify a scale factor to scale the ground acceleration. This factor is by default set equal to 1.0. It is important to note that the input ground acceleration in all cases will be multiplied internally by the gravity acceleration, g, to transform it into displacement/sec^2 units. If the user has a ground motion data file in which the acceleration is

in units of m/sec^2, the user can still import this file after making sure to specify a scale factor equal to $1/g$. Note that the value of g used in IIDAP are by default, 9.81 m/sec^2 or 386.2 in/sec^2.

The Ground Motion module also gives the user the option to specify an additional free vibration time following the main ground motion. This free vibration time can be specified in Panel D (see Figure 13). By default, the free vibration time is zero. Note that the additional free vibration time is automatically assigned the same time step of the main ground motion.

The user can inspect the defined/imported ground motion record(s) using the "scope" button next to the "Define Ground Motion" button in the main console (see Figure 1). The "scope" button will open the ground motion plot window shown in Figure 16. The user can toggle between plots of the acceleration/velocity/displacement histories as well as the Fourier transform of the ground motion. The user can use the dropdown menu at the top of the plot to alternate between the different ground motions (if applicable).

(a) format #1   (b) format #2   (c) format #3
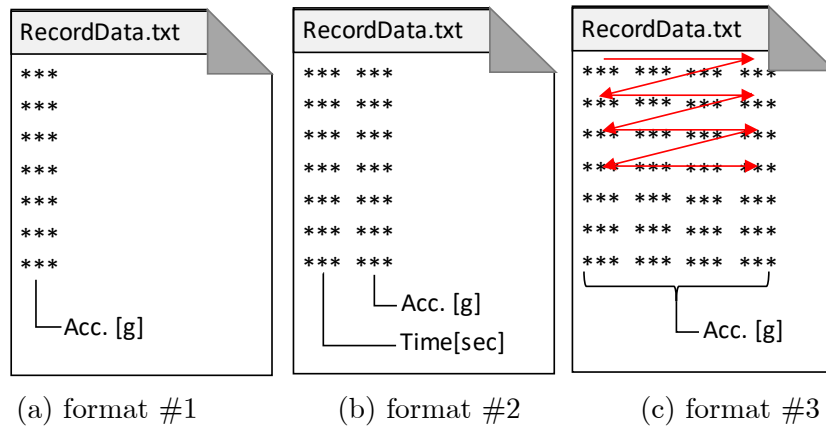
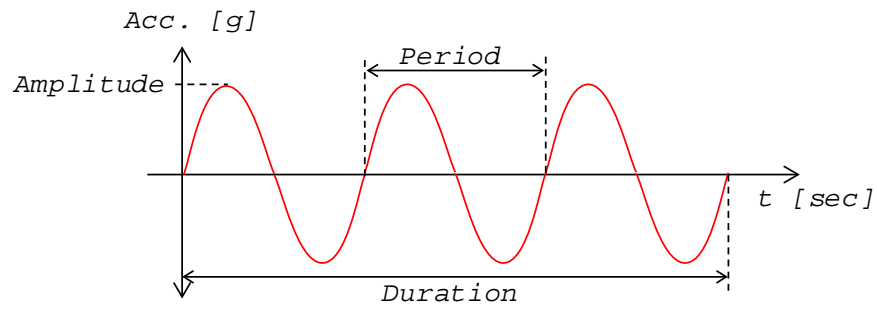Figure 14. Ground motion record data file formats

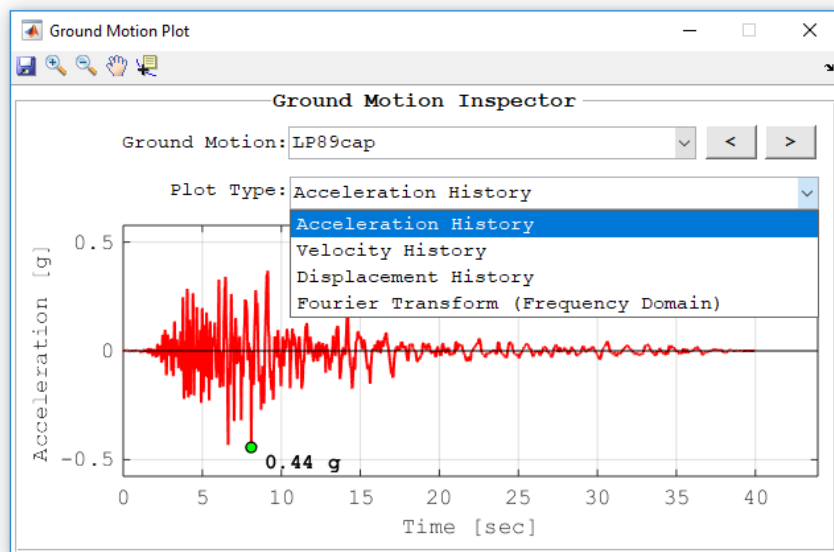Figure 15. Definition parameters of a harmonic wave

Figure 16. Ground motion plot

2.5 Analysis Type Module

The Analysis Type module is shown in Figure 17. The current version of IIDAP includes four types of dynamic analysis procedures. These are summarized as follows.

2.5.1 Response History Analysis (RHA)

This is the simplest form of dynamic analysis provided by IIDAP. The defined SDoF system is analyzed one time using the defined ground motion multiplied by a specified scale factor as discussed above. No additional parameters need to be specified by the user for this analysis type. Output results in this case include the relative/absolute/pseudo response-history quantities (displacement, velocity, acceleration, and force). This is discussed in more detail in Section 3.
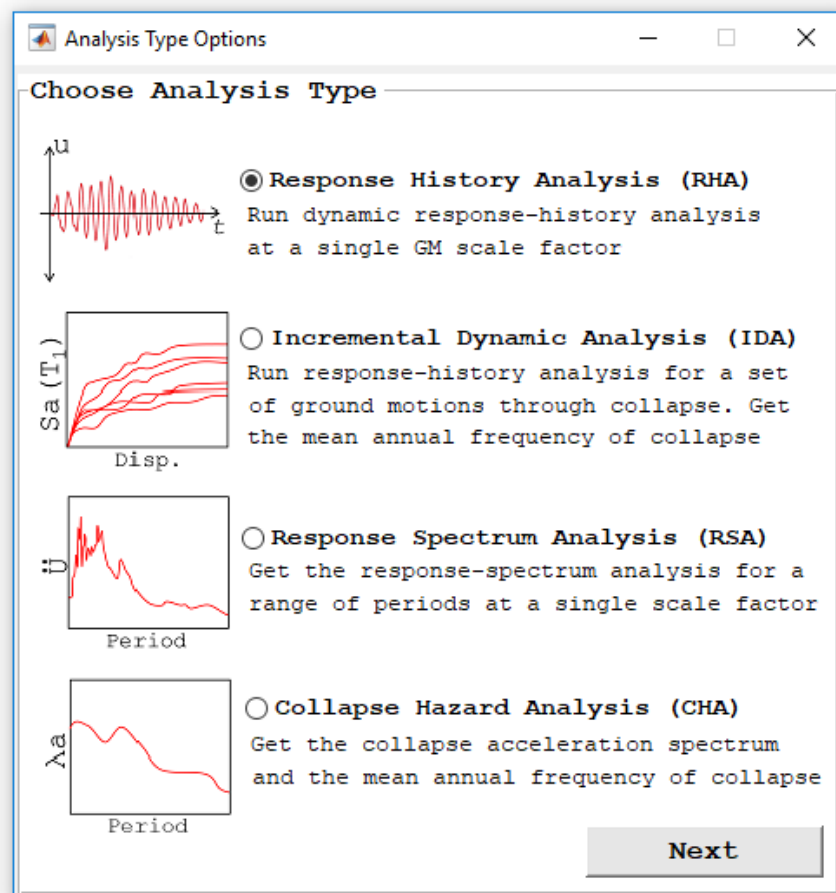

Figure 17. Analysis Type module

2.5.2   Incremental Dynamic Analysis (IDA)

This option performs incremental dynamic analysis (IDA) through collapse (Vamvatsikos and Cornell 2002). The IDA requires additional parameters to be specified by the user through the IDA Module shown in Figure 18a. These include the incremental spectral acceleration step, SA_{incr}, used for scaling the corresponding ground motion and the displacement/acceleration criteria that define structural collapse. The latter is mainly important for non-deteriorating system models (such as the Linear and Bi-Linear models). If one or both of those collapse criteria are to be ignored, the user can assign a large value, say 999, in the associated field. IIDAP traces the collapse point using the hunt and fill algorithm with a fixed spectral acceleration tolerance, SA_{tol}, equal to 0.01g. By default, a given ground motion is scaled using the spectral acceleration of the ζ%-damped elastic system at the given system period, T (as defined in the SDoF module); i.e., $Sa(T, \zeta\%)$. The user has the option to use the average spectral acceleration within a specified period range as an intensity measure; i.e., $Sa_{avg}(aT \sim bT, \zeta\%)$, in which a and b are positive scaler coefficients that define the start and end of the period range (Bojórquez and Iervolino 2011; Eads et al. 2015). This intensity measure can be selected by checking the optional Sa_{avg} checkbox in the IDA module (see Figure 18a) and specifying the period range coefficients. Sa_{avg} is then calculated from the elastic spectrum as follows:

$$Sa_{avg}\left(T_1 \sim T_n\right) = \frac{\sum_{i=1}^{i=n} Sa(T_i)}{n} \qquad , n = \frac{T_2 - T_1}{0.1\text{sec}} \tag{5}$$

2.5.3   Response Spectrum Analysis (RSA)

The SDoF system is analyzed several times using the specified ground motion(s) so as a response spectrum analysis is conducted. Each time, the system's period is changed. For this analysis type, the user only needs to specify the period range and increment through the RSA

module shown in Figure 18b. To conduct the RSA, three "analysis methods" are available to choose from using the popup menu shown in Figure 18b:

- **Method 1, "Vary Mass":** For each analysis, the SDoF elastic stiffness is kept constant as originally defined while the SDoF mass, M, as well as the damping ratio, C, are recomputed each time based on the original stiffness, K, and the period being analyzed.

- **Method 2, "Vary stiffness but maintain F_y":** In this method, the SDoF mass, M, as originally defined, is kept constant while the system's elastic stiffness and the yield displacement are changed as illustrated in Figure 19c. Consequently, the stability coefficient θ is changed based on the new K_e. In this method, the yield force remains constant as well as the rest of the previously defined backbone parameters. Therefore, the backbone curve is shifted based on the updated yield displacement (see Figure 19c).

- **Method 3, "Vary Stiffness but maintain δ_y":** In this method, the SDoF mass, M, as originally defined, is kept constant while the system's elastic stiffness and yield strength vary as illustrated in Figure 19b. Consequently, the stability coefficient θ varies based on the new K_e. In this method, the yield displacement and the rest of the previously defined backbone parameters remain constant.

2.5.4 Collapse Spectrum Analysis (CSA)

This option performs collapse spectrum analysis to construct collapse spectra (Adam and Jäger 2012; Adam et al. 2017). This is mainly response spectrum analysis but the response quantity in this case is the collapse intensity, $Sa_{collapse}$. For a given period, IDA is conducted until the collapse point is reached. For this analysis type, the user shall define the IDA and RSA as discussed earlier (see Figure 18c). Because the collapse capacity $Sa_{collapse}$ is the only quantity of interest in this analysis, the user shall use a relatively large incremental acceleration step size to reduce the simulation time.

(a) Incremental Dynamic Analysis

(b) Response Spectrum Analysis

(c) Collapse Spectrum Analysis

Figure 18. The different analysis modules

(a) Vary M

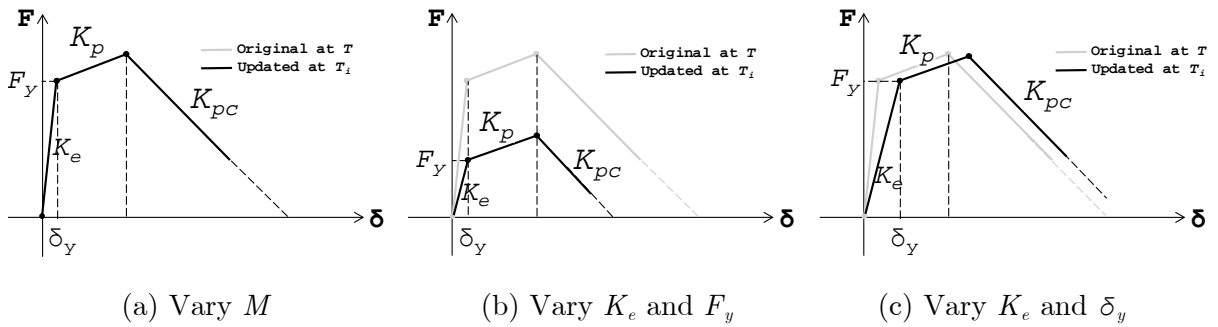(b) Vary K_e and F_y

(c) Vary K_e and δ_y

Figure 19. Illustration of the different RSA options

2.6 Seismic Hazard Module

For both IDA and CSA analyses, if the number of defined ground motion records is larger or equal to 3, the user will be prompted to the message window shown in Figure 20a to define a seismic hazard curve. If this optional step is defined by the user, the mean annual frequency of collapse, λ_c, is computed by numerically integrating the collapse fragility curve over the user-defined hazard curve using Equation 6 (Eads et al. 2013; Ibarra et al. 2002; Medina and Krawinkler 2003), in which, $P_c|Sa$ is the probability of collapse at a given spectral acceleration and $d\,\lambda_{Sa}(Sa)/d(Sa)$ is the slope of the seismic hazard curve given this spectral acceleration. Furthermore, assuming that the earthquake occurrence follows a Poisson distribution in time, the probability of collapse in n years, $P_c(n\ years)$, is computed using Equation 7 by assuming that earthquakes follow a Poisson distribution.

$$\lambda_c = \int_0^\infty \left(P_c\big|Sa\right).\left|\frac{d\lambda_{Sa}\left(Sa\right)}{d(Sa)}\right|.d\left(Sa\right) \tag{6}$$

$$P_c\left(n\ years\right) = 1 - \exp\left(-\lambda_c\ n\right) \tag{7}$$

The Seismic Hazard module is shown in Figure 20b. The user has two options to define the seismic hazard curve:

- **Option 1:** Import the hazard curve data directly from a text file. This file shall contain two columns of data (with no headers). The first column is the spectral acceleration in units of g while the second column is the mean annual frequency of exceedance. Such curves can be computed through conventional probabilistic seismic hazard analysis (e.g. Iervolino et al. (2016) http://wpage.unina.it/iuniervo/doc_en/REASSESS.htm). It is important to note that the spectral acceleration provided by the user should

correspond to the priory selected intensity measure within the IDA or the CSA modules (i.e., $Sa(T)$ or Sa_{avg}).

- **Option 2:** for buildings located in the US, the user can utilize the 2008 updated seismic hazard maps (Petersen et al. 2008), which are integrated in the toolbox, to directly extract the hazard curves. User must be aware that these hazard curves are based on the intensity measure $Sa(T)$ and not Sa_{avg}. For this option, the user shall browse for the hazard maps data folder named "USGS Hazard MAT Data" that is provided within IIDAP supporting files. The user also need to specify the *Latitude* and *Longitude* coordinates, the hazard period T_{hazard} (can be different from the SDoF period), and the shear wave velocity of the soil *Vs30* (i.e., soil type) as shown in Figure 20b. The discrete data points of the seismic hazard curve are then fitted internally with a fourth order polynomial function (Eads et al. 2013). This polynomial function is used for the numerical integration of the hazard curve. Note that currently the integrated USGS hazard maps cover the following ranges:

- $24.6^\circ \leq Latitude \leq 50^\circ$

- $-125^\circ \leq Longitude \leq -100^\circ$

- $0.0 \quad \sec \leq T_{hazard} \leq 5.0 \sec$

- $180 \text{ m/sec} \leq Vs30 \leq 760 \text{ m/sec}$
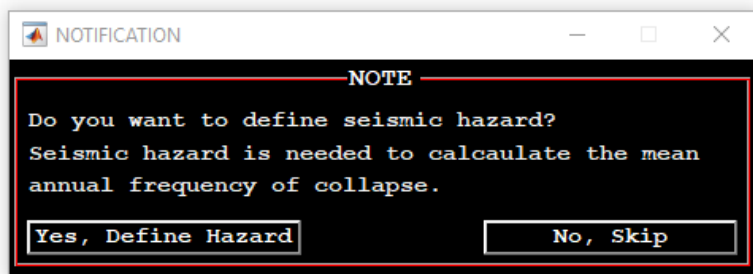
(a)

(b)

Figure 20. (a) Seismic hazard message; (b) Seismic Hazard module

2.7 Numerical Integrator Module

Before running the analysis, the user is prompt to the window message shown in Figure 21a. The user is informed that, by default, IIDAP runs dynamic analysis using the Newmark-Constant acceleration method. The user is given the option to control the integrator type and the integration time step. This is done through the Integrator module shown in Figure 21b. Three integrators are available for this purpose,

1. Recurrence formulas: only available for linear systems.

2. Central difference method

3. Newmark: this is the default integrator used by IIDAP. For this integrator, the user need to specify the integrator constant, γ and β. By default, IIDAP uses the "constant acceleration" method (i.e., $\gamma=0.5$ and $\beta=0.25$).

When selecting an integrator, IIDAP automatically computes the critical time step that ensures stability/accuracy of the integration. This time step is taken as the minimum of the ground motion time step and the integrator critical time step. The user can override this critical time step and specify a smaller time step.
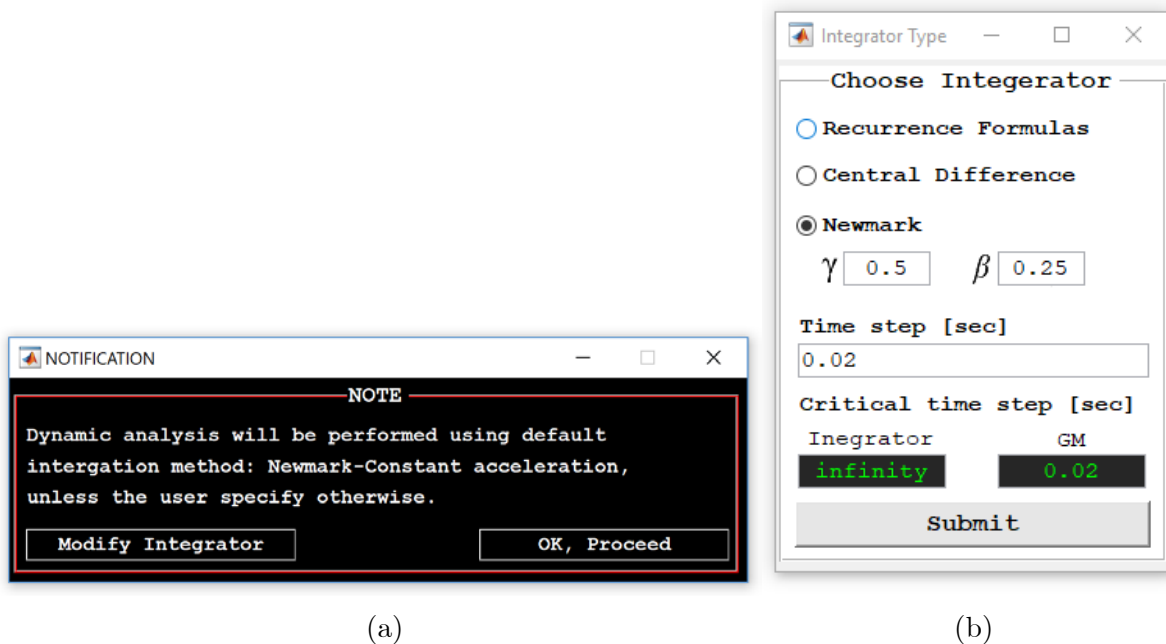

(a) (b)

Figure 21. (a) Integrator options message; (b) Integrator module

3. IIDAP Results Visualization Using Illustrative Examples

3.1 Response history of a damped linear elastic SDoF

In this example a SDoF with linear elastic damped response is analyzed under the El Centro record (North-south horizontal component at the Irrigation District substation) of the 1940 Imperial Valley earthquake. This ground motion record is commonly used in textbooks of structural dynamics (e.g., Chopra (2017)). This ground motion record is also included with the IIDAP support documents/files under the name "Sample GM – ElCentro.txt". The summary description of this project is as follows:

- **SDoF:** T= 2 seconds, M= 1 kN.sec^2/m, K= 9.87 kN/m, and ζ= 2%. System P-Delta is neglected.

- **System Model:** Linear (no limit on force or displacement).

- **Ground Motion:** Single GM: El Centro, Imperial Valley. Scale factor=1.0. No free vibration time.

- **Analysis Type:** Response History Analysis (RHA).

- **Integrator:** Default: Newmark constant acceleration (γ=0.5 and β=0.25) with $\Delta t_{integration}$=Δt_{GM}.

After defining the four components of the project, the "Run Analysis" button becomes active. By clicking this button and choosing the default integrator, the analysis is executed and subsequently the "View Results" and "Save Results" buttons become active. For this type of analysis (i.e., RHA), clicking the "View Results" button opens the results-plots-options window shown in Figure 22. There are six plot options:

- **"RHA Plots":** This option shows, in a single figure, the plots of the response history for a given response quantity as shown in Figure 23a. The user can use the dropdown

menus or the previous/next buttons to scroll between the different ground motion records and response quantities. Seven response quantities can be plotted: the relative displacement, relative velocity, relative acceleration, absolute acceleration, pseudo velocity, pseudo acceleration, and force. Note that the pseudo velocity and acceleration are as follows.

$$V_{pseudo} = U_{relative}\omega \tag{8a}$$

$$A_{pseudo} = U_{relative}\omega^2 \tag{8b}$$

The maximum absolute value of each quantity is superimposed in those plots. Note that from the toolbar at the top-left side of the window, the user can: 1) inspect the response values at any time step using the toolbar data cursor icon "⬚", 2) zoom in or out a specific region in the plot using the zoom controls "🔍 🔍", and 3) save the whole plot as *.png file using the save button "💾". Using the "save plot data" button at the bottom-right corner of the window, the user can save the x and y data of the current plot, under a name and location of choice.

- **"Composite RHA Plots":** This option is similar to the "RHA Plots" except that, in this one, four plots of the force, relative displacement, relative velocity, and absolute acceleration, are shown in a single figure.

- **"Force-Displacement Response":** This option shows a plot of the force-displacement response as shown in Figure 23b. In the same plot, the backbone curve will be superimposed.

- **"Animation Deformation Response" and "Animation Force-Displacement Response":** These two options provide dynamic animation plots of the SDoF deformation and force-displacement responses as shown in Figure 23c and d, respectively. From these

31

plots, the user can play/pause and speed up/slow down the animation as desired. Note that animation can be only played for one plot at a time.

- **"Animation: Combined":** This option shows the dynamic animation plots of the SDoF deformation and force-displacement response in single figure, where they can be synchronized.
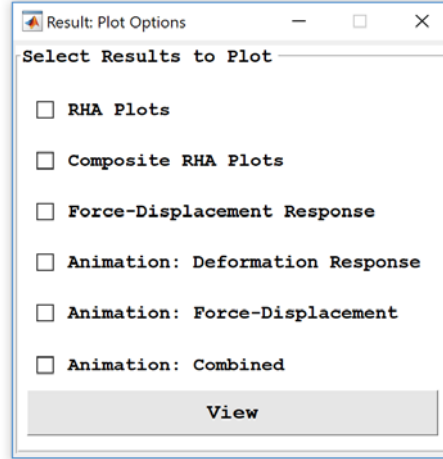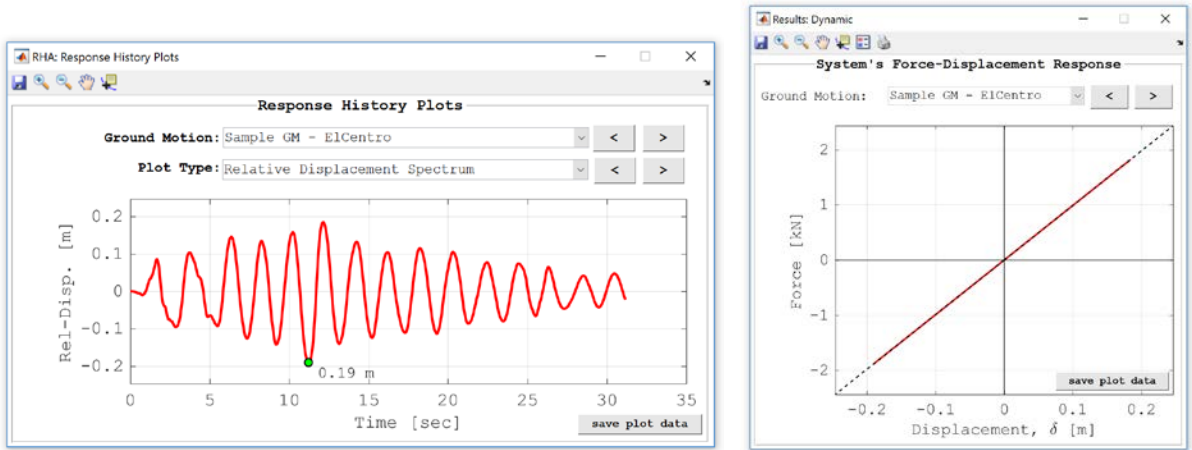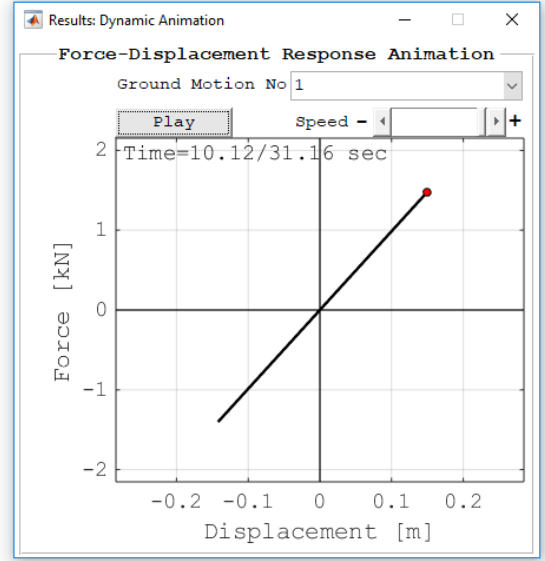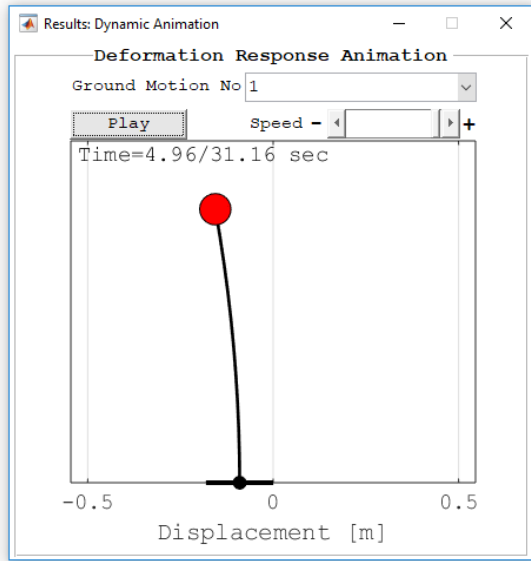

Figure 22. RHA-results visualization-options window

(a) Response history plot   (b) Force-displacement response plot

(c) Deformation animation plot (d) Force-displacement animation plot

Figure 23. Response history analysis plots

3.2 Response spectrum analysis of a damped linear elastic SDoF

The previous project is re-analyzed using the response spectrum analysis procedures. The following parameters are specified:

- **Analysis Type:** RSA: T_{start}= 0.0 sec, T_{incr}= 0.01 sec, T_{end}= 5.0 sec. Analysis method is "Vary Mass", although this does not affect the outcome of this analysis since the system is Linear.

For RHA, clicking the "View Results" button opens the results-plots-options window shown in Figure 24. There are two plot options:

- **"RSA Plots":** This option shows the response spectra of the different response quantities as shown in Figure 25a. The user can switch between the different ground motion records and response quantities using the popup menus at the top. Types of spectra that can be plotted include the "relative" displacement/velocity/acceleration spectra, the force spectrum, the "absolute" acceleration spectrum, as well as the "pseudo" velocity and acceleration.

- **"Force-Displacement Response at Ti":** This option shows a plot of the system's force-displacement response at each analyzed period as shown in Figure 25b. The user may use the slider control to scroll between the different period.
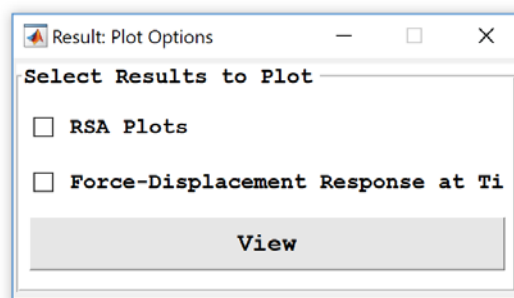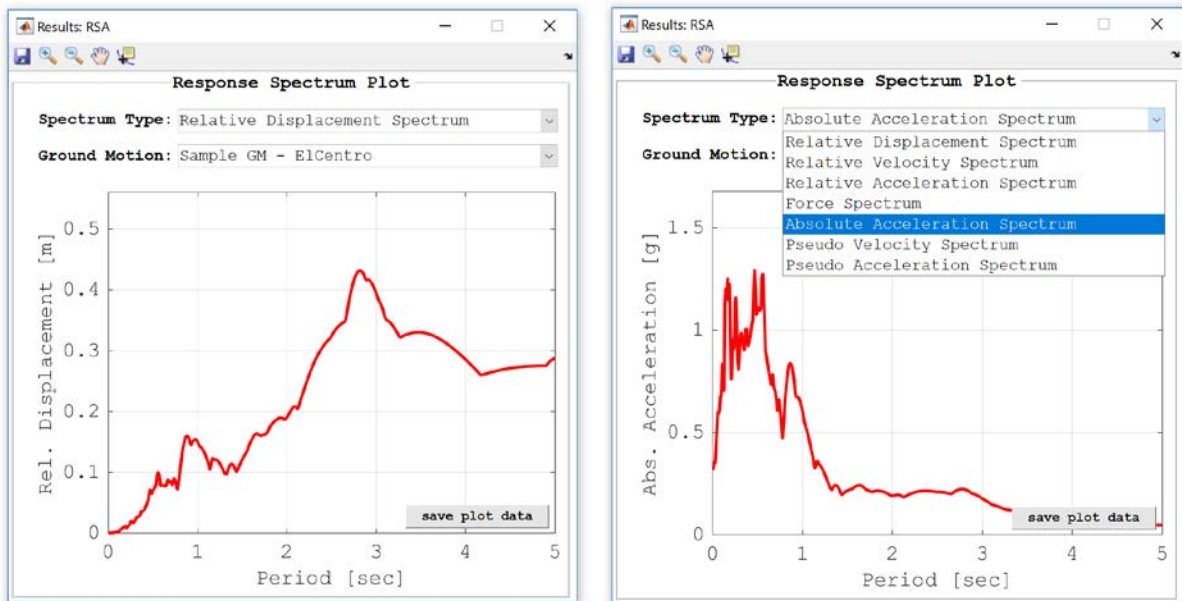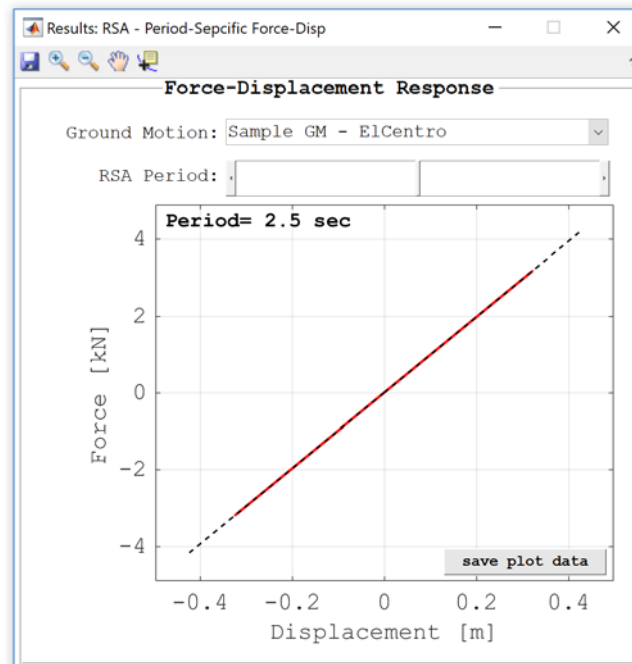

Figure 24. RSA-results visualization-options window

(a) Response spectrum plots

(b) Force-displacement plot a given period

Figure 25. Response spectrum analysis plots

3.3 Incremental dynamic analysis of a damped SDoF with IMK-Bilin model

In this example a SDoF with bi-linear damped response is analyzed to collapse under a suite of ground motions. The summary description of this project is as follows:

- **SDoF:** T= 1 seconds, M= 10 kN.sec^2/m, K= 394.8 kN/m, and ζ= 2%. System P-Delta is considered with a stability coefficient $\theta = 0.06$.

- **System Model:** IMK-Bilin: F_y/W=0.2 ((i.e., F_y=20 kN and δ_y=0.05 m); F_{max}/F_y =1.05; F_{res}/F_y =0.2; δ_p/δ_y =2; δ_{pc}/δ_y =8; δ_u/δ_y =10. Default deterioration parameters are used (λ=50 and c=1.0). No curve smoothing is considered.

- **Ground Motion:** Multiple GMs: the Far-Field ground motion set (44 records) specified in FEMA (2009). This set is included in the IIDAP support files within the folder named "GroundMotions-FarField". Scale factor=1.0. No free vibration time.

- **Analysis Type:** IDA with an increment of $SA_{incr} = 0.1$ g. The displacement limit for collapse, $\delta_{collapse}$, is set equal to the specified ultimate displacement of the system model, δ_u. An acceleration limit, $SA_{collapse}$, of 5 g is also specified. The average spectral acceleration $S_{a,avg}(T_{range}, \zeta\%)$ is used as an intensity measure with a period range of 0.3 to 2.0 seconds.

- **Seismic Hazard:** USGS Seismic Hazard Maps: *Latitude*: 33.996º, *Longitude*= -118.162 º, T_{hazard}= 1.0 sec, and *Vs30*= 259 m/sec.

- **Integrator:** Default: Newmark constant acceleration (γ=0.5 and β=0.25) with $\Delta t_{integration}=\Delta t_{GM}$.

For IDA, clicking the "View Results" button will open the results-plots-options window shown in Figure 26. There are four plot options:

- **"IDA Curves":** This option shows the IDA curve plot (i.e, spectral acceleration at SDoF period versus displacement) as shown in Figure 27a. The user can plot the IDA

curve for a single ground motion or for all ground motions using the dropdown menu at the top. In this plot, the collapse point(s) is highlighted by a red-circle marker. In the same figure, the median IDA curve as well as the 16th and 84th percentile curves are superimposed in red lines.

- **"Displacement Fragility at Target SA":** For a given spectral acceleration, this plot shows the probability of reaching or exceeding a given displacement as shown in Figure 27b. In this plot, the user may specify any target spectral acceleration, SA_{target}, of interest using the slider controller at the top of the window. IIDAP will subsequently find the system displacement at SA_{target} for each ground motion. The empirical cumulative probability distribution is then calculated for these displacement values and fitted with a lognormal cumulative distribution function. The fitted function is plotted as a red line. The population parameters of the fitted function (median and dispersion) are annotated at the top-left corner of the plot. It should be noted, that only the ground motions that reached SA_{target} are considered in this plot. The number of these ground motions are annotated as well in the top-left corner of the plot.

- **"Incremental Force-Displacement":** This option shows a plot of the system's force-displacement response at any incremental step as shown in Figure 27c. In this plot, the user can examine the system response at any incremental step for any ground motion. The user can go through the different increments using the slider control at the top of the window. The values of the spectral acceleration and the ground motion scale factor for the current increment are annotated at the top-left corner of the plot.

- **"Collapse Fragility Curve":** This option is only active if the number of ground motion records is larger than or equal to three. This option shows a plot of the collapse fragility curve (i.e., probability of collapse versus spectral acceleration) as shown in Figure 27d. This plot shows the empirical cumulative probability distribution scatter of the collapse

intensities (marked by grey-circles). These scatter point are fitted with a lognormal cumulative distribution function which is plotted in a red line. The population parameters of the fitted lognormal distribution (median and dispersion) are annotated on the top-left corner of the plot. If seismic hazard was defined, as in this example, the mean annual frequency of collapse, λ_c, and the corresponding probability of collapse (P_c) in n years are calculated (see Section 2.5) and displayed on the plot. The value of P_c is calculated and updated based on the n value specified by the user at the lower right corner of the plot. In this example, $\lambda_c = 2.27 \times 10^{-4}$/year and $P_c = 1.13\%$ in 50 years.
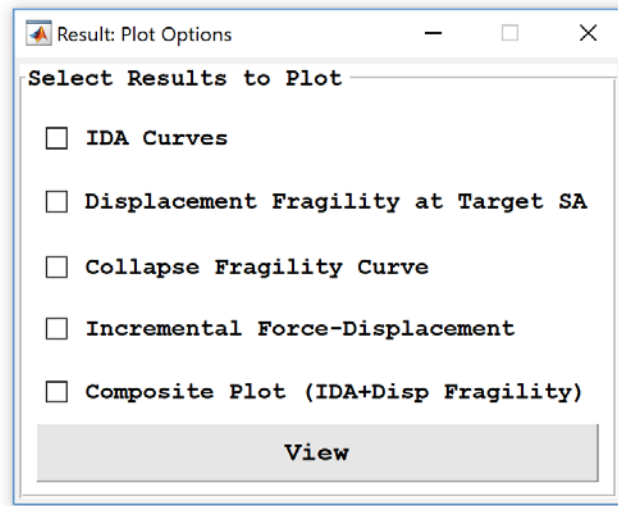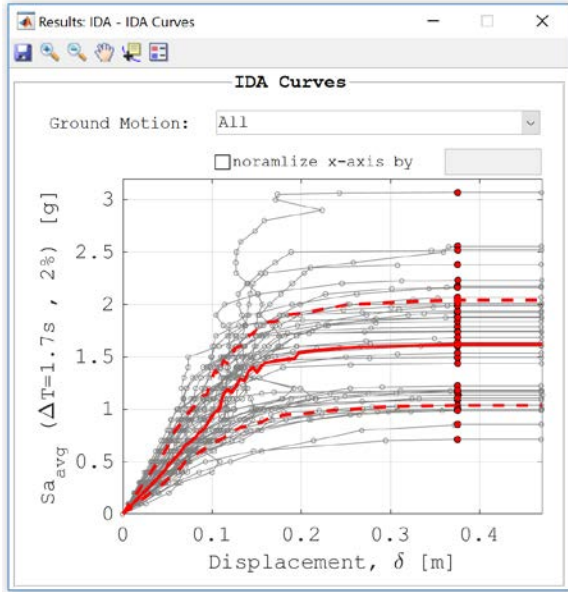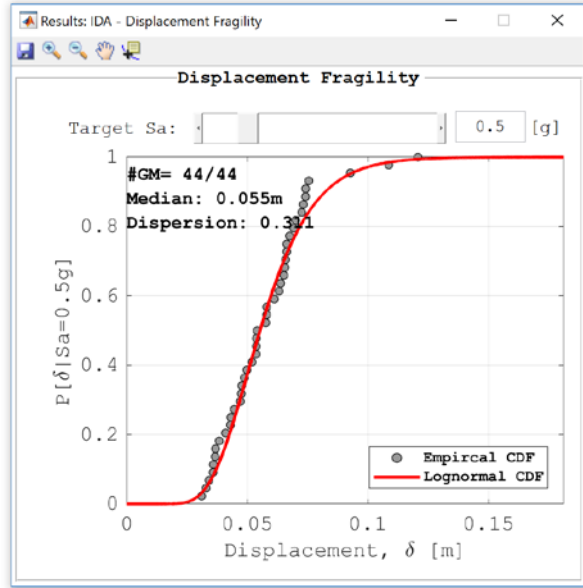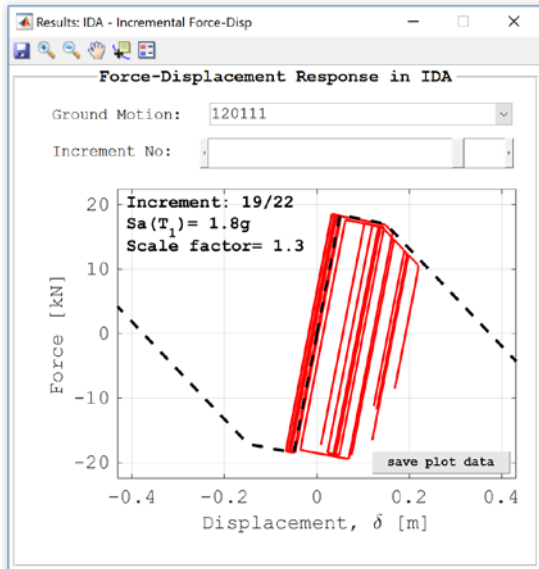

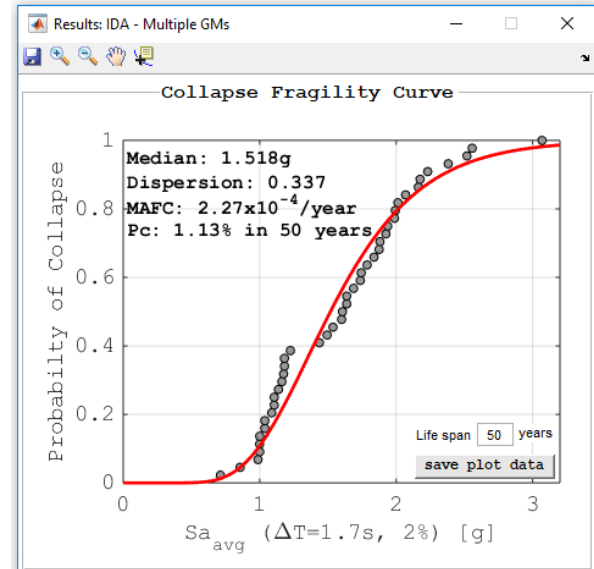Figure 26. IDA-results visualization-options window

(a) IDA curves

(b) Displacement fragility at a specific Sa

(c) Force-Displacement at SA increments

(d) Collapse fragility curve

Figure 27. Incremental dynamic analysis plots for multiple ground motions

4. Saving Results

After running one type of analysis, the "Save Results" button becomes active. When this button is clicked, a folder named "Results - *ProjectName*" is created at the project directory. Inside this folder, text files containing the analysis results are saved. The saved data depend on the analysis type. The descriptions of the different results files are summarized below:

- **For any analysis type,** a file named "Project_Summary_Data.txt" is saved. This file contains a summary of the project parameters similar to those shown in the project summary window when the project "scope" button is clicked (see Figure 3).

- **For RHA,** the response history of the SDoF system for each ground motion i is saved under the name "Response History *GMi*.txt", where *GMi* is the name of the ground motion record. This files contain 8 columns of data. The first column contains the ground motion time value while the remaining columns contain the response quantities for the ground acceleration, relative displacement/velocity/acceleration, pseudo velocity/acceleration, absolute acceleration, and force.

- **For RSA analysis,** the response spectra of the SDoF system for each ground motion i are saved under the name "Response Spectra *GMi*.txt". This file contains 8 columns of data. The first column contains the period values while the remaining columns contain the response quantities for the relative displacement/velocity/acceleration, pseudo velocity/acceleration, absolute acceleration, and force.

- **For IDA analysis**, the IDA curve for each ground motion i is saved under the name "IDA Curve *GMi*.txt". This files contains two columns of data. The first column includes the relative displacement values while the second column contains the incrementally increasing spectral acceleration values. If the number of ground motion would be larger than or equal to three, the collapse fragility data points would be saved

under the name "Collapse Fragility Curve - Empirical.txt". This file contains two columns of data. The first column contains the spectral intensities at collapse for each ground motion (arranged in ascending order) while the second column includes the empirical probability of collapse values. Additionally, the data points of the fitted lognormal collapse fragility curve is saved under the name "Collapse Fragility Curve – Fitted Lognormal.txt". The first two rows of this file contains the values of the median collapse capacity and dispersion of the fitted lognormal curve. The remainder of the file contains two columns of data. The first column contains the incrementally increasing spectral intensity while the second column includes the probability of collapse based on the fitted lognormal cumulative distribution function.

5. Indicative Simulation Time

The table below summarizes the approximate simulation time for performing different types of analyses in IIDAP. The simulation time is based on a 64-bit Windows machine with i7 (4GHz) CPU, 32 GB of RAM and an SSD disk.

	Simulation Time [sec]	
Analysis Type	Single Ground Motion (30s record)	Multiple Ground Motions (40 records)
RHA	2	7
IDA ($\approx$7 increments to collapse)	4	190
RSA (10 periods)	5	250

6. References

Adam, C., and Jäger, C. (2012). "Seismic Collapse Capacity of Basic Inelastic Structures Vulnerable to the P-delta Effect." *Earthquake Engineering & Structural Dynamics*, 41(4), 775-793.

Adam, C., Kampenhuber, D., and Ibarra, L. F. (2017). "Optimal Intensity Measure Based on Spectral Acceleration for P-Delta Vulnerable Deteriorating Frame Structures in the Collapse Limit State." *Bull Earthquake Eng*, 1-25.

Bojórquez, E., and Iervolino, I. (2011). "Spectral Shape Proxies and Nonlinear Structural Response." *Soil Dynamics and Earthquake Engineering*, 31(7), 996-1008.

Chopra, A. K. (2017). *Dynamics of structures: Theory and applications to Earthquake Engineering*, Pearson Prentice Hal.

Eads, L., Miranda, E., Krawinkler, H., and Lignos, D. G. (2013). "An Efficient Method for Estimating the Collapse Risk of Structures in Seismic Regions." *Earthquake Engineering & Structural Dynamics*, 42(1), 25-41.

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.

FEMA (2009). "Quantification of Building Seismic Performance Factors." Federal Emergency Management Agency, Washington, DC.

Ibarra, L., Medina, R., and Krawinkler, H. (2002). "Collapse Assessment of Deteriorating SDOF Systems." *Proc., 12th European Conference on Earthquake Engineering (Collapse Assessment of Deteriorating SDOF Systems)*, London, UK, 9-13.

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.

Iervolino, I., Chioccarelli, E., and Cito, P. (2016). "REASSESS V1. 0: A Computationally-Efficient Software for Probabilistic Seismic Hazard Analysis." *Proc., VII European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS)*, Crete Island, Greece, 5-10.

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.

Medina, R. A., and Krawinkler, H. (2003). "Seismic Demands for Nondeteriorating Frame Structures and their Dependence on Ground Motions." The John A. Blume Earthquake Engineering Center, Stanford University, California, USA.

Petersen, M. D., Frankel, A. D., Harmsen, S. C., Mueller, C. S., Haller, K. M., Wheeler, R. L., Wesson, R. L., Zeng, Y., Boyd, O. S., and Perkins, D. M. (2008). "Documentation for the 2008 Update of the United States National Seismic Hazard Maps." Geological Survey (US).

Vamvatsikos, D., and Cornell, C. A. (2002). "Incremental Dynamic Analysis." *Earthquake Engineering & Structural Dynamics*, 31(3), 491-514.