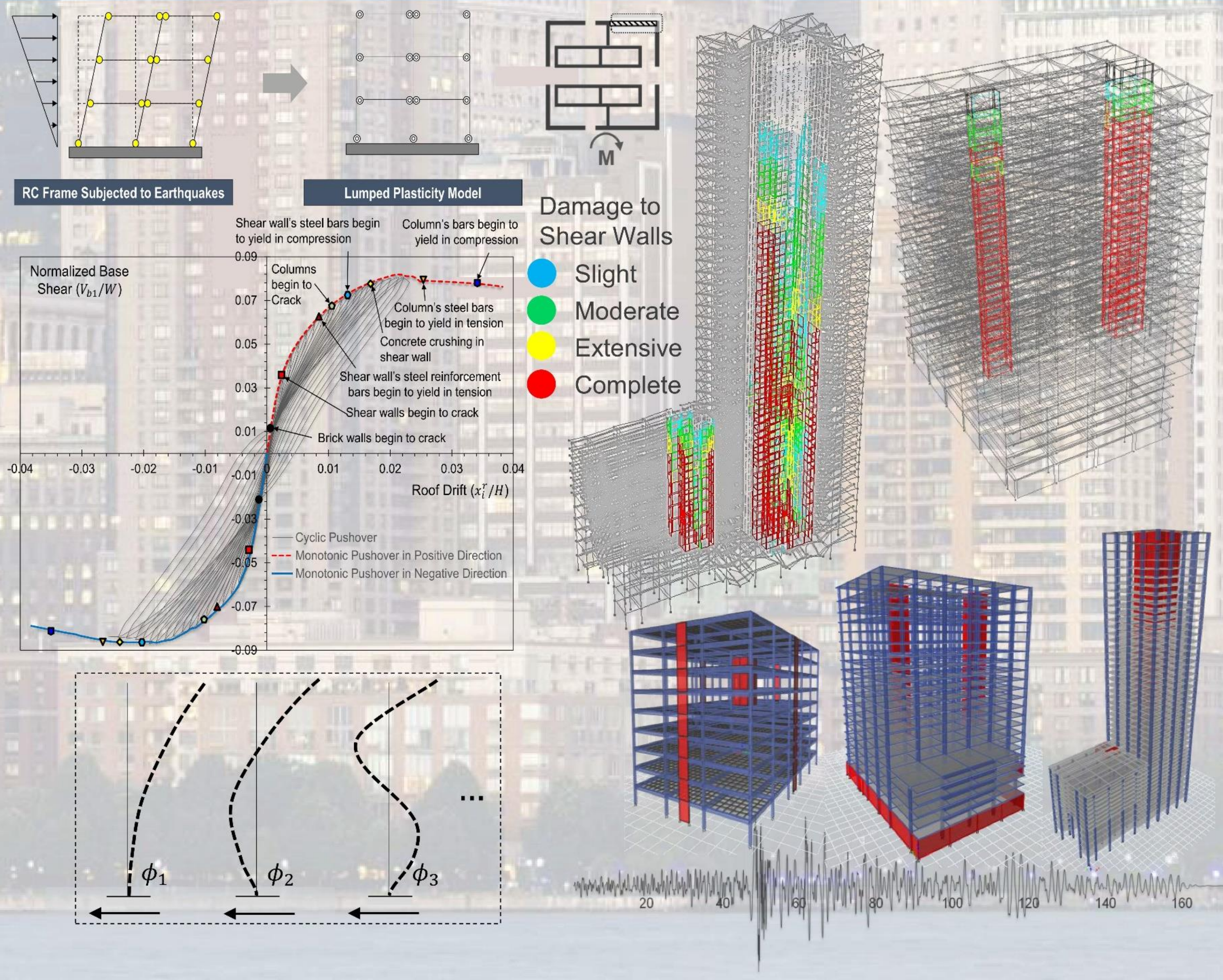
NONLINEAR MODELLING AND ANALYSIS OF RC BUILDINGS USING ETABS (v 2016 and onwards)

[Document Version 0]

This document compiles the basic concepts of inelastic computer modelling and nonlinear analysis of building structures. It also presents a step-by-step methodology to construct the nonlinear computer models of RC building structures (for their detailed performance evaluation) using CSI ETABS 2016.



Nonlinear Modelling and Analysis of RC Buildings using ETABS (v 2016 and onwards)

This document compiles the basic concepts of inelastic computer modelling and nonlinear analysis of building structures. It also presents a step-by-step methodology to construct the nonlinear computer models of RC building structures (for their detailed performance evaluation) using CSI ETABS 2016.

Compiled by

Hamza Mazhar
Fawad Ahmed Najam
Luqman Ahmed
Hafiz Zain Akram

NUST Institute of Civil Engineering (NICE)
National University of Sciences and Technology (NUST)
H-12 Islamabad, Pakistan
Cell: 92-334-5192533, Email: fawad@nice.nust.edu.pk
Office No: 118, 1st Floor, NIT Building, SCEE, NUST

27 March 2021

Acknowledgement

The material compiled in this document is mostly taken from the following references. It is intended to be used only for the educational purposes. All these sources are duly acknowledged and cited. No infringement of copyrights or commercial activity is intended through this document.

- CSI Analysis Reference Manual (SAP 2000, ETABS and CSI Bridge), Computers and Structures Inc., USA.
- Seismic Evaluation and Retrofit of Existing Buildings, ASCE/SEI 41-17 (formerly FEMA 356), American Society of Civil Engineers.
- Graham H. Powell (2010) Modeling for Structural Analysis, Computers and Structures Inc., ISBN-10: 0923907882.
- Modeling and Acceptance Criteria for Seismic Design and Analysis of Tall Buildings, PEER/ATC 72-1, Applied Technology Council and Pacific Earthquake Engineering Research Center, 2010.
- An Alternative Procedure for Seismic Analysis and Design of Tall Buildings Located in the Los Angeles Region, 2017 Edition with 2018 Supplements, Los Angeles Tall Buildings Structural Design Council, March 20, 2018.
- Experimental Evaluation and Analytical Modeling of ACI 318-05/08 Reinforced Concrete Coupling Beams Subjected to Reversed Cyclic Loading, UCLA-SGEL Report 2009/06
- Design Recommendations for Steel-reinforced Concrete (SRC) Coupling Beams, UCLA-SGEL Report 2013/06.

Table of Contents

List of Figures

Acronyms and Abbreviations

ASCE	American Society of Civil Engineers
CP	Collapse Prevention
DBE	Design-basis Earthquake
IBC	International Building Code
IO	Immediate Occupancy
LS	Life Safety
MCE	Maximum Considered Earthquake
MW	Moment Magnitude
PEER	Pacific Earthquake Engineering Research Center
PGA	Peak Ground Acceleration
PSHA	Probabilistic Seismic Hazard Analysis
SA	Spectral Acceleration
UBC	Uniform Building Code
UHS	Uniform Hazard Spectrum
USGS	U.S. Geological Survey

Summary

A computer model of a structure is a compromise between the real structure and its mathematical representation. For the purpose of structural design, an understanding of these models, their underlying assumptions and analysis procedures is very important in order to arrive at an adequate and efficient design solution. With the advent of performance-based seismic design methodology, the use of inelastic computer modeling and nonlinear analysis has rapidly increased in recent years. This document compiles the basic concepts of inelastic computer modelling and nonlinear analysis of building structures. It also presents a step-by-step methodology to construct the nonlinear computer models of RC building structures (for their detailed performance evaluation) using CSI ETABS 2016. For the purpose of an example demonstration, the following modeling scheme is followed in this document.

The RC girders are modelled with moment-rotation hinges at both ends. The ASCE 41-17 modelling parameters are used for this purpose. The RC shear walls are modelled with nonlinear (concrete and steel) fiber elements throughout their lengths. The RC columns are modeled as a combination of nonlinear fiber elements at plastic hinge zone and elastic frame element at mid-section. For concrete fibers, the Mander's stress-strain model is used with expected material strength properties. For the steel fibers, the bilinear elasto-plastic model is used with expected yield strengths. The shear and torsional responses of beams, columns and shear walls are modelled as elastic. The slabs are modeled using elastic thin shell elements. The mass of floors are lumped at each floor. No nonlinear action is considered in RC retaining walls. No effects of soil-structure interaction are considered and the base of all columns and shear walls are assigned with idealized fixed or hinge support conditions.

Chapter 1

Basic Approaches in Nonlinear Modeling of Buildings

1.1. The Need for Nonlinear Modeling of Structures

Over last few decades, the structural design against earthquakes has passed through a continuous process of evolution. The story which started from a simple mass-proportional lateral load resisted by elastic action has now evolved into an explicit consideration of design earthquakes applied to the detailed nonlinear finite-element models. The exponential growth in computational power in recent years is continuously narrowing the industry-academia gap by providing the cutting-edge research and technology to practicing engineers at their doorstep. As a result, the structural designers nowadays are equipped with far more aids and tools compared to a couple of decades ago. Moreover, recent advancements in nonlinear modeling techniques have also opened a whole new research area dealing with constructing computer models with close-to-real behaviors. With such a range of options available, the choice of modeling scheme and the analysis procedure for design decision-making often becomes a matter of "*the more the sweat; the more the reward*" for designer.

Nonlinear modeling and analysis of complex structures (e.g., high-rise buildings with RC shear walls) is generally considered a difficult area in structural engineering practice due to many reasons. Firstly, it requires a detailed understanding of various complex interactions and phenomena (associated with individual inelastic components). Secondly, nonlinear analysis also demands significant computational effort and the use of specialized computer software. In some cases, the obtained results can be significantly sensitive to nonlinear modeling assumptions and inelastic properties of components which may not always be well-defined. An ordinary design office may not have necessary resources to undergo this process for each project. For most practical cases, the linear elastic analysis may serve the purpose of estimating design demands within their required degree of accuracy. However, with the advent of latest *"Performance-based Design (referred onwards as PBD)" methodology*, the need for nonlinear modeling and analysis is growing rapidly as the structural engineers are constantly trying to equip themselves with the latest technological advancements. The above-mentioned limitations are also diminishing with the development of latest seismic analysis solvers, software tools and guidelines (e.g., ASCE/SEI 41-06/13) which provide a significant help in understanding and implementing the nonlinear modeling of structural components.

The nonlinear model of a structure is capable of clearly identifying the structural damage and performance in terms of deformation demand-to-capacity ratios. The seismic simulation is more realistic and meaningful compared to a linear elastic model. It is, therefore, need of the hour to equip the next generation of structural engineers with this valuable tool so as to make them understand the complex inelastic structural behavior. As an example of how clearly the structural performance can be understood from the results of nonlinear analysis, Figure 1-1 presents an example of structural damage as obtained from the nonlinear response history analysis procedure. The damage in masonry infill walls and RC shear walls under an example ground motion is shown. The damage is characterized, and color coded by the strain demand-to-capacity ratios in individual elements.

This visual representation of material cracking or yielding or any other damage can provide a clear idea about the expected structural performance and condition at a certain earthquake level. These damage figures can be shown and made understandable even to clients and other stakeholders. Using such representations, architects, clients, designers, consultants and all related professionals can have a meaningful discussion in case of any conflict and can easily arrive at a compromise. *The designers can answer "what will happen, if…?" type questions from the building owners.* It is also possible to understand the progression of structural damage using a nonlinear analysis of building. As an example, Figure 1-2 presents the results obtained from the monotonic and reversed-cyclic pushover analysis of an example building (in its strong direction). The limit states achieved at different roof drift levels can be marked on pushover curves to conveniently understand the damage progression at the global structure level. These two examples indicate how effective are the results of nonlinear static or dynamic analysis in clearly understanding the complex inelastic response of building structures.

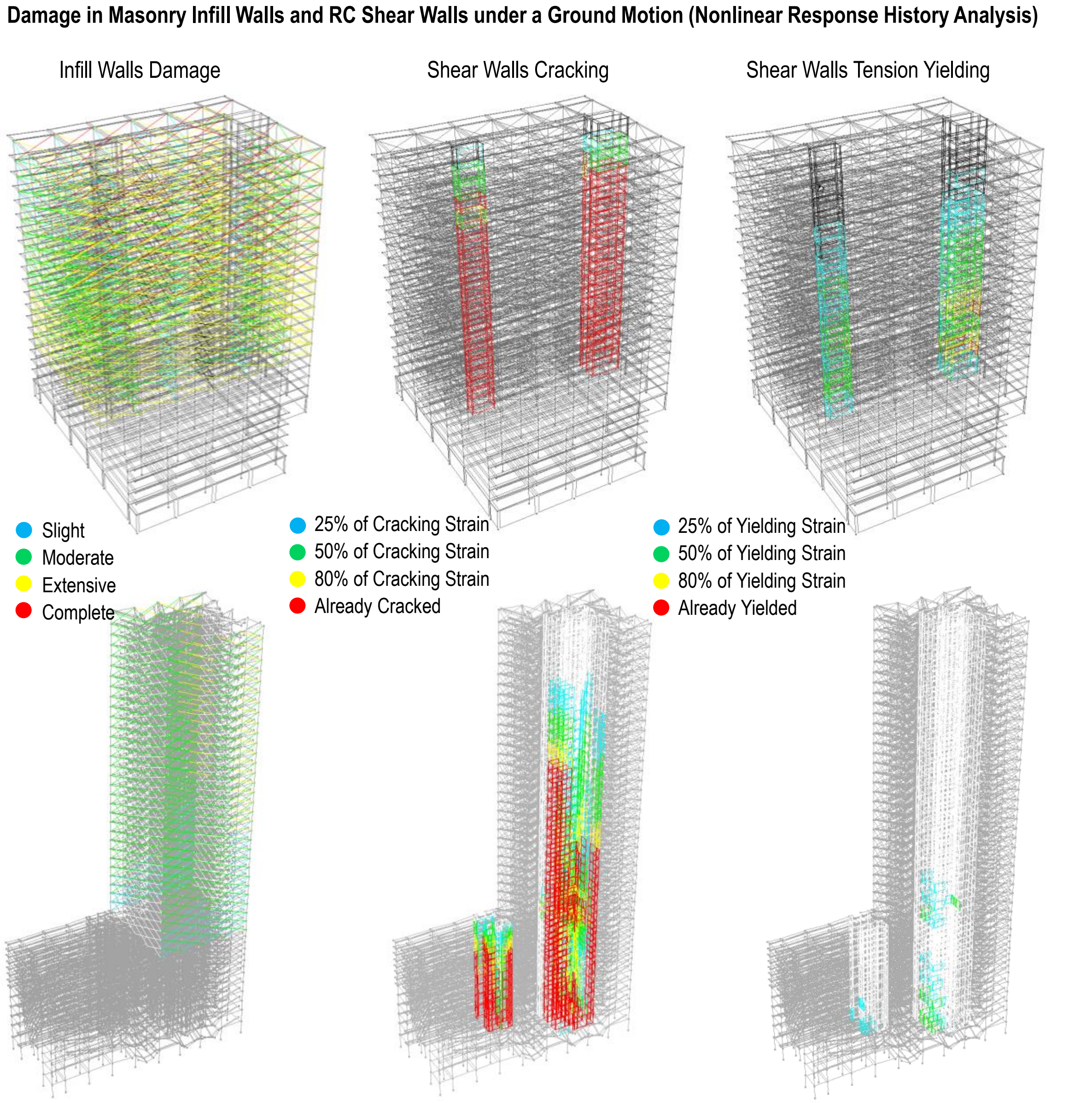


Figure 1-1: An example of structural damage characterized by the strain demand-to-capacity ratios as obtained from the nonlinear response history analysis procedure. The damage in masonry infill walls and RC shear walls under an example ground motion is shown.

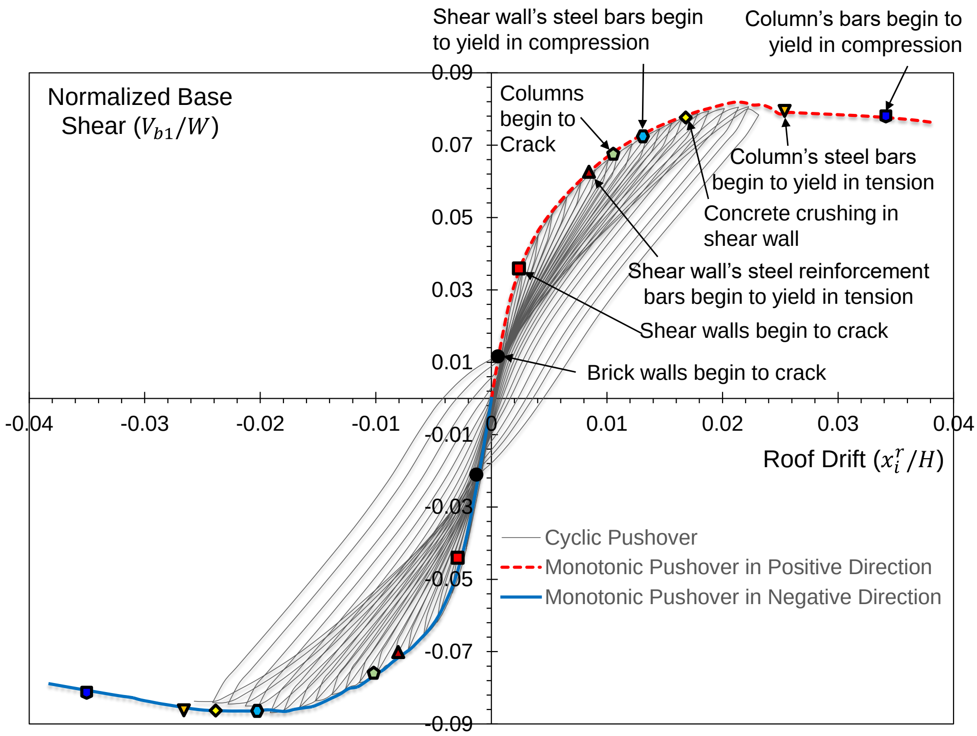


Figure 1-2: An example of the progression of structural damage as obtained from the monotonic and reversed-cyclic pushover analysis procedures.

1.2. Introduction to This Document

This document is compiled to provide an elementary tutorial for nonlinear modeling and nonlinear static and dynamic analysis of an RC buildings using a commercial software package ETABS 2016 (CSI 2016), a product of Computers and Structures Inc. (CSI) for structural analysis and performance assessment of structures. This document assumes that the reader is already familiar with the linear analysis and design of building structures and is well conversant with various modeling concepts used in linear modeling using ETABS (CSI 2016) or SAP 2000 (CSI 2006) etc. Generally, the step-by-step tutorials provide a systematic procedure of using a software without explaining the underlying theoretical concepts which are separately provided in technical manuals and documentation. *In this document, a mixed approach is used in which the step-by-step procedure will also be accompanied with a brief theoretical explanation of the process being conducted.*

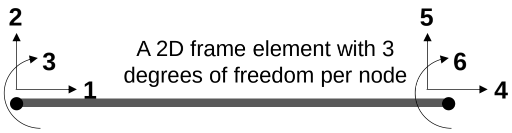
This document will make several references to the following documents. The readers are referred to these documents for detailed description of some of the concepts used in this document.

- Seismic Evaluation and Retrofit of Existing Buildings, ASCE/SEI 41-17 (formerly FEMA 356), American Society of Civil Engineers.
- Graham H. Powell (2010) Modeling for Structural Analysis, Computers and Structures Inc., ISBN-10: 0923907882.
- Modeling and Acceptance Criteria for Seismic Design and Analysis of Tall Buildings, PEER/ATC 72-1, Applied Technology Council and Pacific Earthquake Engineering Research Center, 2010.

- An Alternative Procedure for Seismic Analysis and Design of Tall Buildings Located in the Los Angeles Region, 2017 Edition with 2018 Supplements, Los Angeles Tall Buildings Structural Design Council, March 20, 2018.
- Experimental Evaluation and Analytical Modeling of ACI 318-05/08 Reinforced Concrete Coupling Beams Subjected to Reversed Cyclic Loading, UCLA-SGEL Report 2009/06
- Design Recommendations for Steel-reinforced Concrete (SRC) Coupling Beams, UCLA-SGEL Report 2013/06.

1.3. Basics of Nonlinear Modeling of Buildings

In order to understand the basic nonlinear modeling approaches, first a quick review of the composition of structural stiffness is useful. Let us take an example of a 2D elastic frame element. *The stiffness matrix of an elastic frame element is a function of four quantities (elastic modulus of its material, area and moment of inertia of its cross-sectional shape and its length)* [Figure 1-3].



$$K = f(\underbrace{E}_{\text{Material Property}}, \underbrace{A, I}_{\text{Cross-section Properties}}, \underbrace{L}_{\text{Member Property}})$$

$$K = \begin{pmatrix} EA/L & 0 & 0 & -EA/L & 0 & 0 \\ 0 & 12EI/L^3 & 6EI/L^2 & 0 & -12EI/L^3 & 6EI/L^2 \\ 0 & 6EI/L^2 & 4EI/L & 0 & -6EI/L^2 & 2EI/L \\ -EA/L & 0 & 0 & EA/L & 0 & 0 \\ & -12EI/L^3 & -6EI/L^2 & 0 & 12EI/L^3 & -6EI/L^2 \\ 0 & 6EI/L^2 & 2EI/L & 0 & -6EI/L^2 & 4EI/L \end{pmatrix}$$

Figure 1-3: The stiffness matrix of a 2D elastic frame element.

In general, the structural stiffness is a function of material stiffness, cross-sectional stiffness and member stiffness (Figure 1-4). The material stiffness is a function of material properties (e.g., the elastic modulus and Poisson's ratio). The cross-sectional stiffness depends on cross-sectional properties as well as the material stiffness. Similarly, the member stiffness depends on the member geometry as well as the cross-sectional stiffness. This composition of structural stiffness is shown in Figure 1-4.

In this formulation of structural stiffness, *the effects of nonlinearity can be introduced either right at material level, cross-section level or member level.* For the first case, the material properties (e.g., elastic modulus) can be specified as a variable to directly account for the effect of inelastic materials. The complete stress-strain curve of the material can be specified in such cases instead of only defining them with a constant elastic modulus. Alternatively, the inelastic effects can also be introduced at the cross-section and member levels by introducing special inelastic components (or elements) in the computer model.

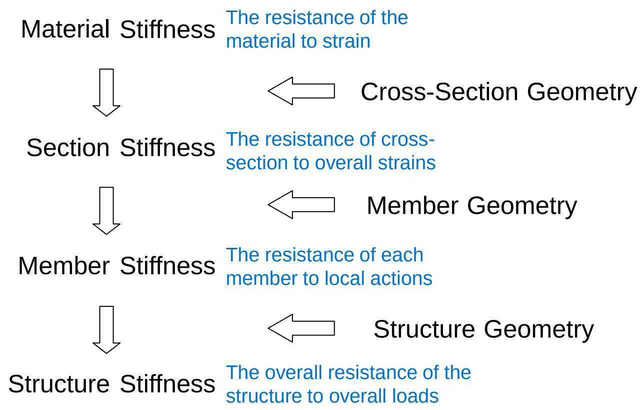


Figure 1-4: What is structural stiffness "made off"?

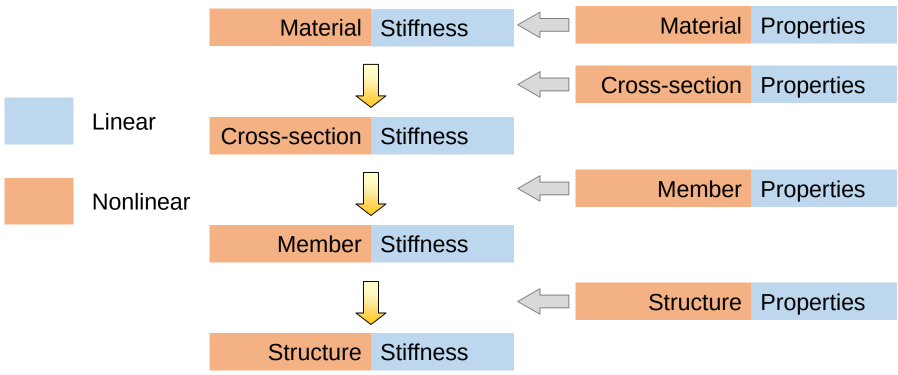


Figure 1-5: Linear and nonlinear properties defined at material, cross-section and member levels.

The nonlinear analysis aims to simulate all significant modes of deformation and deterioration in the structure from the onset of damage to complete collapse. Therefore, *unlike a linear elastic model, the nonlinear model of an RC structure should be able to capture all local inelastic phenomenon including concrete cracking, crushing, steel yielding, buckling, fracture and bond slip between steel and concrete, etc.* The nonlinear models can generally be classified based on the degree of idealization used in the model. A comparison of three idealized model types for simulating the nonlinear response of a reinforced concrete beam-column is shown in Figure 1-6 [taken from ATC 72 (2010)].

1) ***Continuum Models:*** At one extreme are the detailed continuum finite element models that explicitly model the nonlinear behavior of the materials and elements that comprise the component. A continuum model might include finite elements representing the concrete, longitudinal reinforcement, and shear reinforcement, in which associated constitutive models (e.g., the nonlinear stress-strain curves of concrete and steel) would represent various nonlinear phenomenon. Continuum models generally do not enforce any predefined behavioral modes and, instead, seek to model the underlying physics of the materials and elements. They do not require definitions of member stiffness, strength or deformation capacity, as these effects are inherently captured in the model through the material properties.

2) ***Lumped Plasticity Models:*** At the other extreme are lumped plasticity (concentrated hinge) models in which the nonlinear action is lumped at certain points of the structure and the nonlinear functions between various actions and corresponding deformations are assigned at those points. In this way, these models are defined entirely by the phenomenological description of the overall force-deformation response of the component. For example, a "concentrated hinge element" assigned at both ends of a beam or column might represent a lumped nonlinear flexural behavior defined by a nonlinear function between end moment and resulting curvature (or rotation) of the member. This nonlinear function should correspond to the observed force-deformation behavior and hysteretic test data of similar beam or column components.

3) ***Distributed Inelasticity Models:*** In between the two extremes are distributed inelasticity (fiber) models, which can explicitly capture some aspects of nonlinear behavior while some effects are captured implicitly. For example, the complete nonlinear stress-strain curves of materials can be defined to capture important aspects of material nonlinearity. However, the integration of flexural stresses and strains through the cross section and along the member is considered implicitly. These models typically enforce some behavior assumptions (e.g., plane sections remain plane) in combination with explicit modeling of uniaxial material response.

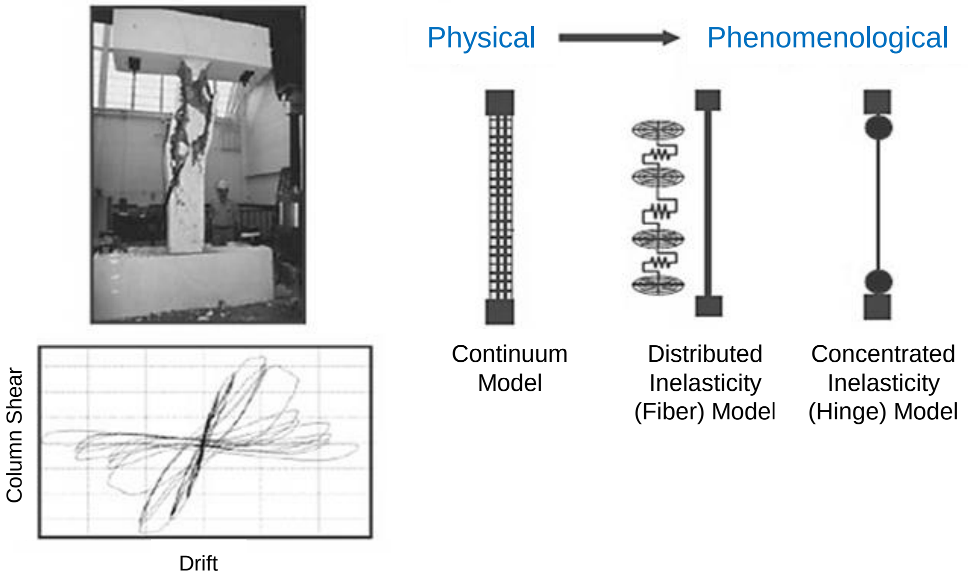


Figure 1-6: Comparison of nonlinear component model types. [Taken from ATC 72 (2010)]

Continuum and distributed inelasticity models can more accurately capture behaviors such as initiation of concrete cracking and steel yielding, but they can be limited in their ability to capture strength degradation such as reinforcing bar buckling, bond slip, and shear failure. While continuum models should not require calibration to component response, in practice, they do require some phenomenological calibration to account for behavior that is not captured by the formulation.

Concentrated hinge models, however, can capture strength degradation effects, but in a more empirical manner. They are highly phenomenological in that the underlying nonlinear functions that describe the structural behavior are based on calibration to overall component behavior. In contrast, the fiber and continuum finite element models are calibrated more at the material level, where the kinematics and equilibrium of the components are represented more directly by the model formulation. Concentrated hinge models are also more consistent with common limit state checks related to stress resultants (forces) and concentrated deformations (hinge or spring deformations) in current building codes and standards. The current practice for nonlinear modeling is mostly based on the use of lumped plasticity (concentrated hinge) and distributed inelasticity (fiber) component models (Figure 1-7). A brief introduction of these two approaches will be given next.

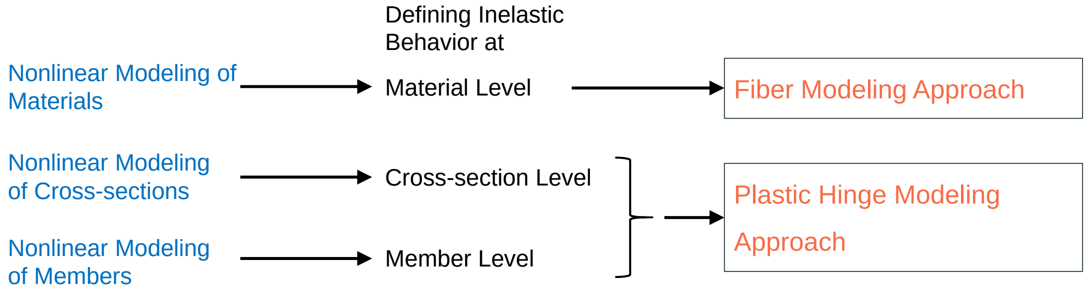


Figure 1-7: The current approaches for nonlinear modeling.

1.4. Fiber Modeling Approach (Distributed Nonlinearity)

In this approach, *the cross-section of a structural member is divided into a number of uniaxial "fibers" running along the larger dimension (length) of the member*. Each particular fiber is assigned a uniaxial stress-strain relationship (e.g., as shown in Figure 1-8) capturing various aspects of material nonlinearity in that uniaxial fiber. These fibers may either be used throughout the whole length of the member or for a fraction of total length (i.e. the fraction of length where the inelastic action is anticipated. It is also sometimes referred to as the plastic length or plastic zone). While defining the fiber model of a member (beams, columns or walls), the length of these fiber segments (plastic length) is defined. The deformation measure for demand-to-capacity ratio (D/C) is material strain (i.e., the strain demand produced by the loading will be divided by the specified material strain capacity) of each material fiber to calculate the deformation D/C ratio for performance-based analysis).

A complete beam, column or wall element may be made up of several fiber segments. For reinforced concrete members, a fiber segment comprises of several fibers of concrete and steel (for reinforcing bars) with their respective stress-strain relationships. *The fiber modeling can account for the axial-flexural interaction (and the axial deformation caused by bending in columns and shear walls). The shear behavior in beams, columns and shear walls needs to be modeled separately (which can either be elastic or inelastic).*

The subsequent sections explain the basics of this approach for each structural element separately.

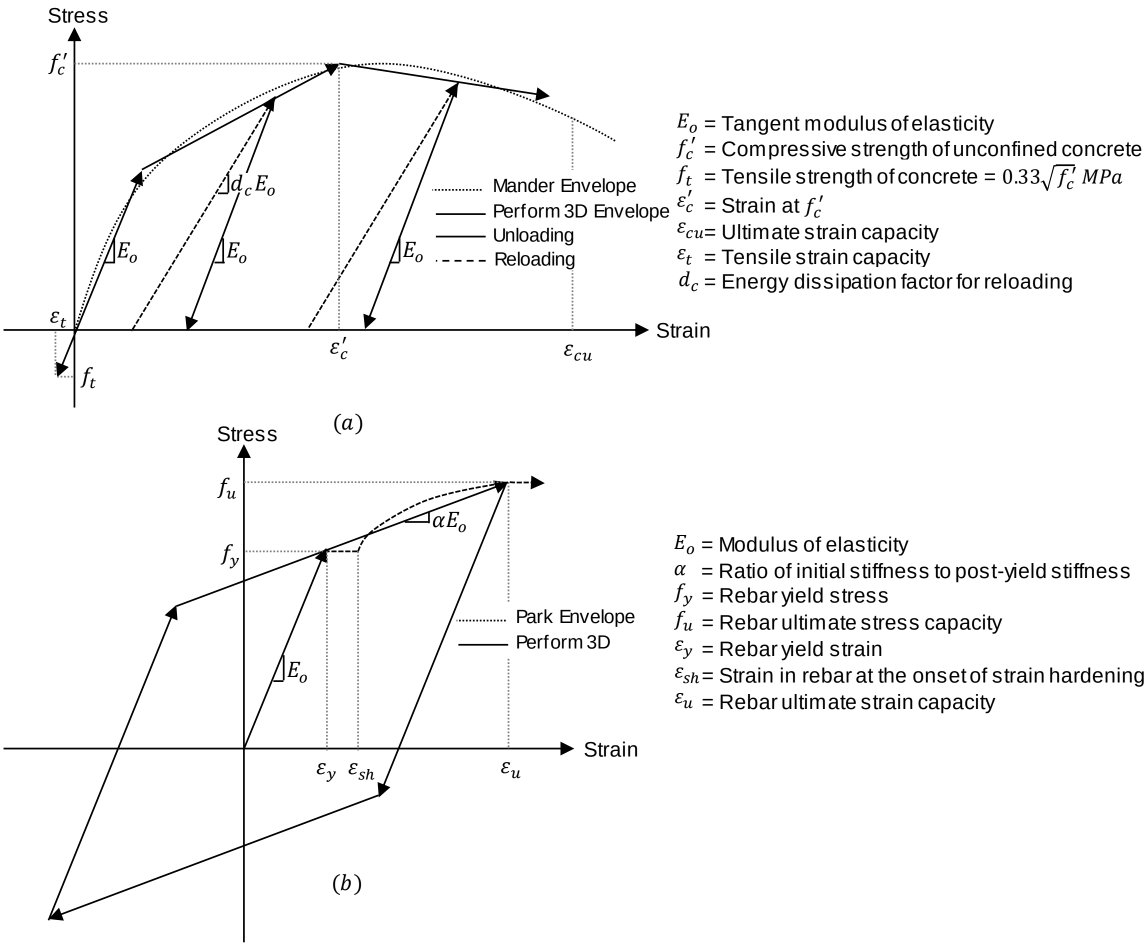
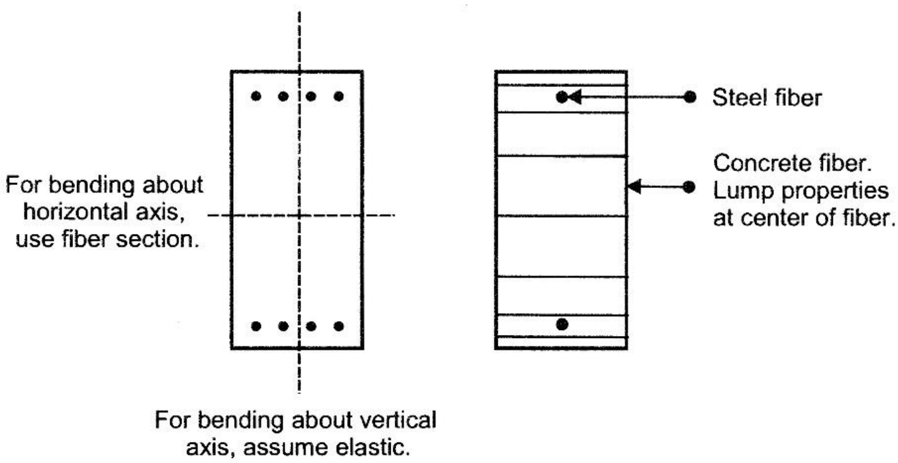


Figure 1-8: An example stress-strain curve for (a) concrete and (b) steel (to be assigned to uniaxial concrete and steel fibers) in a reinforced concrete cross-section.

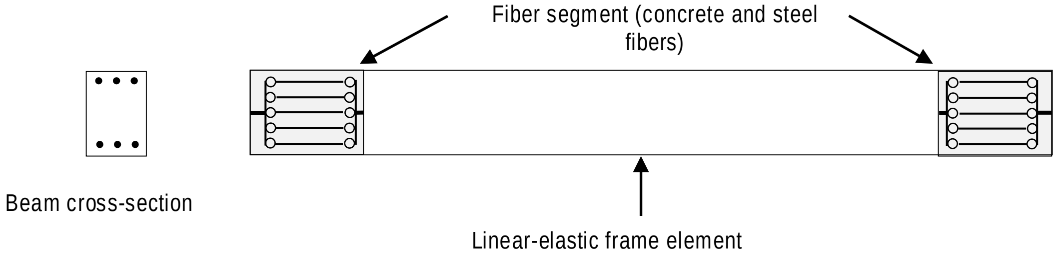
1.4.1. Fiber Sections for Beams

Figure 1-9 shows the type of fiber model that can be used for a beam cross-section. A common assumption for a beam is that there is inelastic bending in only one direction, usually in the vertical direction (bending about the horizontal axis). For horizontal bending the behavior is assumed to be elastic. Often there is little or no horizontal bending, because the beam is braced, for example, by a floor slab (Powell 2010).

To model bending behavior in the vertical direction, fibers are needed only through the depth of the beam, as indicated in the figure. Therefore, the cross-section is divided (sliced) in one direction only to define the uniaxial concrete and steel fibers. For horizontal bending, an elastic bending stiffness is separately specified (i.e., an EI value). For vertical bending, the EI is determined by the fiber model. For horizontal bending, the model assumes that there is no P-M interaction. It also assumes that there is no coupling between vertical and lateral bending (Powell 2010).



(a) Fiber section of a reinforced concrete beam [Modified from Powell (2010)]



(b) Fiber segments at both ends of a reinforced concrete beam with linear-elastic frame element in-between. The length of fiber segments (plastic length) is a small fraction of the total beam length.

Figure 1-9: Fiber section of a reinforced concrete beam.

1.4.2. Fiber Sections for Columns

A fiber model for a column must usually account for biaxial bending. Hence, fibers are needed in both cross-sectional directions, as indicated in Figure 1-10. This type of model accounts for axial-flexural interaction in both cross-sectional axes (generally referred to as the P-Mx-My interaction, where x and y are cross-sectional axes).

For both beams and columns, the behavior in torsion is usually assumed to be elastic. It is also assumed to be uncoupled from the axial and bending behavior.

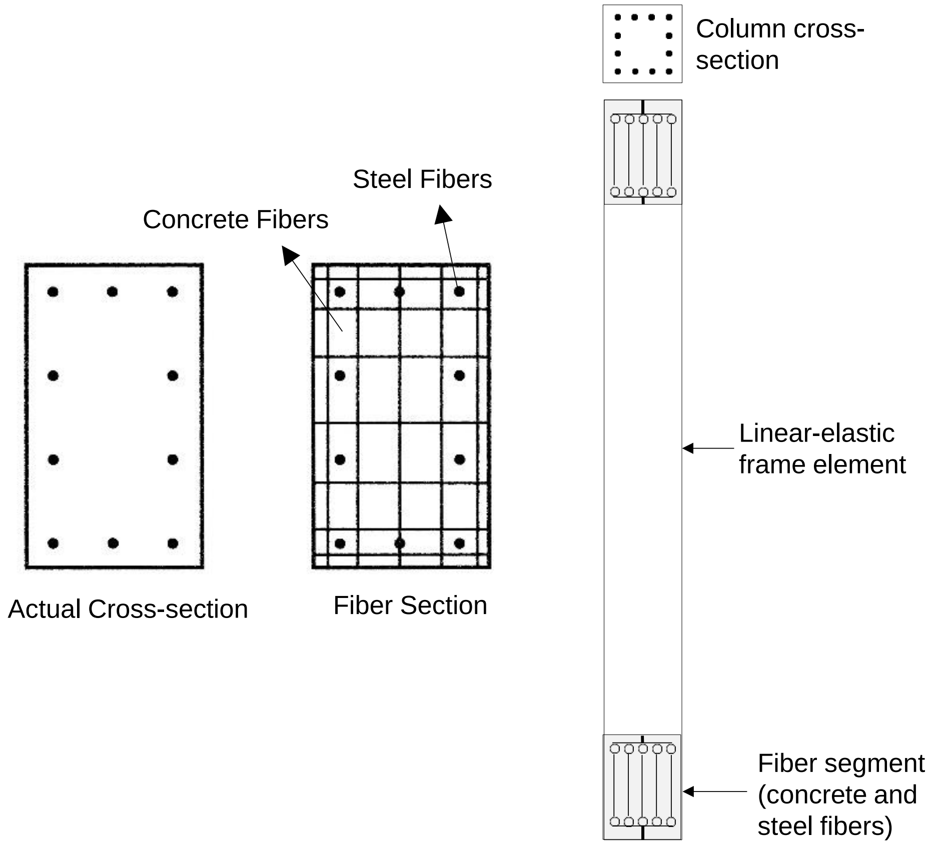


Figure 1-10: Fiber section of a reinforced concrete column, cross-sectional view (left), side elevation (right) [Modified from Powell (2010)]

1.4.3. Fiber Sections for Walls

A shear wall has bending in two directions, namely in-plane and out-of-plane. Often it is accurate enough to consider inelastic behavior only for in-plane bending (membrane behavior), and to assume that the behavior is elastic for out-of-plane bending (plate bending behavior). In this case the fiber model can be similar to that for a beam, with fibers only for membrane behavior, as shown in Figure 1-11.

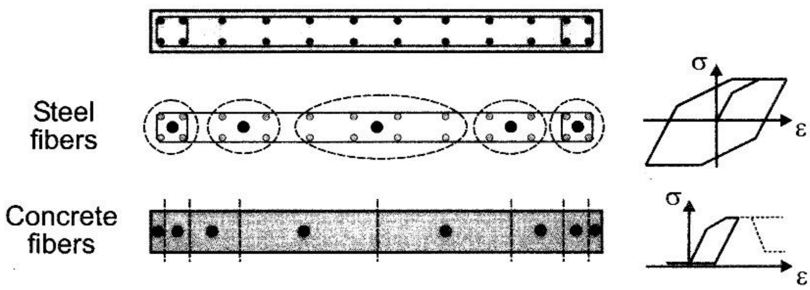
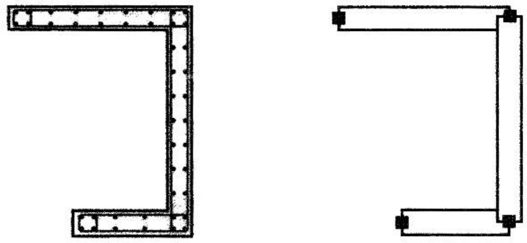


Figure 1-11: Fiber section for membrane behavior of a reinforced concrete wall (Modified from Powell [5])

As with a beam, an effective EI is specified for out-of-plane bending, and there is no coupling between membrane and plate bending effects. If inelastic plate bending is to be considered, there must also be fibers through the wall thickness. In this case the fiber model is similar to that for a column.

Figure 1-11 shows the cross section for a plane wall. More complex cross sections can be divided into a number of plane walls, as shown in Figure 1-12. The cross section in Figure 1-12(a) could be treated as a single section, rather like a column. However, this is likely to be inaccurate because it does not allow for warping of the cross section. For a fiber section it is usual to assume that plane sections remain plane. This can be reasonable for a plane wall, even if it is quite wide, but it can be incorrect for an open, thin-walled section. It is more accurate to divide the section into plane parts, as in Figure 1-12(b).



(a) Wall cross section (b) Model with plane walls

Figure 1-12: Wall section modeled as several plane walls [Taken from Powell (2010)]

1.4.4. Limitations of Fiber Models [Taken from Powell (2010)]

Fiber models can capture:

(a) the cracking of reinforced concrete cross sections in the elastic range,

(b) P-M strength interaction, and

(c) both axial and bending deformations after yield.

However, fiber models can not necessarily predict ductile limits and subsequent strength loss. These depend on complex aspects of behavior that are not necessarily included in fiber models. Some of these are as follows.

1. The ductile limit for a reinforced concrete section may be reached when the concrete crushes and loses strength. In an actual cross section, crushing starts at the extreme edge or comer of the cross section and progresses continuously into the section. In a fiber model, crushing occurs fiber-by-fiber and progresses discontinuously into the section as more fibers yield. If relatively few fibers are used to model a cross section, the fibers are large, crushing starts later in the model than in the actual section, and the crushed part of the cross-section changes in relatively large jumps. As more fibers are used the model becomes more accurate, but the computational cost increases.

2. The strength of concrete in compression, and also its ductility, depend on the amount of confinement. In a column, some concrete will be within the confinement and some will be in an unconfined outer shell. This may need to be accounted for in the fiber model. Also, in any given column the effectiveness of the confinement, and hence the strength and ductility of the concrete, may be uncertain. The same is true in a wall.

3. Under cyclic loading the reinforcement can yield in tension in one half cycle and compression in the next half cycle. Also, cracks that open in one cycle may not close completely. The ductile limit of a cross section may be governed by buckling of the reinforcement as it yields in compression. In a fiber model, the stress-strain relationship for steel fibers can, in principle, account for buckling, but the buckling behavior is uncertain and difficult to model.

4. The largest inelastic deformations in a column are likely to occur at the column ends, either at a beam-to-column connection or at the foundation level of the structure. At these locations there can be large bond stresses, with significant penetration of bond slip into the connection region or the foundation. This can have a substantial effect on the column stiffness, and possibly on its strength. Bond slip is not considered in a basic fiber model. Bond slip can be modeled, but the process is usually too complex to be included in a model of a complete structure.

5. The strength of a column may be controlled by shear, or by P, M and V (shear force) acting in combination. The basic fiber model considers only P-M interaction. As noted later, it is much more difficult to model P-M-V interaction.

In summary, a fiber model can be useful, but it is not a complete solution. A fiber model may not be accurate for large cyclic deformations, it does not account for bond slip or shear force effects, and it probably cannot predict the ductile limit and the amount of strength loss. Fiber models can certainly be better than models based on plasticity theory, but they still have major limitations.

1.5. Plastic Hinge Modeling Approach (Concentrated Nonlinearity)

The fiber modeling approach is based on defining the nonlinear properties directly at the fundamental material level which then manifest at the cross-section, member and structure levels. *In the plastic hinge modeling approach, it is assumed that all inelastic deformation is concentrated at certain points or locations defined by the zero-length hypothetical elements known as “plastic hinges”.* Using these plastic hinges, the inelastic relationship (relating a particular structural action to the corresponding deformation) is directly defined at cross-section or member level to include the effects of nonlinearity in the overall structural stiffness (instead of specifying the inelastic material properties).

In a structural model*, the locations at which the inelastic action is assumed concentrated (i.e., the locations of plastic hinges) are selected based on the expected zones of damage under the anticipated loadings.* For example, the Figure 1-13 shows the bending moment and shear force diagrams of a 2D frame under the gravity and lateral loads. Under the lateral loads, the bending moments are maximum at the ends of all beams and columns. Therefore, under a lateral dynamic loading, the inelastic action (flexural yielding of the beams and columns) is expected to mostly concentrate in these regions of high bending moments. Therefore, the inelastic flexural action can be captured in the model by specifying the plastic hinges at both ends of these elements.

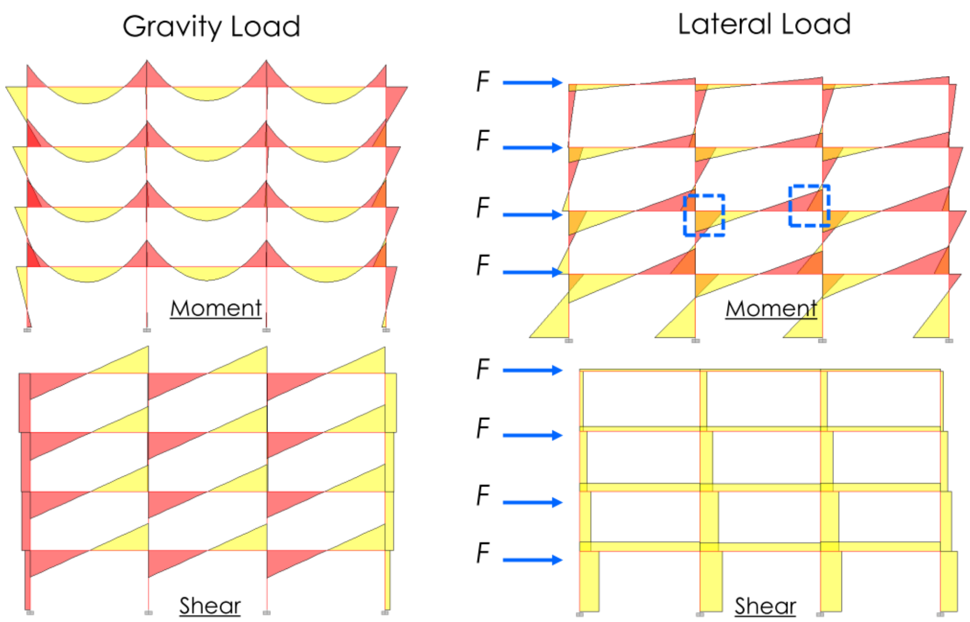


Figure 1-13: The bending moment and shear force diagrams of a 2D frame under the gravity and lateral loads (Source: Pramin Norachan, AIT Solutions)

Figure 1-14 shows the lumped plasticity (or plastic hinge) model of a 2D RC frame (left) subjected to lateral earthquake loading. The yellow circles (at both ends of beams and base of ground floor columns) show the potential locations of inelastic actions due to this loading. In the computer model (right), the plastic hinges are introduced at all such locations. *In this case, these plastic hinges are specified with the nonlinear relationships between flexural action (i.e., moment) and flexural deformation (i.e., cross-sectional curvature or member rotation).* In between the hinges, the elastic frame element is used to capture the elastic response of the member. The nonlinearity is assumed to be concentrated (lumped) only at the plastic hinges.

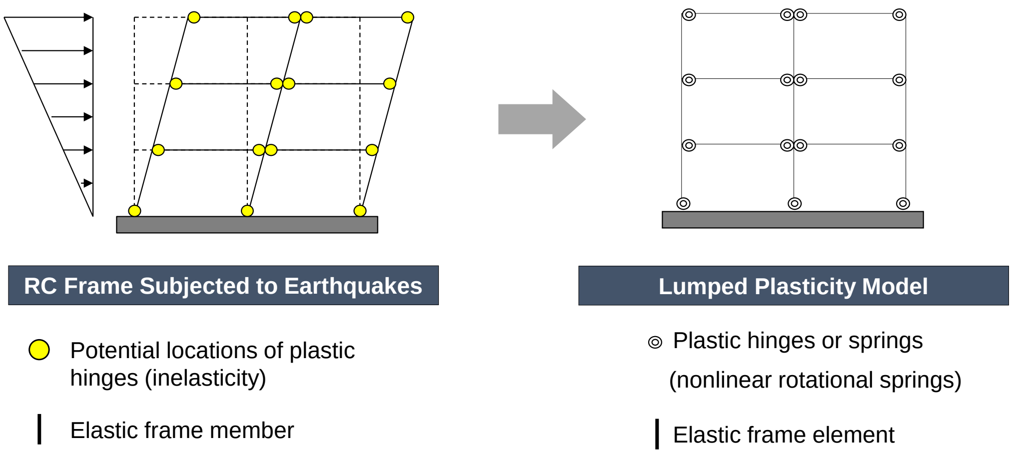


Figure 1-14: The lumped plasticity (plastic hinge) model of a 2D frame subjected to lateral earthquake loading.

Similarly, the Figure 1-15 [Taken from ATC-72 (2010)] illustrates the key features of an inelastic plastic hinge model for reinforced concrete frame elements. The nonlinear relationship between the moment and flexural deformation of member (rotation) is assigned to plastic hinges at both ends of the column shown in Figure 1-15a. The features of this element are generally applicable to other types of elements. This example is taken from a study of reinforced concrete columns by Haselton et al. (2008) making use of a degrading cyclic model developed by Ibarra and Krawinkler (2005). In this example, inelastic response is idealized by a backbone curve (Figure 1-9b) that relates moment to rotation in the concentrated hinges. The definition of the backbone curve and its associated parameters depend on the specific attributes of the nonlinear model used to simulate the hysteretic cyclic response (Figure 1-9c).

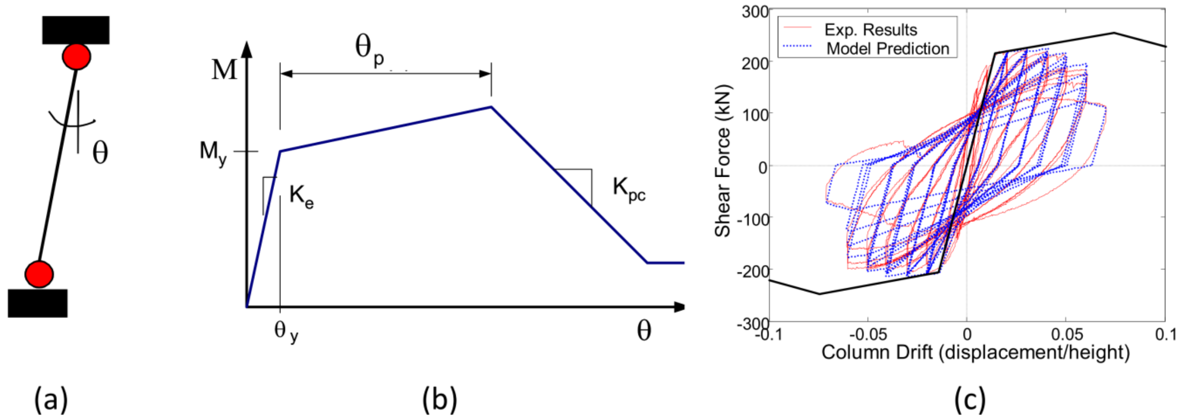


Figure 1-15: Illustration of modeling components for a reinforced concrete beam-column: (a) inelastic hinge model; (b) initial (monotonic) backbone curve; and (c) cyclic response model (Haselton et al. 2008). [Taken from ATC-72]

In the above example, the plastic hinges defined at the ends of column model are flexural hinges (with a specified moment vs. rotation relationship in the primary flexural direction). However, *several types of plastic hinges can be used to specify the nonlinear force-deformation relationship (or in general, the action-deformation relationship) for any degree-of-freedom at a particular location along the member.* Several types of plastic hinge elements are available in ETABS or PERFORM 3D and can be used for this purpose. For example, Figure 1-10 below shows the six degrees-of-freedom and corresponding actions for a node in 3D space. These include moments in two directions, shears in two directions, axial force and torsion. There are six deformations corresponding to these six actions. The relationship between each of the action and corresponding deformation is defined by an "action-deformation relationship". *The plastic hinge elements allow us to specify a nonlinear action-deformation relationship for each of the action in a structural model.* These include, for example, a nonlinear shear force vs. shear deformation relationship, a nonlinear axial force vs. axial deformation relationship etc. (see Figure 1-11).

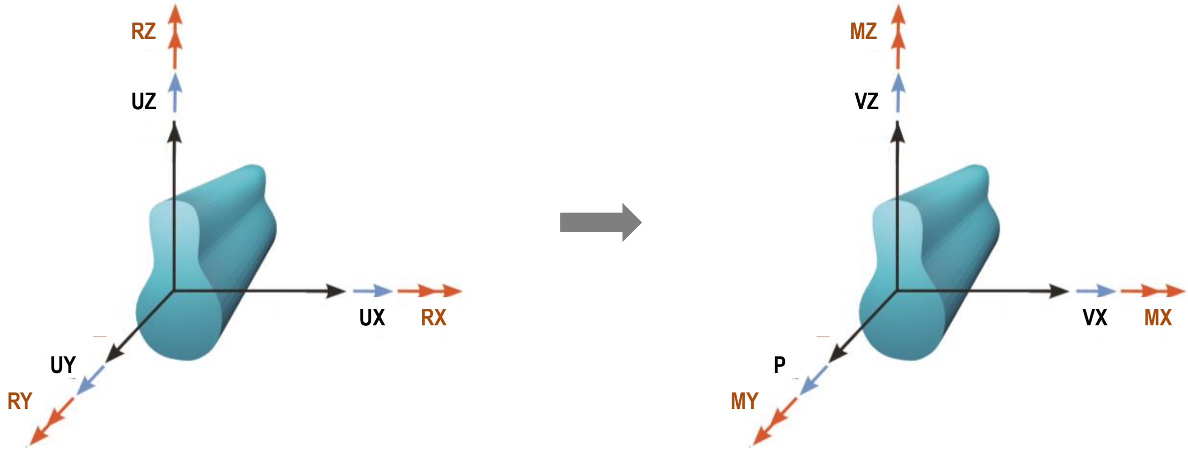


Figure 1-16: The six degrees-of-freedom and corresponding actions for a node in 3D space.

The moment (flexural) plastic hinges can be of "curvature-type" or "rotation type". The curvature-type plastic hinges are used to introduce the nonlinearity at the cross-section level. The primary force-deformation relationship assigned to these hinges is the *moment-curvature relationship*. The deformation measure for demand-to-capacity ratio (D/C) is hinge curvature. The rotation-type plastic hinges (as shown in the example in Figure 1-9) are used to introduce the nonlinearity directly at the member level. The primary force-deformation relationship assigned to these hinges is the *moment-rotation relationship*. The deformation measure for demand-to-capacity ratio (D/C) is hinge rotation.

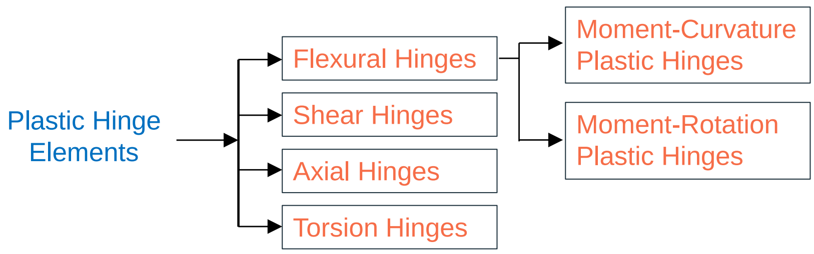


Figure 1-17: Difference types of plastic hinges.

The following are some important features of this model [taken from ATC 72 (2010)].

- *The backbone curve* is generally expected to capture both hardening and post-peak softening response. The peak point of the curve is sometimes referred to as the "capping point," and the associated deformation capacity is the "capping deformation." The extent to which cyclic deterioration is modeled in the analysis will determine the extent to which the backbone curve is calibrated to initial or degraded component response, and how the characteristic points on the curve correspond to component acceptance criteria for the onset of damage and significant deterioration.

- *The cyclic model* incorporates deterioration in strength and stiffness, which degrades the backbone curve as a function of the damage and energy dissipated in the component. Accordingly, the initial backbone curve (Figure 1-9b) is calibrated to component response that is representative of monotonic loading. When properly calibrated, cyclic deterioration enables the model to capture the deteriorated cyclic response of the component. However, not all models can capture this strength and stiffness deterioration. Where the cyclic deterioration is not accounted for in the model, the backbone curve should be modified by appropriate reductions in the peak strength and inelastic deformation quantities.

- The backbone curve and cyclic deterioration properties should be *calibrated to the median response of the component*. In this way, basic modeling results will represent a median (or statistically neutral) assessment of response. Variability in component properties, or system response, can then be applied to establish appropriate margins against exceeding certain limit states, as evaluated through acceptance criteria (forces or deformations) on structural components, or the overall system.

1.5.1. Plastic Hinge Modeling of RC Beams [Taken from Powell (2010)]

The inelastic behavior of beams in bending can be modeled using (moment-rotation type) plastic hinges. The RC beams are modeled as elastic frame elements with plastic hinges at both ends. The moment vs. rotation relationship (rigid-plastic type) is specified for each of the plastic hinge element. This is shown in Figure 1-18. *It is assumed that all inelastic deformation is concentrated in zero-length plastic hinges and that the rest of the beam remains elastic.* The plastic hinges are initially rigid and begins to rotate (or participate in structural response) at first yield. Therefore, they are also sometimes referred to as the “rigid plastic hinges”. This means that the initial elastic flexural stiffness of the beam is used in the analysis prior to the yield point. However, after the yield, the flexural behavior is governed by the nonlinear action-deformation relationship (e.g., moment-curvature, or moment vs. rotation relationship) specified at the plastic hinge location.

The deformation measure for demand-to-capacity ratio (D/C) is hinge rotation (i.e., the rotation demand produced by the loading will be divided by the specified rotation capacity) of the beam to calculate the deformation D/C ratio for performance-based analysis).

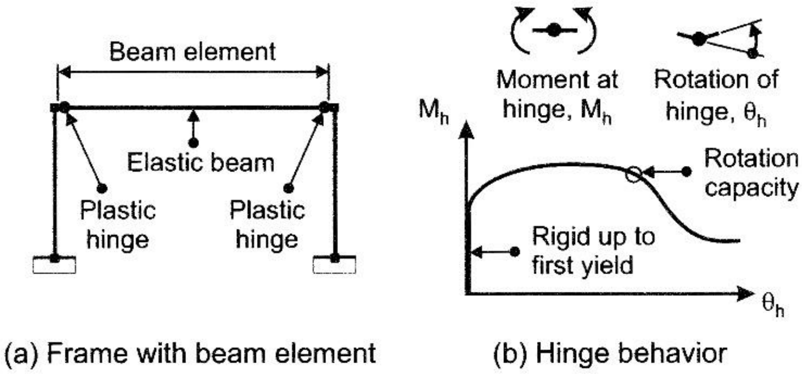
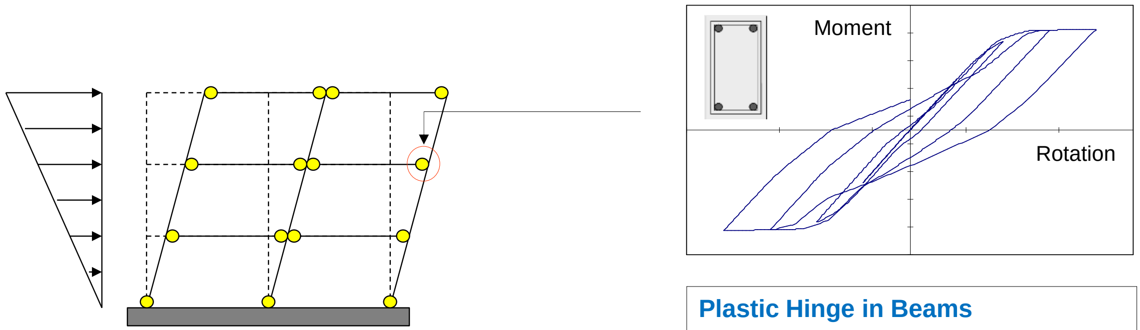


Figure 1-18: Plastic hinge modeling of beams [taken from Powell (2010)].

The primary properties required to define a plastic hinge are its bending strength and the hinge rotation capacity, which is the rotation at the ductile limit. These properties are usually determined experimentally. However, if the experimental results are not available, the plastic hinge properties can also be estimated by analysis. An alternate way to obtain these properties is to use specialized guidelines e.g., the modeling parameters prescribed in ASCE 41-17 (2017). Since the plastic hinges are initially rigid, the initial elastic deformation is zero and all the deformation is plastic for monotonically increasing deformation. Also, there is no difference between plastic and post-yield deformation.

As mentioned earlier, the complete definition of a plastic hinge requires to specify both the backbone curve and the cyclic behavior. The cyclic behavior is defined by a set of rules for unloading and re-loading of the backbone curve (force-deformation relationship). Several of such rules (also known as the *idealized cyclic or hysteretic models*) are developed for different types of components and are directly available in analysis software e.g., ETABS or SAP 2000. The user can directly select and customize them according to his/her requirement. For example, the Figure 1-19 shows the complete nonlinear cyclic moment vs. rotation relationship assigned to the plastic hinges of an RC frame.



An RC Frame subjected to lateral loads (Beams and ground floor columns are modeled using plastic hinges)

Plastic Hinge in Beams

Seismic energy dissipation mechanism is relied on plastic flexural-deformation of beams and ground floor columns.

Figure 1-19: The complete nonlinear moment vs. rotation relationship is assigned to each plastic hinge. This behavior should either be determined experimentally, through analysis or from empirical modeling parameters specified in standards or guidelines.

Question: Since the inelastic behavior in an actual beam is likely to be distributed over a significant length of the beam, a zero-length plastic hinge is an approximation. *Is it reasonable to assume that the inelastic action is concentrated at only certain points along the length (e.g., the ends of a frame element)?*

Answer: The Figure 4.20 (b) shows plastic hinges for bending moment and for shear. In an actual structural member, plastic deformation must be distributed over a finite length, often referred to as a "plastic zone". For bending, the plastic zones are often short, and it can be physically reasonable to lump the plastic deformation into a zero-length hinge. Plastic shear deformations are likely to be distributed over longer lengths, and it may be less reasonable to lump the plastic deformation into a zero length hinge, with a sudden shear displacement.

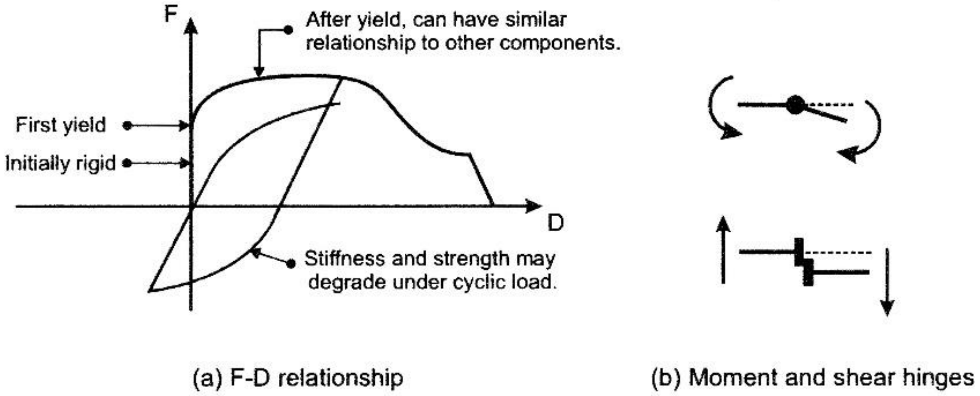


Figure 1-20: Rigid plastic hinges. For a flexural hinge, F = moment and D = rotation. For a shear hinge, F = shear force and D = shear deformation. [taken from Powell (2010)].

There are several important phenomena associated with the complete cyclic behavior of a plastic hinge. For example, Figure 1-21(a) shows a "*full*" loop, where the shape of the loop is the same as the shape of the basic force-deformation relationship. Figure 1-21(b) shows a loop that has "*stiffness degradation*". In this loop the unloading-reloading stiffness is smaller than the stiffness in the basic force-deformation relationship. The area of the loop is a measure of the amount of inelastic energy that is dissipated under cyclic deformation. The loop in Figure 1-21(b) has a smaller area than that in Figure 1-21(a), so stiffness degradation also leads to energy degradation. Figure 1-21(c) shows a "*pinched*" loop. There can be strength degradation as well as stiffness degradation. Degradation of both types may develop progressively, with the amount of degradation increasing as the number of deformation cycles increases, as shown in Figure 1-21(d). Some components increase in strength under cyclic deformation, as shown in Figure 1-21(e). Degradation may be affected by strength loss, as indicated in Figure 4.4(f). If a component is deformed beyond its ductile limit in one direction, and the deformation is then reversed, the strength in the opposite direction may or may not be reduced.

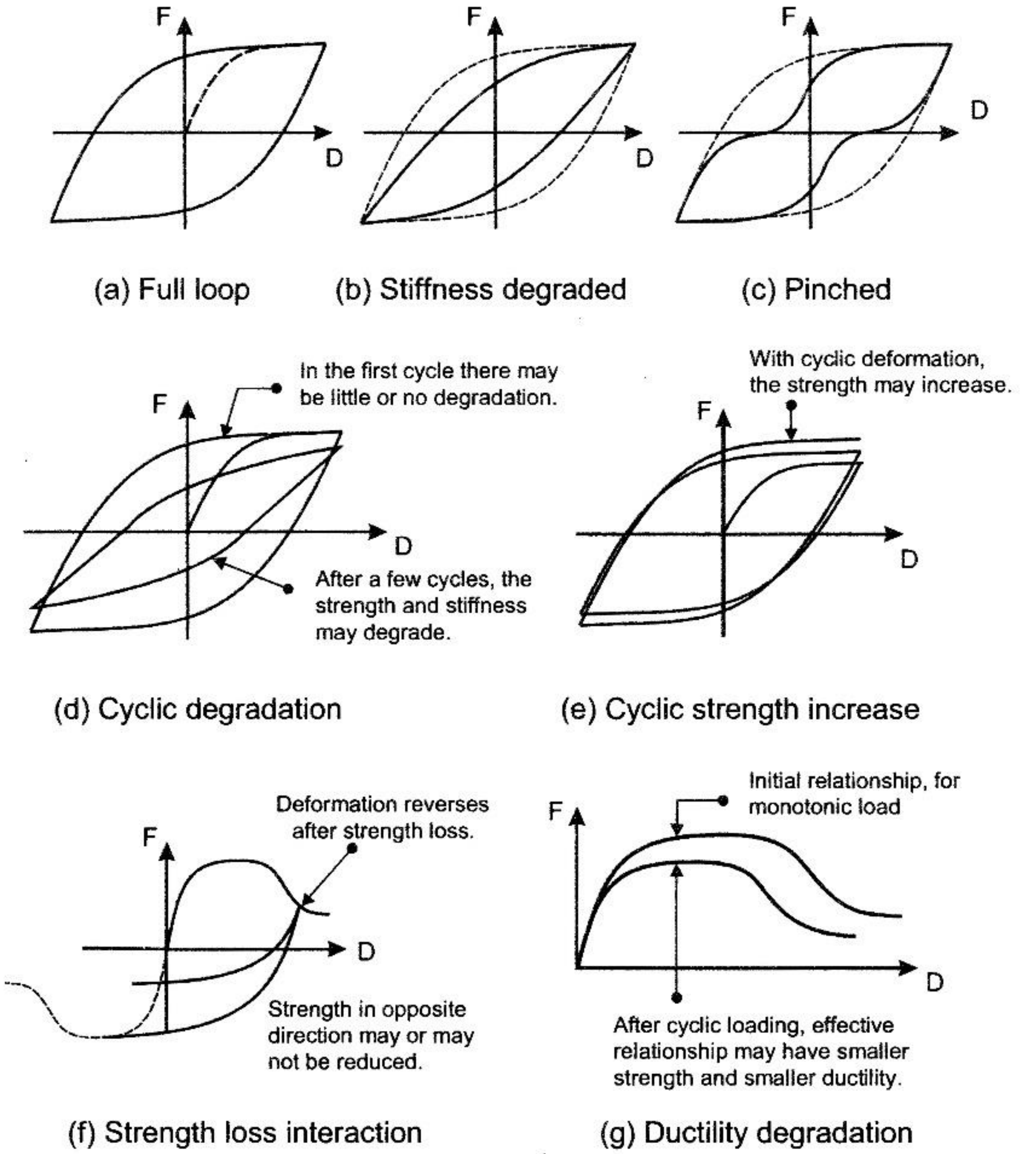


Figure 1-21: Some phenomenon and complications related to the cyclic deformation [taken from Powell (2010)].

In summary, the Figure 1-22 shows a 2-hinge beam element as compared to an elastic beam element. The stiffness of an elastic beam depends on elastic material and cross-sectional properties. The stiffness of a beam with plastic hinges will be same as the elastic beam prior to the yielding. However, after the yielding, it will be controlled by the nonlinear moment-rotation relationship specified for the plastic hinges.

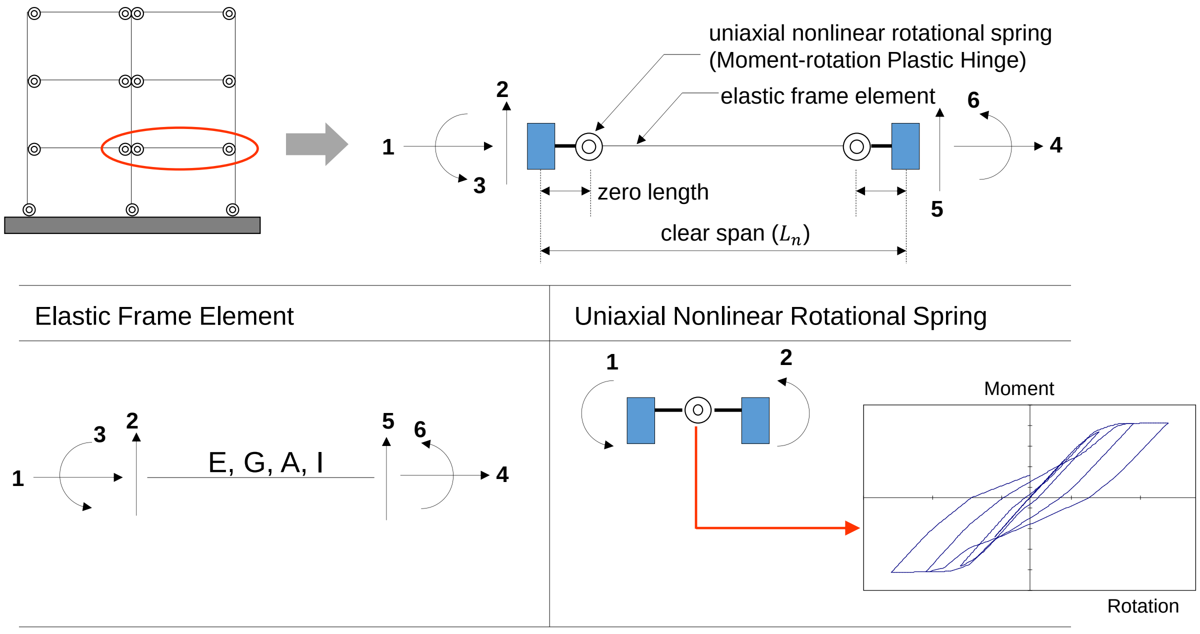


Figure 1-22: A 2-hinge beam element as compared to an elastic beam element.

1.5.2. Force-Deformation Relationships in ASCE 41 and Performance-based Evaluation

Seismic Design Guidelines for Tall Buildings (PEER, 2010) discusses two levels of performance-based seismic assessment: (1) service level evaluation; and (2) Maximum Considered Earthquake (MCE) level evaluation, which generally involve comparisons of force and deformation demands imposed by the specified earthquake hazard to corresponding limit state capacities of the structural components and systems.

For the performance-based seismic evaluation of components, several performance levels (e.g., Immediate Occupancy, Life Safety and Collapse Prevention) are defined and marked on the nonlinear force-deformation relationships assigned to plastic hinges (or material stress-strain curves in case of fiber modeling). Generally, in the plastic hinge modeling, five points labeled A, B, C, D, and E are used to define the force-deformation behavior of the plastic hinge. The points corresponding to various performance levels (e.g., Immediate Occupancy, Life Safety and Collapse Prevention) are also specified as the *"acceptance criteria"* for the hinge (Figure 1-23).

What are Acceptance Criteria?

Performance assessment using nonlinear analysis requires a set of criteria defining acceptable performance. These criteria are the capacities of a particular member (or material) corresponding to a particular performance level. Therefore, the terms "deformation capacity" and "acceptance criterion" can be interchangeably used in the context of for nonlinear modeling and analysis. For deformation-controlled actions, these capacities are specified in terms of deformation measures (e.g., the plastic rotations for beam hinges, or material strains for column fibers, etc.). These capacities should either be determined experimentally or from reliable guidelines developed based on extensive experimentation.

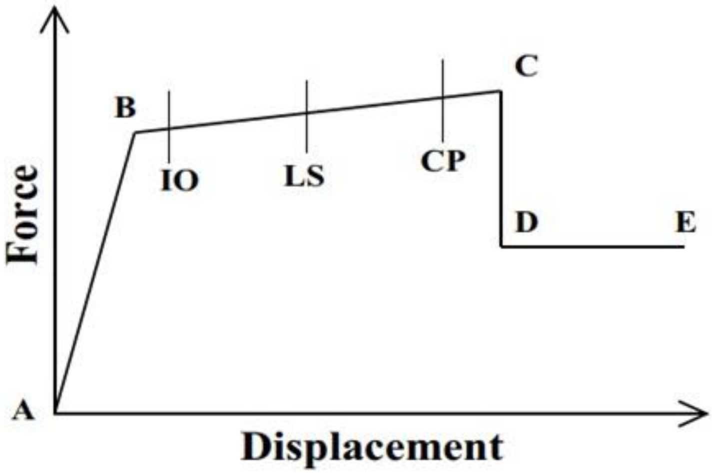


Figure 1-23: The acceptance criteria (capacities) marked on the force-deformation behavior of hinges.

One of such reliable and useful guidelines is ASCE 41. It is a standard for the seismic rehabilitation (retrofit) of existing buildings. However, it may also be used for the design and performance-evaluation of new buildings. It was published in 2006 by SEI and ASCE, following earlier publications and "pre-standards", including ATC 40 (1996), FEMA 273 (1997) and FEMA 356 (2000). Beside detailed structural evaluation and retrofit methodology and analysis procedures, ASCE 41 provides modeling guidelines for inelastic analysis and performance assessment of different structural components. For the plastic hinge modeling, it recommends deformation capacities for a wide range of components, for the Immediate Occupancy (IO), Life Safety (LS) and Collapse Prevention (CP) performance levels.

ASCE 41 also recommends “modeling parameters”, which essentially define the shape of the force-deformation relationship. It also allows the force-deformation relationship for a structural component to be obtained from experimental data. If this is not feasible, the relationship shown in Figure 1-24 can be used.

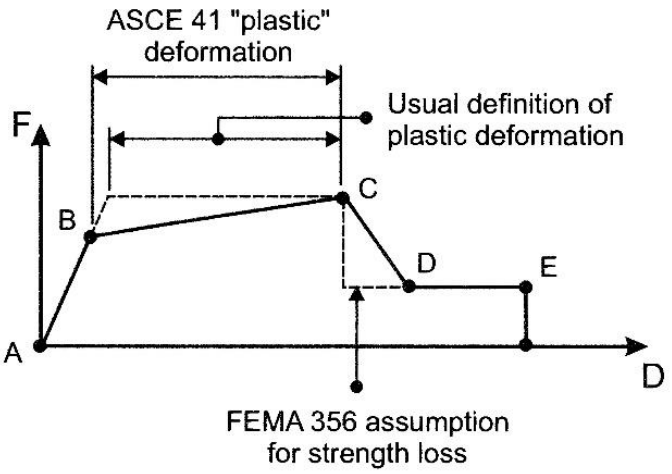


Figure 1-24: ASCE 41 force-deformation relationship

1.6. Which Modeling Approach Should be Used for What Application?

The choice of model type for a given application involves a balance between reliability, practicality, and computational efficiency, subject to the capabilities of available software and computational resources. The optimal model type depends on many factors, including the structural system and materials, governing modes of behavior, the expected amount of nonlinearity, and the level of detail available for the input and output data. The reliability of the model comes from its ability to capture the critical types of deformation that are of interest to the modeler and control the response.

Plastic hinge (and spring) models have the advantage of being computationally efficient by modeling highly nonlinear effects in localized regions of the structure with few degrees of freedom. The models generally employ pre-defined functions to define the nonlinear response of the component. Concentrated spring models are typically implemented to capture single degree of freedom response (e.g., moment vs. rotation), but they may include multiaxial response through yield surfaces (e.g., $P_x - M_x - M_y$ interaction) or other means. By capturing complicated behavior with highly idealized models, concentrated hinge models are very versatile, but they are also empirical and limited to modeling phenomena over the range of components and behavior modes for which they have been calibrated.

For beam-columns, fiber models provide the capability to numerically integrate material response through the member cross sections at a more fundamental material level. The fiber integration through the cross section can be used either in conjunction with a finite-length hinge zone or with model formulations that simulate distributed inelasticity along the member length. Fiber-type models for beams and beam-columns generally invoke kinematic assumptions, such as the Euler-Bernoulli (plane sections remain plane) assumption, to relate uniaxial stresses and strains through the member cross section to stress resultants and generalized strains for the cross section (e.g., moment vs. curvature or axial force vs. axial strain). To the extent these assumptions reflect the underlying member response, they offer tremendous benefits; however, one should be mindful of the limitations posed by these kinematic assumptions. For example, the Euler-Bernoulli assumption precludes cross-section warping or distortion due to: (1) shear and torsion effects; (2) local buckling of steel reinforcing bars or steel webs and flanges; (3) slip between components of the section; or (4) material cracking and crushing. Although certain behavior can be incorporated in the fiber material response (e.g., steel materials that are calibrated to simulate reinforcing bar buckling, or concrete models with tension softening), these adjustments require empirical calibration that negate some of the appeal of the fundamental aspects of fiber-type models.

Continuum finite element models represent the behavior at the most fundamental level and provide the ability to model the complete interaction of three-dimensional behavior, including complex geometries and multi-axial stress and strain states. However, three-dimensional (3D) continuum models are the most computationally intensive, particularly where the numerical mesh refinement is controlled by the smallest dimension of the member (e.g., where finite element meshing through the thickness or depth of a member will dictate the mesh size required along the member length). For this reason, 3D continuum models are typically only used to simulate portions of overall systems. In addition, while continuum models offer the potential for capturing response at very fundamental levels, their practical application is limited by both computational resources and data to calibrate certain localized behavioral effects. For example, 2D shell or 3D continuum finite element models can capture the response of isotropic steel materials fairly well, whereas many unresolved challenges remain for simulating the detailed behavior of reinforced concrete members, considering concrete cracking/dilation and interactions between steel reinforcing bars and concrete (e.g., bond and anchorage).

The modeling decisions are centered on how simplified versus complex the structural model should be and the needed level of complexity of the model depends on aspects such as the expected failure modes and their consequences, the anticipated locations of damage, and the anticipated level of nonlinearity.

Chapter 2

Nonlinear Modeling Capabilities of ETABS 2016

This chapter is focused on introduction to various nonlinear capabilities of CSI ETABS. Most part of this chapter is taken from the Analysis Reference Manual (for SAP 2000, ETABS and CSI Bridge).

2.1. Inelastic Components (Plastic Hinges) in CSI ETABS

In CSI ETABS, the yielding and post-yielding behavior is modeled using discrete user-defined "hinges". Hinges can be assigned to a frame element at any location along the clear length of the element. *Each hinge represents concentrated post-yield behavior in one or more degrees of freedom. Hinges only affect the behavior of the structure in nonlinear static and nonlinear time-history analyses.* Hinge behavior does not affect nonlinear modal time-history (FNA) analyses unless the hinges are modeled as links.

The following two basic types of hinges are available.

a) *Uncoupled moment, torsion, axial force and shear hinges*
b) *Coupled P-M2-M3 hinges which yield based on the interaction of axial force and bi-axial bending moments at the hinge location. Subsets of these hinges may include P-M2, P-M3, and M2-M3 behavior.*

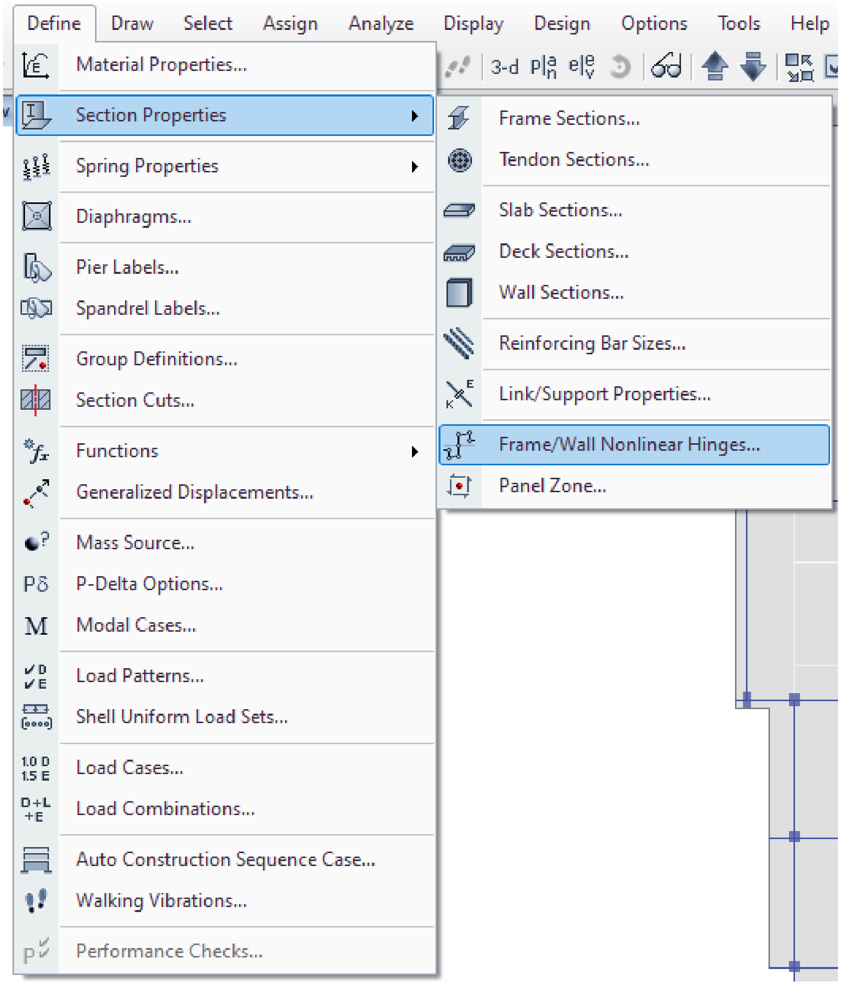
Fiber hinges P-M2-M3 can be defined, which are a collection of material points over the cross section. Each point represents a tributary area and has its own stress-strain curve. Plane sections are assumed to remain planar for the section, which ties together the behavior of the material points. Fiber hinges are often more realistic than force-moment hinges, but are more computationally intensive.

More than one type of frame hinge can exist at the same location, for example, you might assign both an M3 (moment) and a V2 (shear) hinge to the same end of a frame element. Hinge properties can be computed automatically from the element material and section properties according to ACSE 41-13 criteria.

For reinforced concrete members, the interacting P-M-M hinges can be used to model RC columns, P-M hinges can be used for shear walls and uncoupled moment-rotation hinges can be used for beams. Separate shear hinges can also be used if the nonlinearity in shear behavior also needs to be considered. Alternatively, in cases where the shear behavior is intended to remain elastic, the shear force demands in each member can be manually checked against that member's shear capacity. This can also be automated in CSI ETABS by using the force-controlled shear hinge (with a given shear force capacity) in the member. The hinge results can be checked after the analysis to confirm that shear behavior remained elastic during the whole duration of loading.

In ETABS, the hinges assigned to vertical shear walls are of type "fiber P-M3", and always act at the center of the shell element. When hinges are present in a shear wall shell element, the vertical membrane stress behavior is governed by hinge, while horizontal and shear membrane stress, as well as out-of-plane bending behavior, are governed by the properties of the shell element.

Another classification of hinges available in ETABS is *force-controlled (brittle)* and *deformation-controlled (ductile)* hinges. In force-controlled (brittle) hinges, the hinge capacities, modeling parameters and acceptance criteria of hinge are input in the form of forces. Therefore, the demand-to-capacity (D/C) ratio is also determined in terms of ratios of forces (e.g. ratio of shear demand to shear capacity, or moment demand to moment capacity). These hinges can be used for the performance assessment of brittle elements. For the deformation-controlled (ductile) hinges, the D/C ratios are determined in terms of deformations (e.g. ratio of plastic rotation demand to plastic rotation capacity, or material strain demand to material strain capacity, etc.). These hinges can be used for the performance assessment of ductile elements.



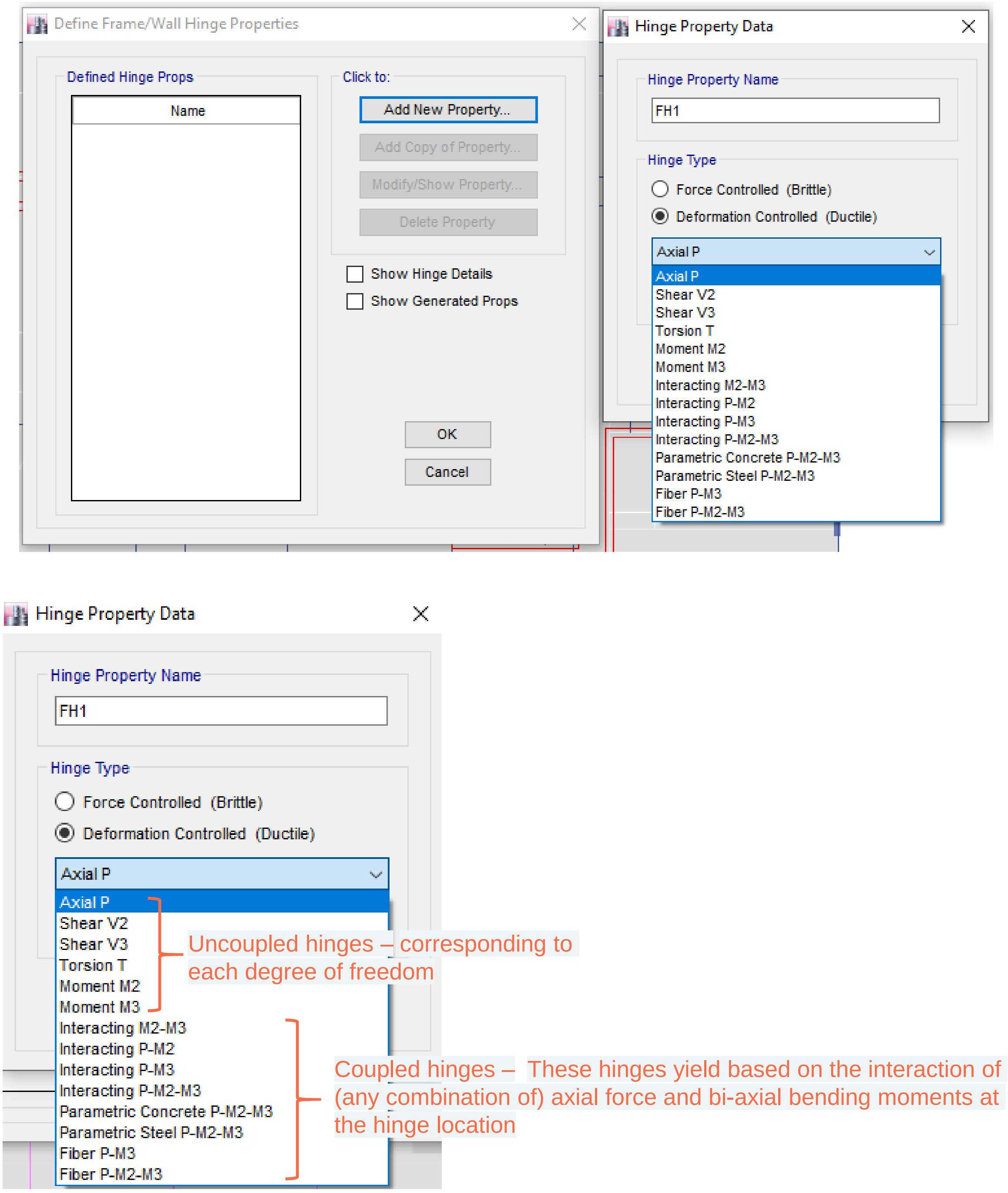


Figure 2-1: The types of inelastic components (hinges) available in CSI ETABS

Before we discuss the types and properties of these hinges, let's have a brief overview of the two most important and widely-used features of all plastic hinges in CSI ETABS. These are the action vs. deformation (backbone) curve and the hysteretic (cyclic) behavior.

2.2. General Action vs. Deformation Curve (for Hinges) in CSI ETABS

For each inelastic component (material, plastic hinge, or link degree of freedom), a uniaxial action vs. deformation curve defines the nonlinear behavior under monotonic loading in the positive and negative directions.

Force- and moment-type hinges are rigid-plastic. *For each force degree of freedom (axial and shear), you may specify the plastic force-displacement behavior. For each moment degree of freedom (bending and torsion) you may specify the plastic moment-rotation behavior. Each hinge property may have plastic properties specified for any number of the six degrees of freedom. The axial force and the two bending moments may be coupled through an interaction surface. Degrees of freedom that are not specified remain elastic.*

Fiber hinges are elastic-plastic and consist of a set of material points, each representing a portion of the frame cross-section having the same material. *In fiber hinges, the force-deflection and moment-rotation curves are not specified, but rather are computed during the analysis from the stress-strain curves of the material points.*

In summary, the action and deformation are an energy conjugate pair as follows:

- For materials, stress vs. strain
- For hinges and multi-linear links, force vs. deformation or moment vs. rotation, depending upon the degree of freedom to which it is applied.

For each model, the uniaxial action-deformation curve is given by a set of points that you define (Figure 2-2). This curve is called the backbone curve, and it can take on almost any shape, with the following restrictions:

- One point must be the origin, (0,0).
- At least one point with positive deformation, and one point with negative deformation, must be defined.
- The deformations of the specified points must increase monotonically, with no two values being equal.
- The action at each point must have the same sign as the deformation (they can be zero).
- The slope given by the last two points specified on the positive deformation axis is extrapolated to infinite positive deformation, or until it reaches zero value. Similarly, the slope given by the last two points specified on the negative deformation axis is extrapolated to infinite negative deformation, or until it reaches zero value.

The given curve defines the action-deformation relationship under monotonic loading. The first slope on either side of the origin is elastic; the remaining segments de fine plastic deformation. If the deformation re verses, it typically follows the two elastic segments before beginning plastic deformation in the reverse direction, except as described below.

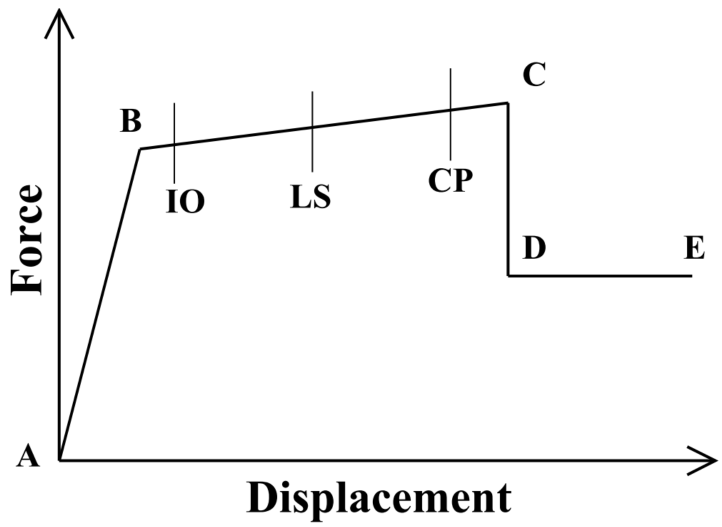


Figure 2-2: The A-B-C-D-E curve for Force vs. Displacement. The same type of curve is used for all hinges (For fiber hinges, it represents the material stress-strain curve. For uncoupled flexural hinges, it represents the moment-rotation curve).

2.3. General Hysteresis Models Available (for Hinges) in CSI ETABS

Hysteresis is the process of energy dissipation through deformation (displacement), as opposed to viscosity which is energy dissipation through deformation rate (velocity). Hysteresis is typical of solids, whereas viscosity is typical of fluids, although this distinction is not rigid.

Hysteretic behavior may affect nonlinear static and nonlinear time-history load cases that exhibit load reversals and cyclic loading. Monotonic loading is not affected.

Several different hysteresis models are available in CSI ETABS to describe the behavior of different types of materials. For the most part, these differ in the amount of energy they dissipate in a given cycle of deformation, and how the energy dissipation behavior changes with an increasing amount of deformation.

Each hysteresis model may be used for the following purposes:

- *Material stress-strain behavior*, affecting frame fiber hinges and layered shell elements that use the material.
- *Single degree-of-freedom frame hinges*, such as M3 (moment) or P (axial) hinges. Interacting hinges, such as P-M3 or P-M2-M3, currently use the isotropic model.
- *Link/support elements* of type multi-linear plasticity.

The hysteresis models available in ETABS are introduced next. Typical for all models, cyclic loading behaves as follows:

- Initial loading in the positive or negative direction follows the backbone curve.
- Upon reversal of deformation, unloading occurs along a different path, usually steeper than the loading path. This is often parallel or nearly parallel to the initial elastic slope.

- After the load level is reduced to zero, continued reversal of deformation causes reverse loading along a path that eventually joins the backbone curve on the opposite side, usually at a deformation equal to the maximum previous deformation in that direction or the opposite direction.

2.3.1. Elastic Hysteresis Model

This model represents a behavior which is *nonlinear but also elastic*. This means that the material always loads and unloads along the backbone curve, and no energy is dissipated. This behavior is illustrated in Figure 2-3. This same backbone curve is used in the figures for all subsequent models, except that the concrete model uses only the positive portion of the curve, with the negative portion being defined separately.

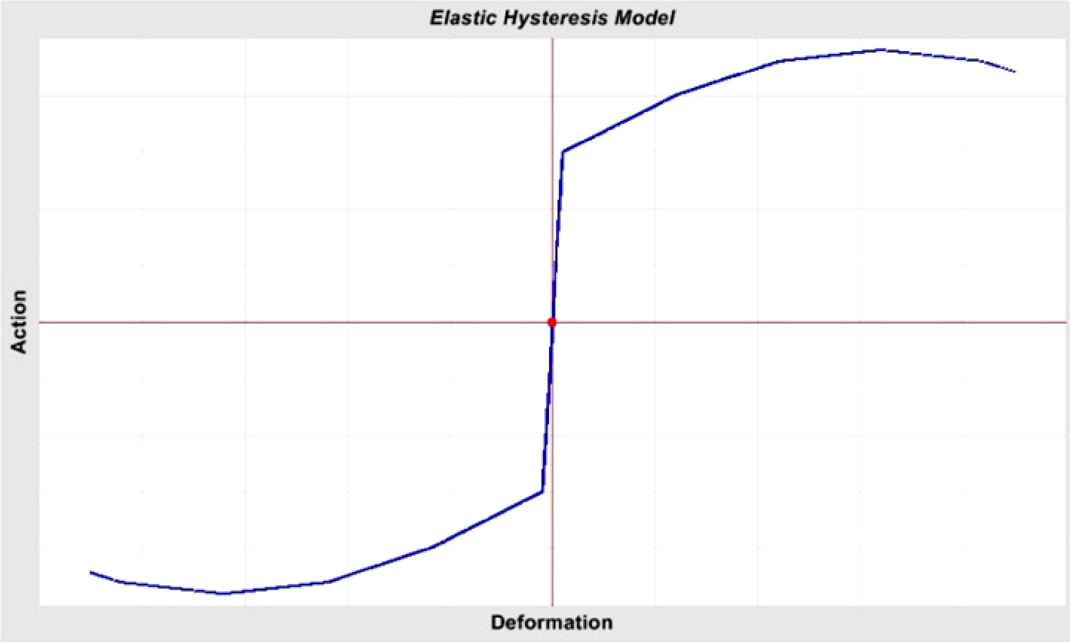


Figure 2-3: Elastic hysteresis model under increasing cyclic load - No energy dissipation showing the backbone curve used for all hysteresis figures

2.3.2. Kinematic Hysteresis Model

This model is based upon kinematic hardening behavior that is commonly observed in metals, and *it is the default hysteresis model for all metal materials* in the program. This model dissipates a significant amount of energy, and is appropriate for ductile materials.

Under the rules of kinematic hardening, plastic deformation in one direction “pulls” the curve for the other direction along with it. No additional parameters are required for this model.

Upon unloading and reverse loading, the curve follows a path made of segments parallel to and of the same length as the previously loaded segments and their opposite-direction counterparts until it rejoins the backbone curve when loading in the opposite direction. This behavior is shown in Figure 2-4 for cycles of increasing deformation.

When you define the points on the multi-linear curve, you should be aware that symmetrical pairs of points will be linked, even if the curve is not symmetrical. This gives you some control over the shape of the hysteretic loop.

The kinematic model forms the basis for several of the other model described below, including Takeda, degrading, and BRB hardening.

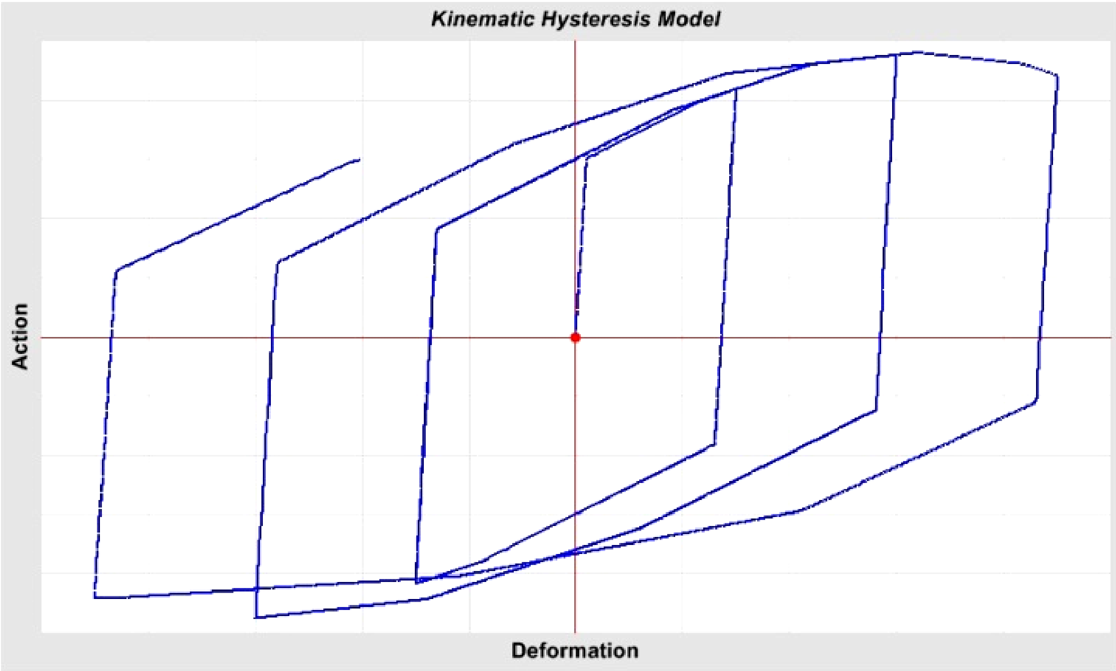


Figure 2-4: Kinematic hysteresis model under increasing cyclic load

2.3.3. Degrading Hysteresis Model

This model is very similar to the Kinematic model but *uses a degrading hysteretic loop* that accounts for decreasing energy dissipation and unloading stiffness with increasing plastic deformation.

Two measures are used for plastic deformation:

- *Maximum plastic deformation* in each the positive and negative directions.
- *Accumulated plastic deformation*, which is the absolute sum of each increment of positive or negative plastic deformation. Plastic deformation is that which does not occur on the two elastic segments of the action-deformation curve.

Accumulated plastic deformation can occur under cyclic loading of constant amplitude, and can be used to represent fatigue.

For this model, the following parameters are required:

- Separately for positive and negative deformations
  - Initial energy factor at yield, $f0$, usually 1.0
  - Energy factor at moderate deformation, $f1$
  - Energy factor at maximum deformation, $f2$

- Moderate deformation level, $x1$, as a multiple of deformation scale factor
- Maximum deformation level, $x2$, as a multiple of deformation scale factor
- Accumulated deformation weighting factor, a

- Stiffness degradation weighting factor, s
- Larger-smaller weighting factor, w, usually 0.0

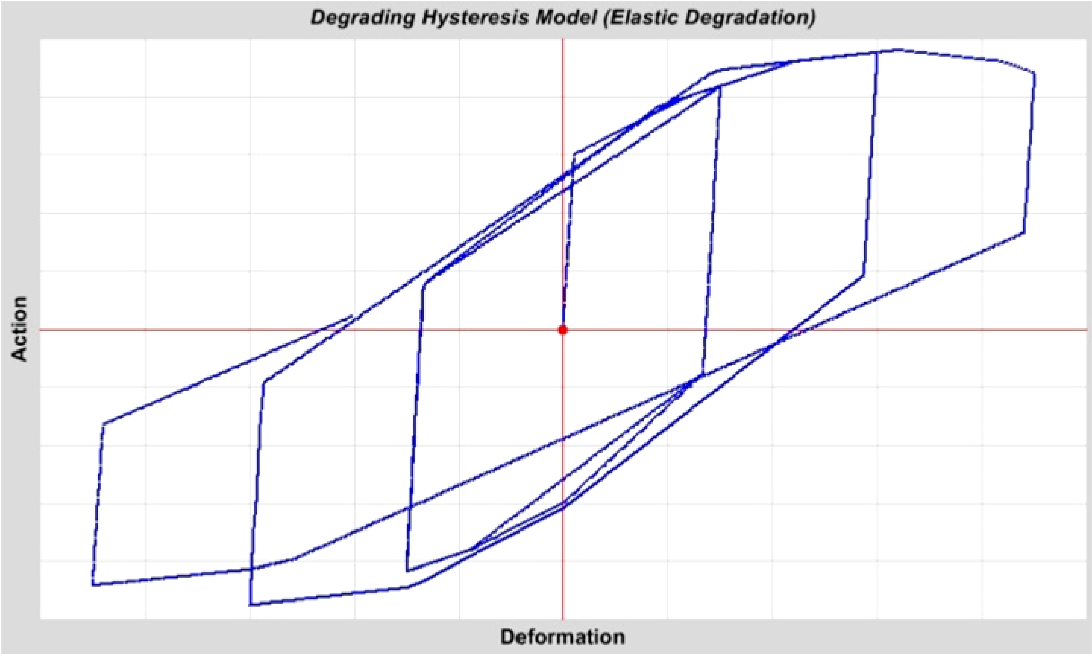


Figure 2-5: Degrading hysteresis model under increasing cyclic load exhibiting elastic degradation (s = 0.0)

Note: When used in a material or link property, the deformation scale factors $x1$ and $x2$ are the yield deformations. When used in a hinge property, the deformation scale factors are those directly specified for that property. The deformation scale factors may be different in the positive and negative directions.

The energy factors represent the area of a degraded hysteresis loop divided by the energy of the non-degraded loop, such as for the kinematic model. For example, energy factor of 0.3 means that a full cycle of deformation would only dissipate 30% of the energy that the non-degraded material would. The energy factors must satisfy $1.0 \geq f0 \geq f1 \geq f2 \geq 0.0$. The deformation levels must satisfy $1.0 < x1 < x2$.

All weighting factors may take any value from 0.0 to 1.0, inclusive. Because the accumulated plastic deformation is constantly increasing, it is recommended that the weighting factor a generally be small or zero.

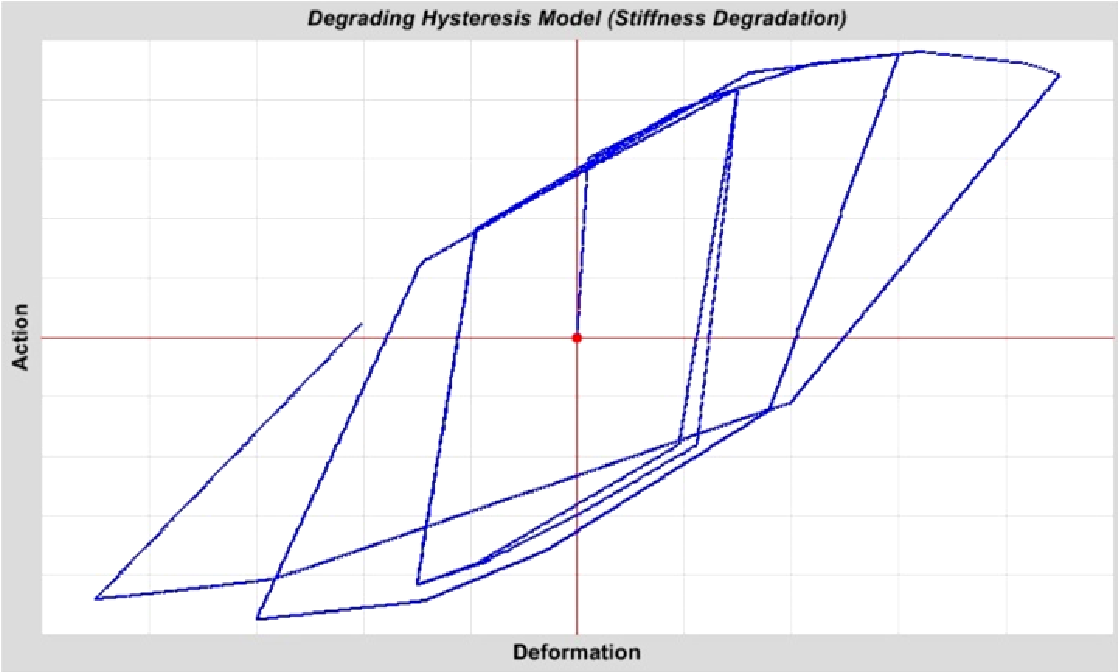


Figure 2-6: Degrading hysteresis model under increasing cyclic load exhibiting elastic degradation (s = 1.0)

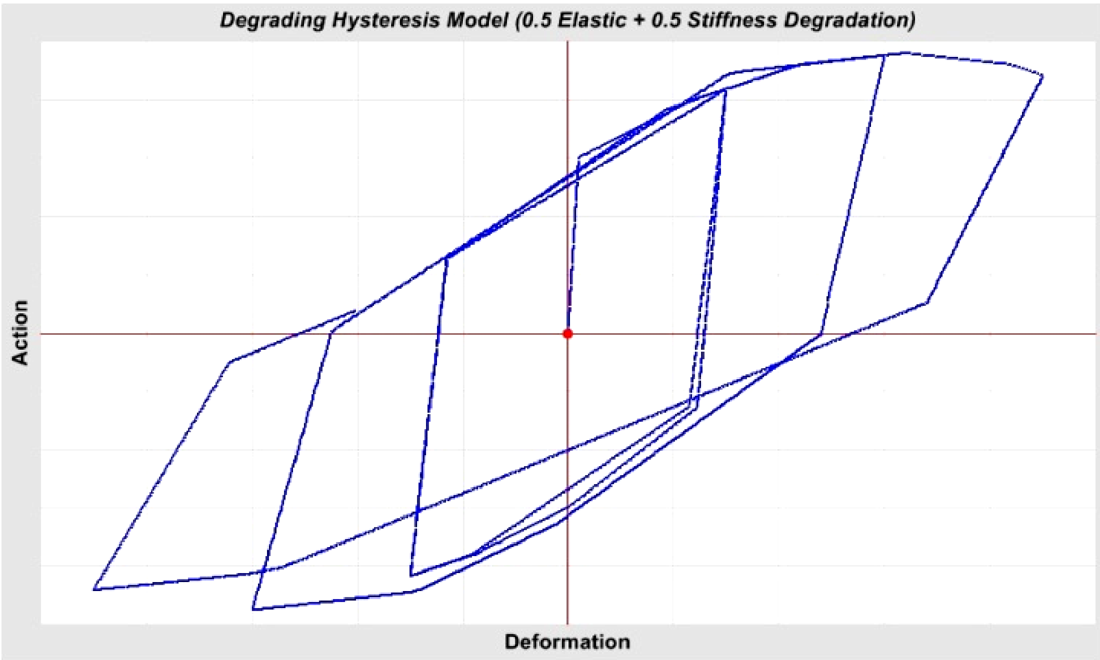


Figure 2-7: Degrading hysteresis model under increasing cyclic load exhibiting elastic degradation (s = 0.5)

Degradation does not occur during monotonic loading. However, upon load reversal, the curve for unloading and reverse loading is modified according to the energy factor computed for the last deformation increment. This is done by squeezing, or flattening, the curve toward the diagonal line that connects the two points of maximum positive and negative deformation.

This squeezing is scaled to achieve the desired decrease in energy dissipation. The scaling can occur in two directions:

- *Parallel to the elastic unloading line, called elastic degradation.*
- *Parallel to the horizontal axis, called stiffness degradation.*

The amount of scaling in each direction is controlled by the stiffness degradation weighting parameter, s. For $s = 0.0$, all degradation is of elastic type. For $s = 1.0$, all degradation is of stiffness type. For intermediate values, the degradation is apportioned accordingly.

While the deformation and individual energy levels are computed separately for the positive and negative directions, the final energy level is a single parameter that affects the shape of the hysteresis loop in both directions.

Note that if all the energy factors are equal to 1.0, this model degenerates to the kinematic hysteresis model.

Figures 2-5, 2-6 and 2-7 show the shape of the hysteresis loop for elastic degradation, stiffness degradation, and a mixture with a stiffness degradation factor of $s = 0.5$. Each of these three cases dissipates the same amount of energy for a given cycle of loading, and less than the energy dissipated for the equivalent kinematic model.

2.3.4. Takeda Hysteresis Model

This model is very similar to the kinematic model, but uses a degrading hysteretic loop based on the Takeda model, as described in Takeda, Sozen, and Nielsen (1970). *This simple model requires no additional parameters, and is more appropriate for reinforced concrete than for metals.* Less energy is dissipated than for the kinematic model.

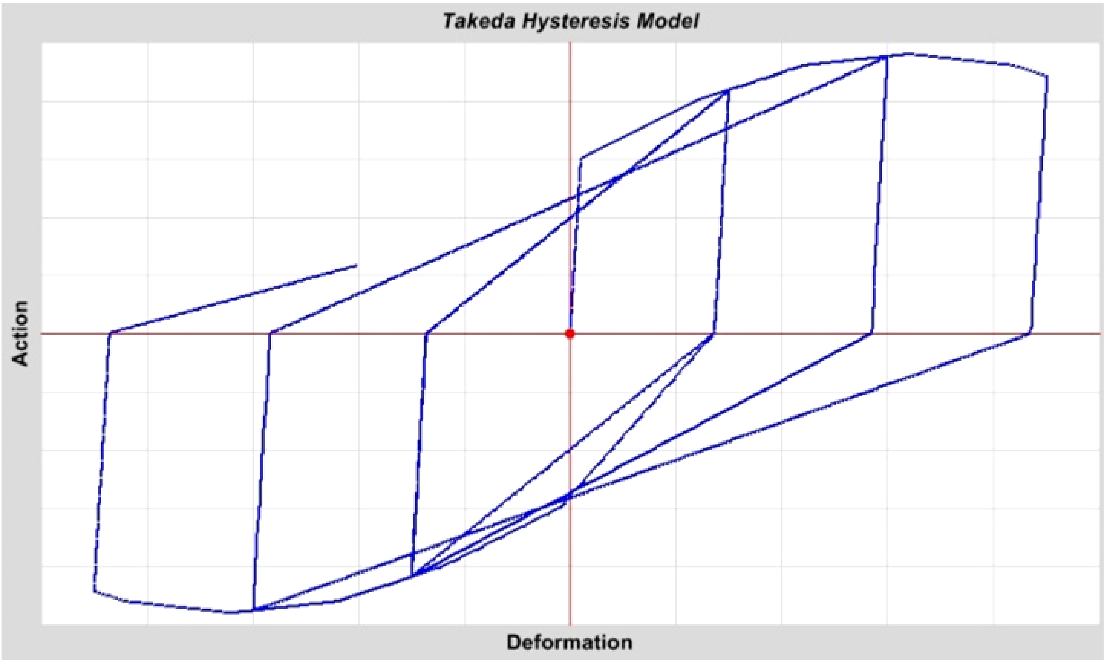


Figure 2-8: Takeda hysteresis model under increasing cyclic load.

Unloading is along the elastic segments similar to the kinematic model. When reloading, the curve follows a secant line to the backbone curve for loading in the opposite direction. The target point for this secant is at the maximum deformation that occurred in that direction under previous load cycles. This results in a decreasing amount of energy dissipation with larger deformations. Unloading is along the elastic segments.

2.3.5. Pivot Hysteresis Model

This model is similar to the Takeda model, but *has additional parameters to control the degrading hysteretic loop*. It is particularly well suited for reinforced concrete members, and is based on the observation that unloading and reverse loading tend to be directed toward specific points, called *pivots points*, in the action-deformation plane. The most common use of this model is for moment-rotation. This model is fully described in Dowell, Seible, and Wilson (1998). *This model is not intended for unreinforced concrete*.

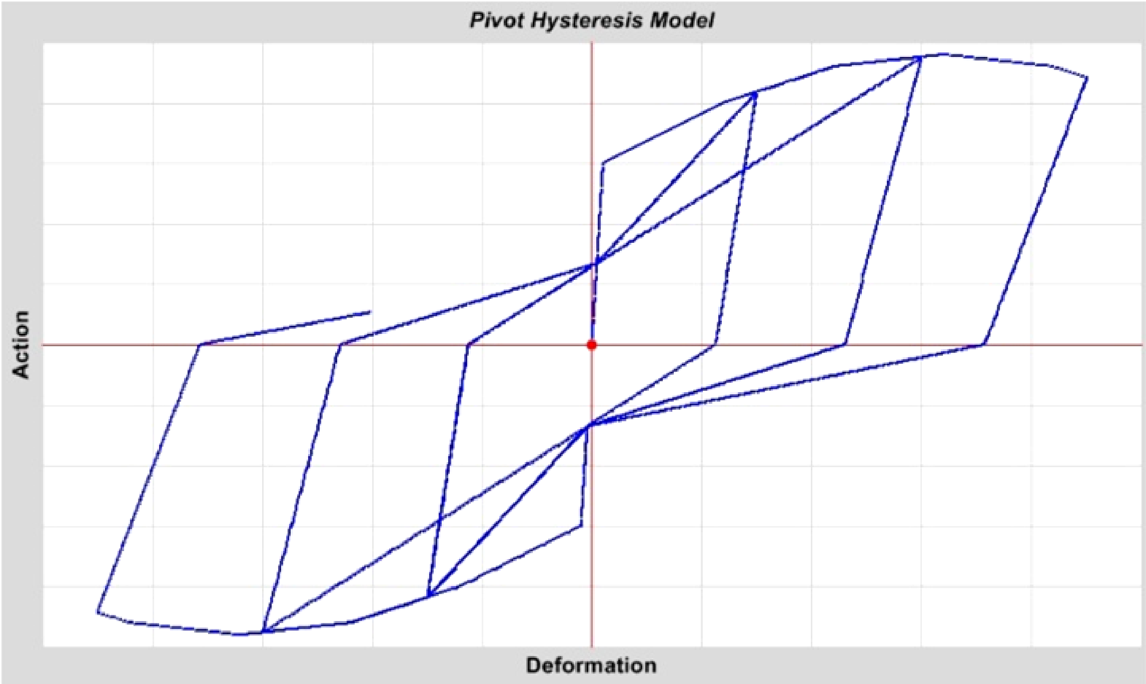


Figure 2-9: Pivot hysteresis model under increasing cyclic load.

The following additional parameters are specified for the Pivot model:

- α_1, which locates the pivot point for unloading to zero from positive force. Unloading occurs toward a point on the extension of the positive elastic line, but at a negative force value of α_1 times the positive yield force.
- α_2, which locates the pivot point for unloading to zero from negative force. Unloading occurs toward a point on the extension of the negative elastic line, but at a positive force value of α_2 times the negative yield force.
- β_1, which locates the pivot point for reverse loading from zero toward positive force. Reloading occurs toward a point on the positive elastic line at a force value of β_1 times the positive yield force, where $0 <$

$\beta_1 < 1.0$. Beyond that point, loading occurs along the secant to the point of maximum previous positive deformation on the backbone curve.

- β_2, which locates the pivot point for reverse loading from zero toward negative force. Reloading occurs toward a point on the negative elastic line at a force value of β_2 times the negative yield force, where $0 < \beta_2 < 1.0$. Beyond that point, loading occurs along the secant to the point of maximum previous negative deformation on the backbone curve.
- η, which determines the amount of degradation of the elastic slopes after plastic deformation, where $0 < \eta < 1.0$

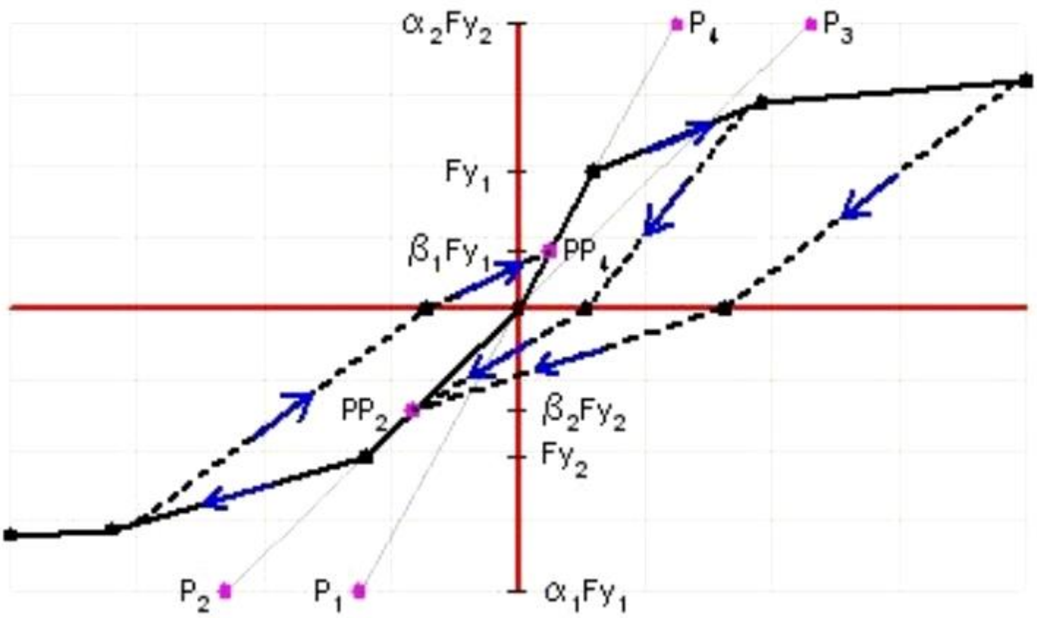


Figure 2-10: Pivot hysteresis model parameters.

2.3.6. Concrete Hysteresis Model

This model is intended for unreinforced concrete and similar materials, and is the default model for concrete and masonry materials in the program. Tension and compression behavior are independent and behave differently. The force-deformation (stress-strain) curve is used to determine the sign of compression, which can be positive or negative. The point having the largest absolute value of stress or force is considered to be in compression, so that the sign of compression can be either positive or negative. Likewise, the concrete model can also be used to represent a tension-only material whose behavior is similar to concrete in compression.

This model is primarily intended for axial behavior, but can be applied to any degree of freedom. *Reinforced concrete is better modeled using the pivot, degrading, or Takeda models*.

A non-zero force-deformation curve should always be defined for compression. The force-deformation curve for tension may be all zero, or it may be non-zero provided that the maximum force value is of smaller magnitude than that for the compression side.

A single parameter, the energy degradation factor f, is specified for this model. This value should satisfy $0.0 < f < 1.0$. A value of $f = 0.0$ is equivalent to a clean gap when unloading from compression and dissipates the least amount of energy. A value of $f = 1.0$ is dissipates the most energy and could be caused by rubble filling the gap when unloading from compression.

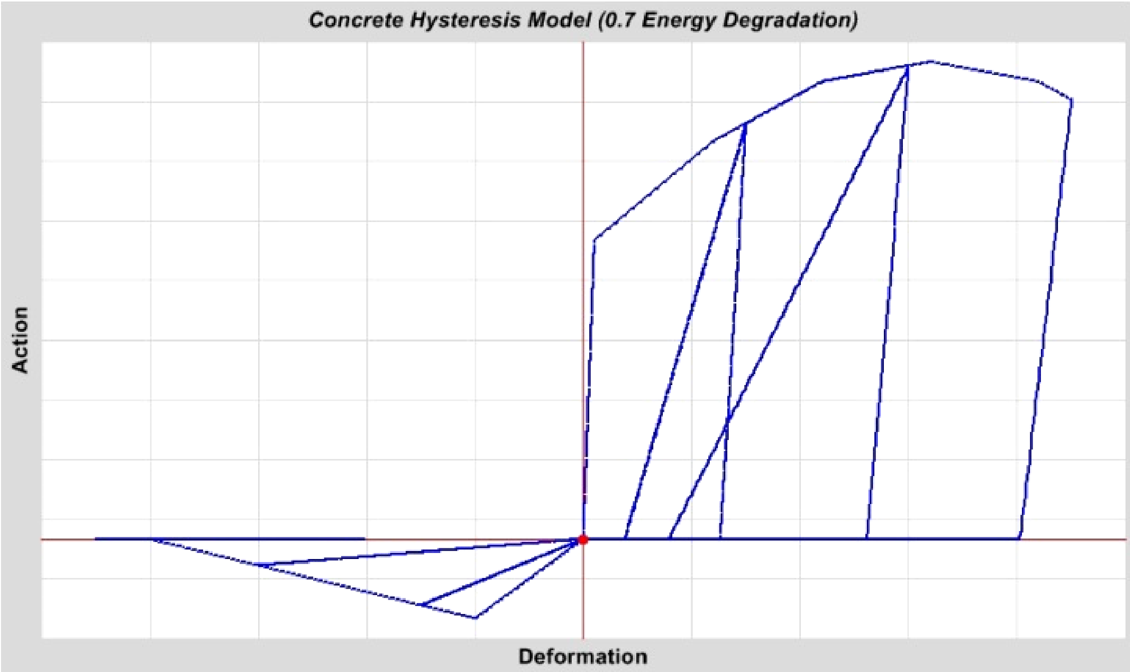


Figure 2-11: Concrete hysteresis model under increasing cyclic load with compression as positive and energy factor f = 0.7.

Compression behavior is modeled as follows:

- Initial loading is along the backbone curve.
- Unloading to zero occurs along a line nearly parallel to the compression elastic line. The line is actually directed to a pivot point on the extension of the compressive elastic line, located so that the unloading slope at maximum compressive force has half the stiffness of the elastic loading line.
- At zero force, reverse loading toward tension occurs at zero force.
- Sub sequent loading in compression occurs along the previous unloading line if the energy factor $f = 0.0$, and along the secant from the origin to the point of maximum previous compressive deformation if the energy factor is 1.0. An intermediate secant from the horizontal axis is used for other values of f.

Tension behavior, if non-zero, is modeled as follows:

- Initial loading is along the backbone curve
- Unloading occurs along a secant line to the origin.
- Subsequent loading occurs along the unloading secant from the origin to the point of maximum previous tensile deformation.

See Figure 2-11 for an example of this behavior for an energy degradation factor of $f = 0.7$.

2.3.7. BRB Hardening Hysteresis Model

This model is similar to the kinematic model, but accounts for the increasing strength with plastic deformation that is typical of bucking-restrained braces, causing the backbone curve, and hence the hysteresis loop, to progressively grow in size. It is intended primarily for use with axial behavior, but can be applied to any degree of freedom.

Two measures are used for plastic deformation:

- Maximum plastic deformation in each the positive and negative directions.
- Accumulated plastic deformation, which is the absolute sum of each increment of positive or negative plastic deformation. Plastic deformation is that which does not occur on the two elastic segment of the force-deformation curve.

Accumulated plastic deformation can occur under cyclic loading of constant amplitude.

For this model, the following parameters are required separately for tension (positive) and compression (negative) deformations.

- Hardening factor at maximum deformation, h, where $h \geq 1.0$.
- Maximum plastic deformation level at full hardening, $x2$, as a multiple of deformation scale factor.
- Maximum accumulated plastic deformation level at full hardening, $x4$, as a multiple of deformation scale factor.
- Accumulated deformation weighting factor, a, where $0 \leq a \leq 1.0$.

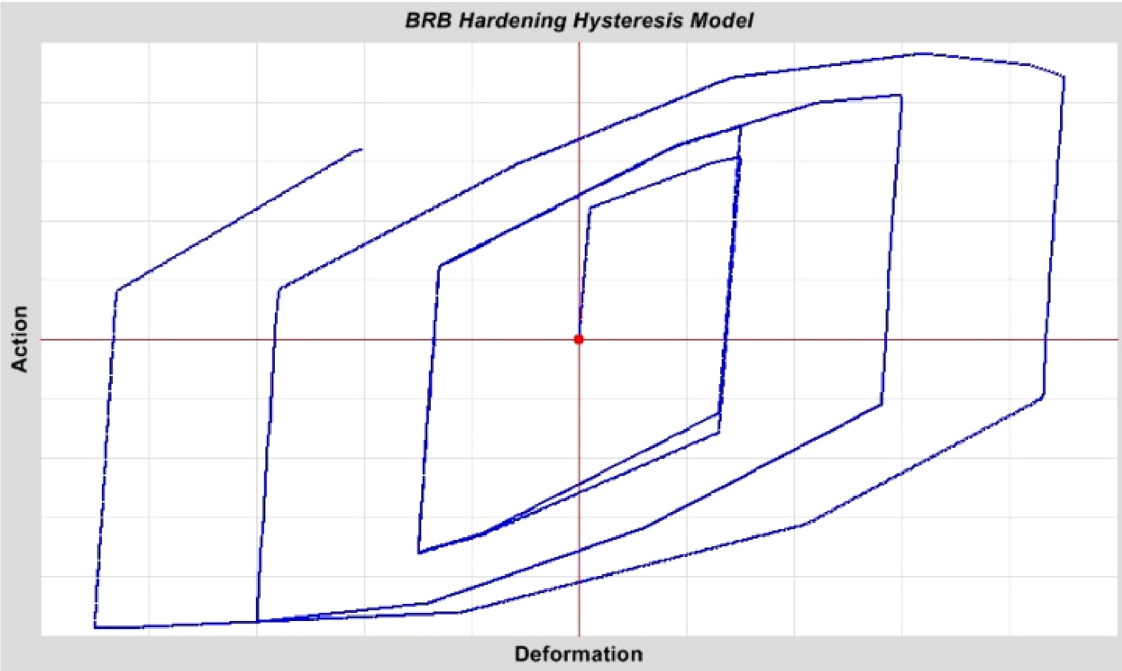


Figure 2-12: BRB hardening hysteresis model under increasing cyclic load with hardening factor h = 1.5

Note: When used in a material or link property, the deformation scale factors are the yield deformations, with $x2 > 1.0$ and $x4 > 1.0$. When used in a hinge property, the deformation scale factors are those directly specified for that property, with $x2 > 0.0$ and $x4 > 0.0$. The deformation scale factors may be different in the positive and negative directions.

The hardening factors scale the size of the backbone curve and hysteresis loop in the action (stress/force/moment) direction. Because the accumulated plastic deformation is constantly increasing, it is recommended that the weighting factor "a" generally be small or zero.

Degradation does not occur during monotonic loading. However, upon load reversal, the curve for unloading and reverse loading in the opposite direction is modified according to the hardening factor computed for the last deformation increment. This is done by scaling the action values in that direction, including the backbone curve for further loading.

Notes:

- Positive deformation and the corresponding hardening parameters only affect the negative strength, and vice versa.
- If the hardening factor is equal to 1.0, this model degenerates to the kinematic hysteresis model.

This behavior is illustrated in Figure 2-12.

2.3.8. Isotropic Hysteresis Model

This model is, in a sense, the opposite of the kinematic model. Plastic deformation in one direction "pushes" the curve for the other direction away from it, so that *both directions increase in strength simultaneously*. Unlike the BRB hardening model, the backbone curve itself does not increase in strength, only the unloading and reverse loading behavior. Matching pairs of points are linked. No additional parameters are required for this model.

Unloading and reverse loading occur along a path parallel to the elastic line until the magnitude of the action in the reverse direction equals that of backbone curve at the same amount of deformation in the reverse direction, and then continues along a horizontal se cant to the back bone curve.

When you define the points on the multi-linear curve, you should be aware that symmetrical pairs of points will be linked, even if the curve is not symmetrical. This gives you some control over the shape of the hysteretic loop.

This model dissipates the most energy of all the models. This behavior is illustrated in Figure 2-13.

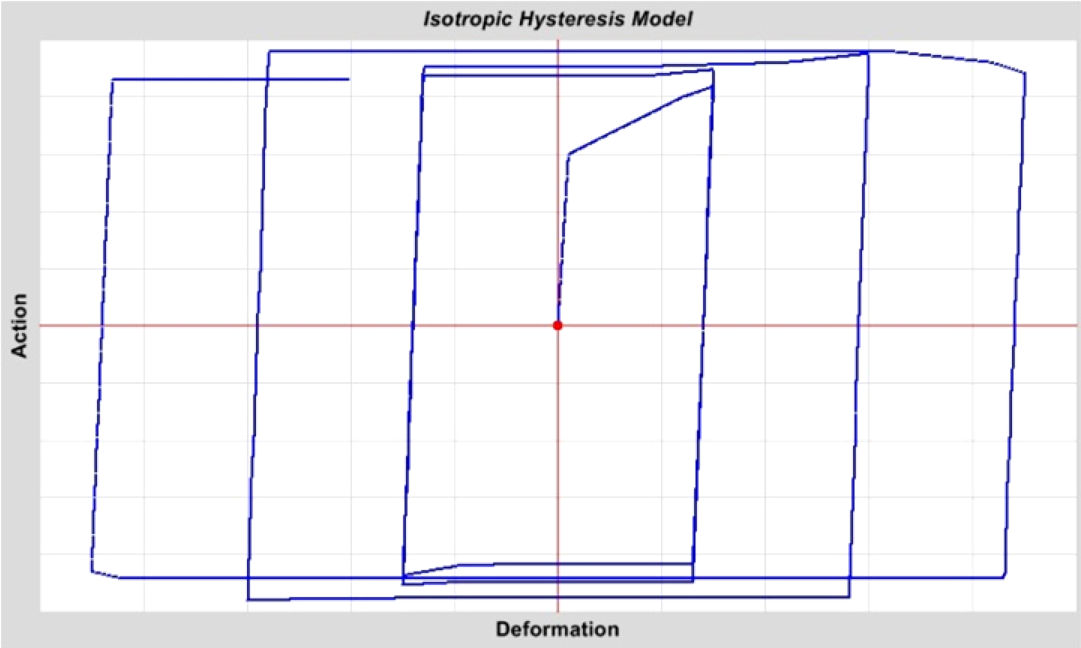


Figure 2-13: Isotropic hysteresis model under increasing cyclic load.

2.4. Hinge Properties (Applicable to All Hinges)

A hinge property is a named set of nonlinear properties that can be assigned to points along the length of one or more Frame elements. The following properties need to be defined for each plastic hinge.

2.4.1. Hinge Length

Each plastic hinge is modeled as a discrete point hinge. All plastic deformation, whether it be displacement or rotation, occurs within the point hinge. This means you must assume a length for the hinge over which the plastic strain or plastic curvature is integrated.

There is no easy way to choose this length, although guidelines are given in FEMA-356 and ASCE 41-13. Typically it is a fraction of the element length, and is often on the order of the depth of the section, particularly for moment-rotation hinges.

You can approximate plasticity that is distributed over the length of the element by inserting many hinges. For example, you could insert ten hinges at relative locations within the element of 0.05, 0.15, 0.25, ..., 0.95, each with deformation properties based on an assumed hinge length of one-tenth the element length. Of course, adding more hinges will add more computational cost, so this should only be done where needed.

For force/moment-type hinges, elastic deformation occurs along the entire length of the Frame element and is not affected by the presence of the hinges. For fiber hinges, elastic behavior along the hinge length is determined from the hinge material stress-strain curves, and the elastic properties of the frame element are ignored within the hinge length. For this reason, the hinge length should not exceed the length of frame element.

2.4.2. Basic Plastic Deformation (Backbone) Curve and Scale Factors

As discussed in Section 2.2, for each force or moment degree of freedom, you define a force-displacement curve that gives the yield value and the plastic deformation following yield. This is done in terms of a curve with values at five points, A-B-C-D-E, as shown in previous Figure. You may specify a symmetric curve, or one that differs in the positive and negative direction.

The shape of this curve is shown in Figure 2-14.

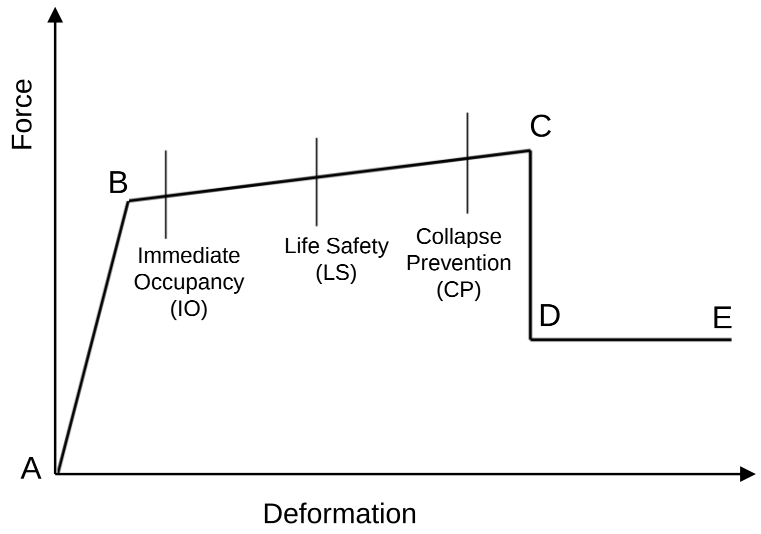


Figure 2-14: The basic force-deformation curve for defining the plastic hinges in ETABS.

You can use any shape you want. The following points should be noted:

- Point A is always the origin.
- Point B represents yielding. No deformation occurs in the hinge up to point B, regardless of the deformation value specified for point B. The displacement (rotation) at point B will be subtracted from the deformations at points C, D, and E. Only the plastic deformation beyond point B will be exhibited by the hinge.
- Point C represents the ultimate capacity for push over analysis. However, you may specify a positive slope from C to D for other purposes.
- Point D represents a residual strength for pushover analysis. However, you may specify a positive slope from C to D or D to E for other purposes.
- Point E represent total failure. Beyond point E the hinge will drop load down to point F (not shown) directly below point E on the horizontal axis. If you do not want your hinge to fail this way, be sure to specify a large value for the deformation at point E.

Prior to reaching point B, all deformation is linear and occurs in the Frame element itself, not the hinge. Plastic deformation beyond point B occurs in the hinge in addition to any elastic deformation that may occur in the element.

When the hinge unloads elastically, it does so without any plastic deformation, i.e., parallel to slope A-B.

When defining the hinge force-deformation (or moment-rotation) curve, you may enter the force and deformation values directly, or you *may enter normalized values and specify the scale factors* that you used to normalized the curve.

In the most common case, the curve would be normalized by the yield force (moment) and yield displacement (rotation), so that the normalized values entered for point B would be (1, 1). However, you can use any scale factors you want. They do not have to be yield values.

Remember that any deformation given from A to B is not used. This means that the scale factor on deformation is actually used to scale the plastic deformation from B to C, C to D, and D to E. However, it may still be convenient to use the yield deformation for scaling.

When automatic hinge properties are used, the program automatically uses the yield values for scaling. These values are calculated from the Frame section properties.

2.4.3. Strength Loss

Strength loss is permitted in the hinge properties. However, you should use this feature judiciously. Any loss of strength in one hinge causes load redistribution within the structure, possibly leading to failure of another hinge, and ultimately causing progressive collapse. This kind of analysis can be difficult and time consuming. Furthermore, any time negative stiffnesses are present in the model, the solution may not be mathematically unique, and so may be of questionable value.

Sudden strength loss (steep negative stiffness) is often unrealistic and can be even more difficult to analyze. When an unloading plastic hinge is part of a long beam or column, or is in series with any flexible elastic sub-system, "elastic snap-back" can occur. Here the elastic unloading deflection is larger than, and of opposite sign to, the plastic deformation. This results in the structure deflecting in the direction opposite the applied load. ETABS have a built-in mechanism to deal with snap-back for certain hinges, but this may not always be enough to handle several simultaneous snap-back hinge failures.

Consider carefully what you are trying to accomplish with your analysis. A well designed structure, whether new or retrofitted, should probably not have strength loss in any primary members. If an analysis shows strength loss in a primary member, you may want to modify the design and then reanalyze, rather than trying to push the analysis past the first failure of the primary members. Since you need to re-design anyway, what happens after the first failure is not relevant, since the behavior will become changed.

Limiting Negative Stiffness: To help with convergence, the program limits the negative slope of a hinge to be no stiffer than specified limiting negative stiffness. The limiting stiffness is assigned as a Hinge Overwrite and is specified as a ratio of the elastic stiffness of the Frame element containing the hinge. By default, the limiting negative stiffness ratio is 10% (0.1). This value is suggested to prevent snap-back within the element, although it may still occur in the larger structure. If you need steeper slopes, the limiting negative stiffness ratio can be increased up to a value of 1. However, steeper slopes may increase the possibility of snap-back occurring in the larger structure.

Hinge Overwrite Lengths: You can also assign a Frame Hinge Overwrite that automatically meshes the Frame object around the hinge. When you assign this overwrite, you can specify what fraction of the Frame object length should be used for the element that contains the hinge. For example, consider a Frame object containing one hinge at each end, and one in the middle. If you assign a Frame Hinge Overwrite with a relative length of 0.1, the object will be meshed into five elements of relative lengths 0.05, 0.4, 0.1, 0.4, and 0.05. Each hinge is located at the center of an element with 0.1 relative length, but because two of the hinges fall at the ends of the object, half of their element lengths are not used. Because these elements are shorter than the object, their elastic stiffnesses are larger, and the program will permit larger negative stiffnesses in the hinges.

By reducing the size of the meshed element, you can increase the steepness of the drop-off, although the slope will never be steeper than you originally specified for the hinge. Again, we recommend gradual, realistic slopes whenever possible, unless you truly need to model brittle behavior.

Figures 2-15 and 2-16 show the CSI ETABS form to define the properties (e.g. type, basic force-deformation relationship, hysteresis, and acceptance criteria) of an M3 and V2 plastic hinge (concrete type), respectively.

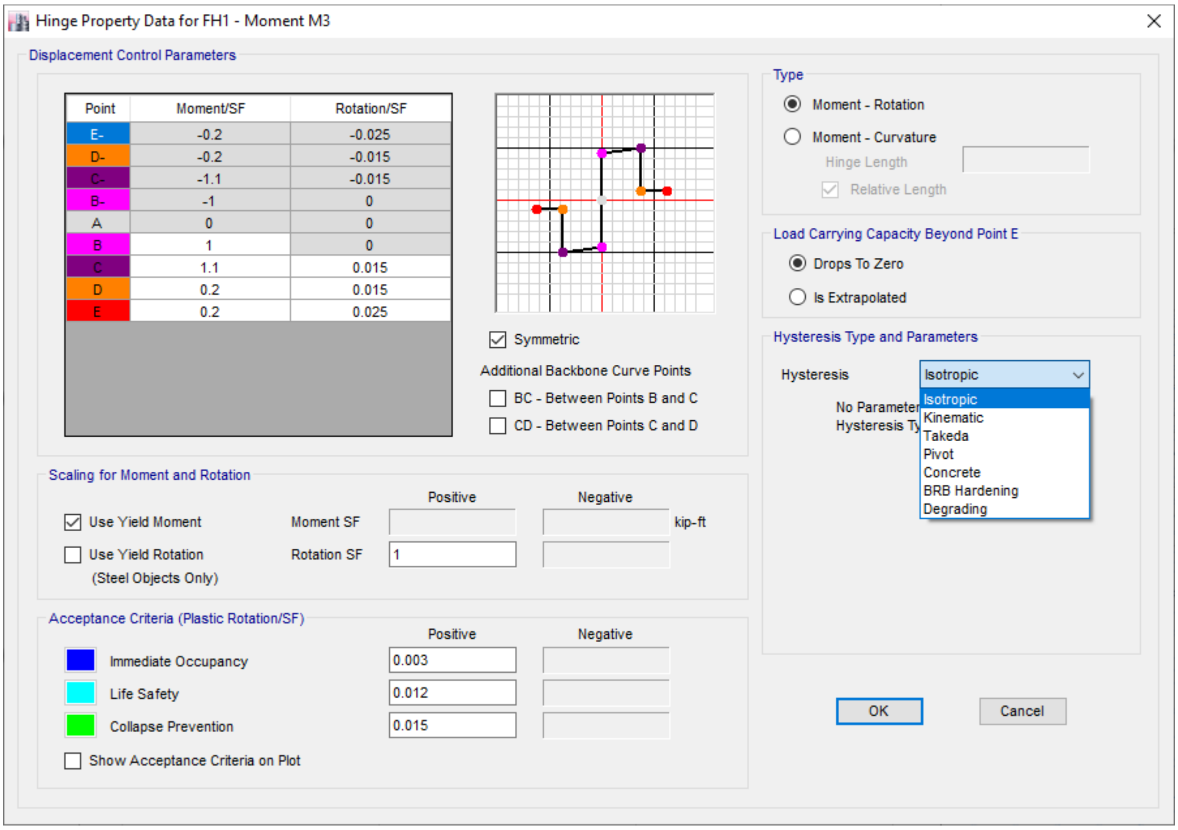


Figure 2-15: CSI ETABS form to define the properties of moment M3 plastic hinge (concrete type).

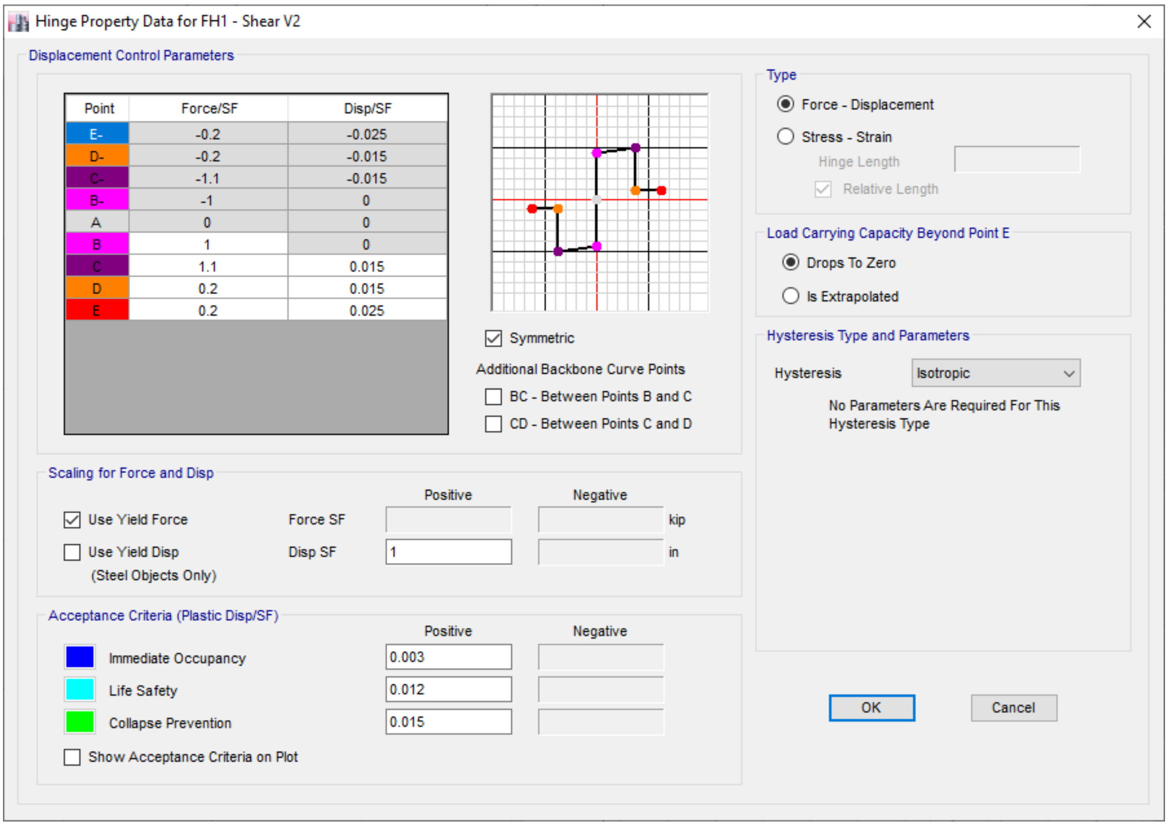


Figure 2-16: CSI ETABS form to define the properties of shear V2 plastic hinge (concrete type).

2.4.4. Acceptance Criteria

Acceptance criteria are the additional deformation values (e.g. material strains in case of inelastic materials and plastic rotations in case of moment-rotation hinges) which need to be specified at points IO (immediate occupancy), LS (life safety), and CP (collapse prevention) [Figure 2-14]. *These are deformation capacities (corresponding to different performance levels e.g., IO, LS and CP) of hinges.* They serve as informational measures (or markers) used for performance-based design and evaluation of members. The program use these capacities to determine D/C ratios corresponding to each performance level after the analysis. *They do not have any effect on the behavior of the structure.*

2.4.5. Types of P-M2-M3 Hinges

Normally the hinge properties for each of the six degrees of freedom are uncoupled from each other. However, you have the option to specify *coupled axial-force/biaxial-moment behavior*. This is called a P-M2-M3 or PMM hinge. *For these hinges, beside the basic force-deformation curve and other inputs, the complete P-M2-M3 interaction surface also needs to be provided.* The following three types of coupled hinges are available in CSI ETABS:

- *Isotropic P-M2-M3 hinge:* This hinge can handle complex and unsymmetrical PMM surfaces and can interpolate between multiple moment-rotation curves. Two-dimensional subsets of the hinge are available. It is limited to isotropic hysteresis, which may not be suitable for some structures.

- *Parametric P-M2-M3 hinge:* This hinge is limited to doubly symmetric section properties and uses a simple parametric definition of the PMM surface. Hysteretic energy degradation can be specified, making it more suitable than the isotropic hinge for extensive cyclic loading.
- *Fiber P-M2-M3 hinge:* This is the most realistic hinge, but may require the most computational resources in terms of analysis time and memory usage. Various hysteresis models are available and they can be different for each material in the hinge.

Figure 2-17 shows the CSI ETABS form to define the properties of interacting P-M2-M3 plastic hinge (concrete type). It can be seen that a set of axial forces and angles are also defined. For each axial force level and for each angle, the basic force-deformation curve needs to be defined. Figure 2-18 shows the CSI ETABS form to define the basic force-deformation relationship (moment-curvature or moment-rotation) of an interacting P-M2-M3 plastic hinge (concrete type). It can be seen that the curve needs to be defined for several levels of axial force and for several angles. Figure 2-19 presents the CSI ETABS form showing the available options to define the PMM interaction surface of an interacting P-M2-M3 plastic hinge (concrete type).The P-M2-M3 interaction surface can either be fully user-defined (as shown in Figure 2-20) or auto-defined using several codes and guidelines as shown.

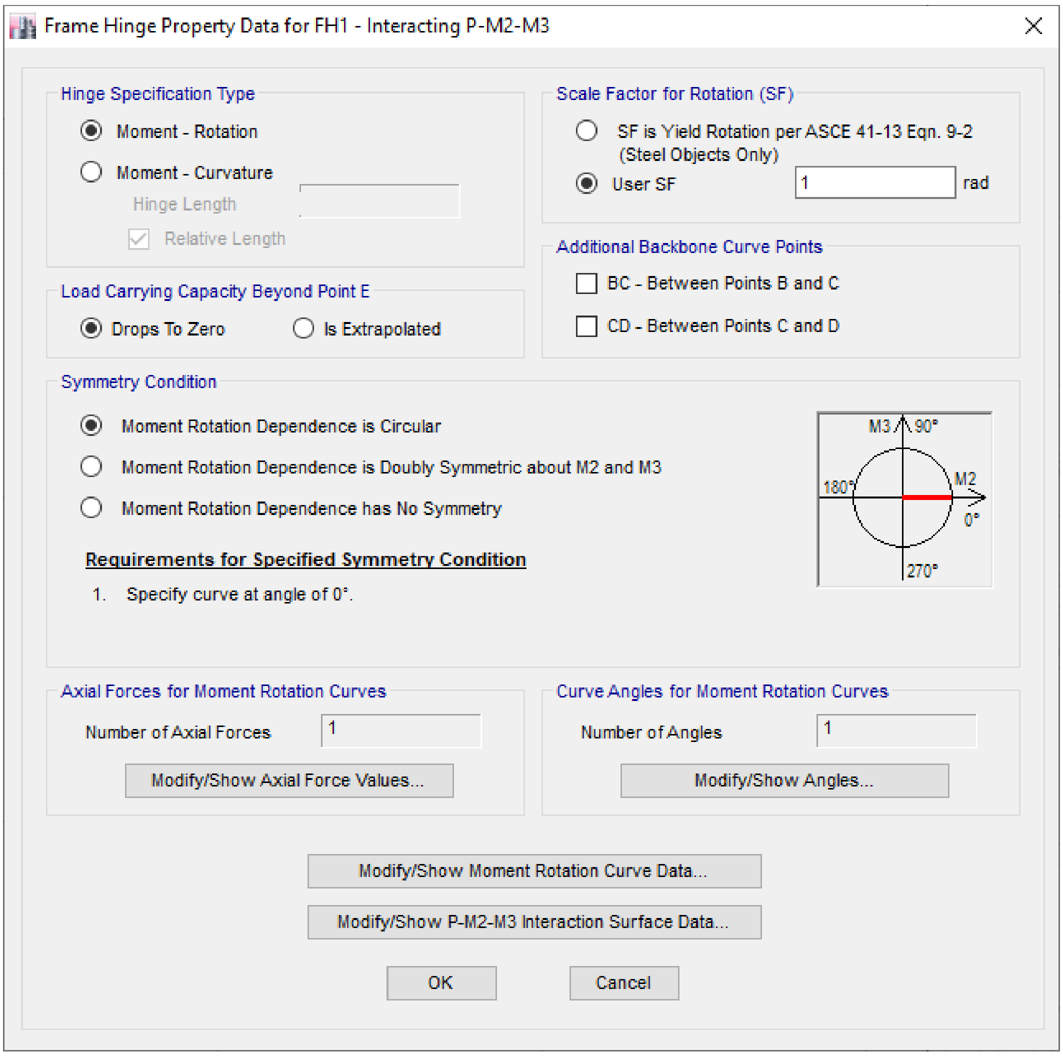


Figure 2-17: CSI ETABS form to define the interacting P-M2-M3 plastic hinge (concrete type).

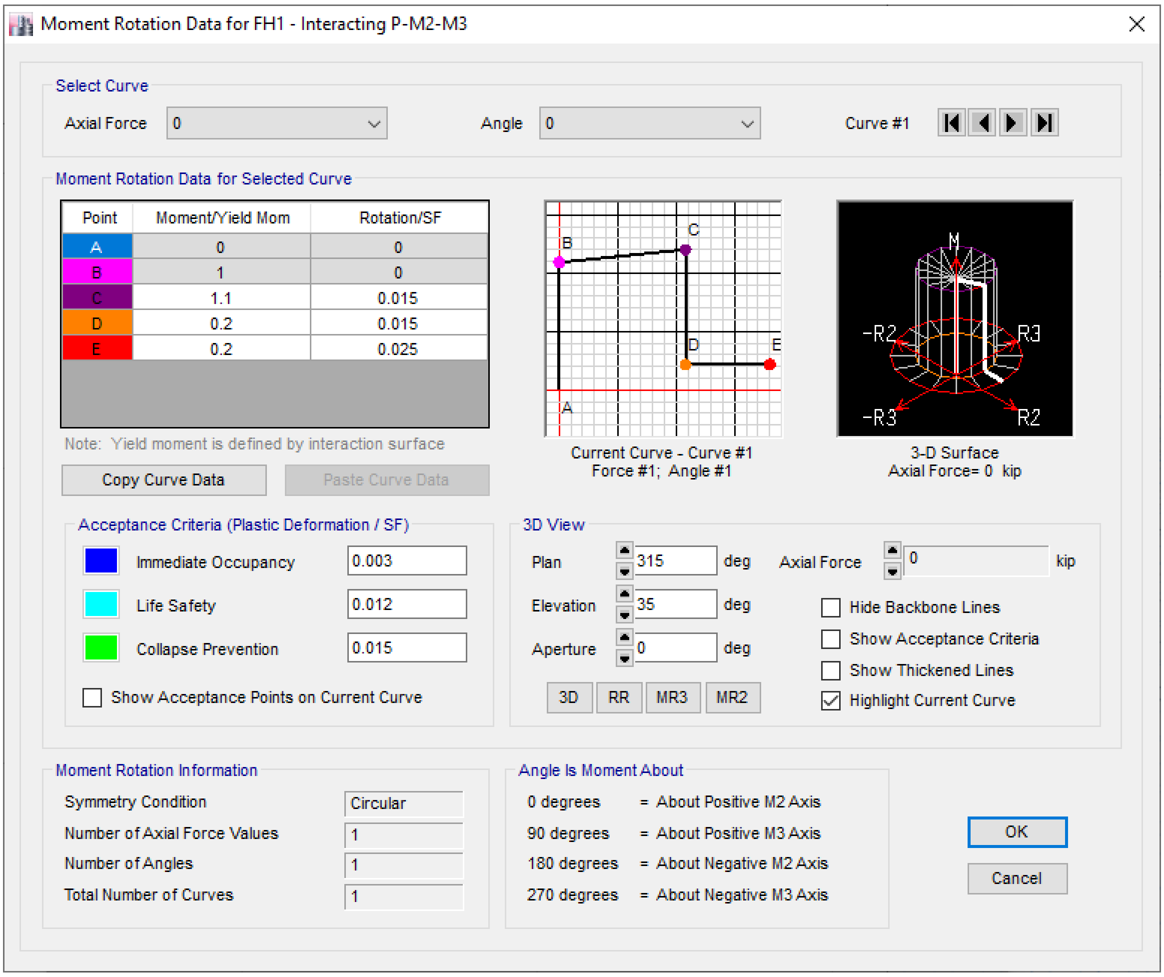


Figure 2-18: CSI ETABS form to define the basic force-deformation relationship (moment-curvature or moment-rotation) of an interacting P-M2-M3 plastic hinge (concrete type). The curve is defined for several levels of axial force and for several angles.

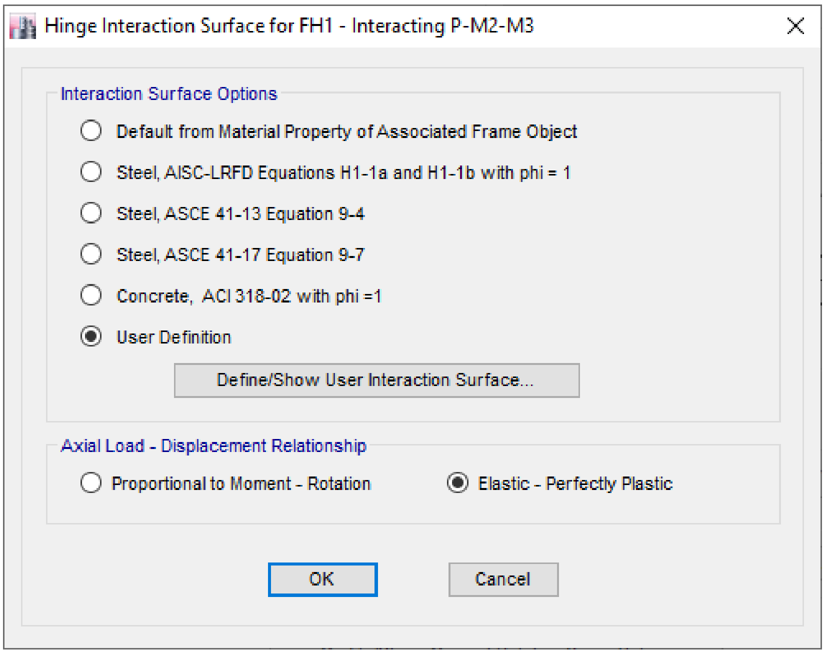


Figure 2-19: CSI ETABS showing the available options to define the PMM interaction surface of an interacting P-M2-M3 plastic hinge (concrete type).

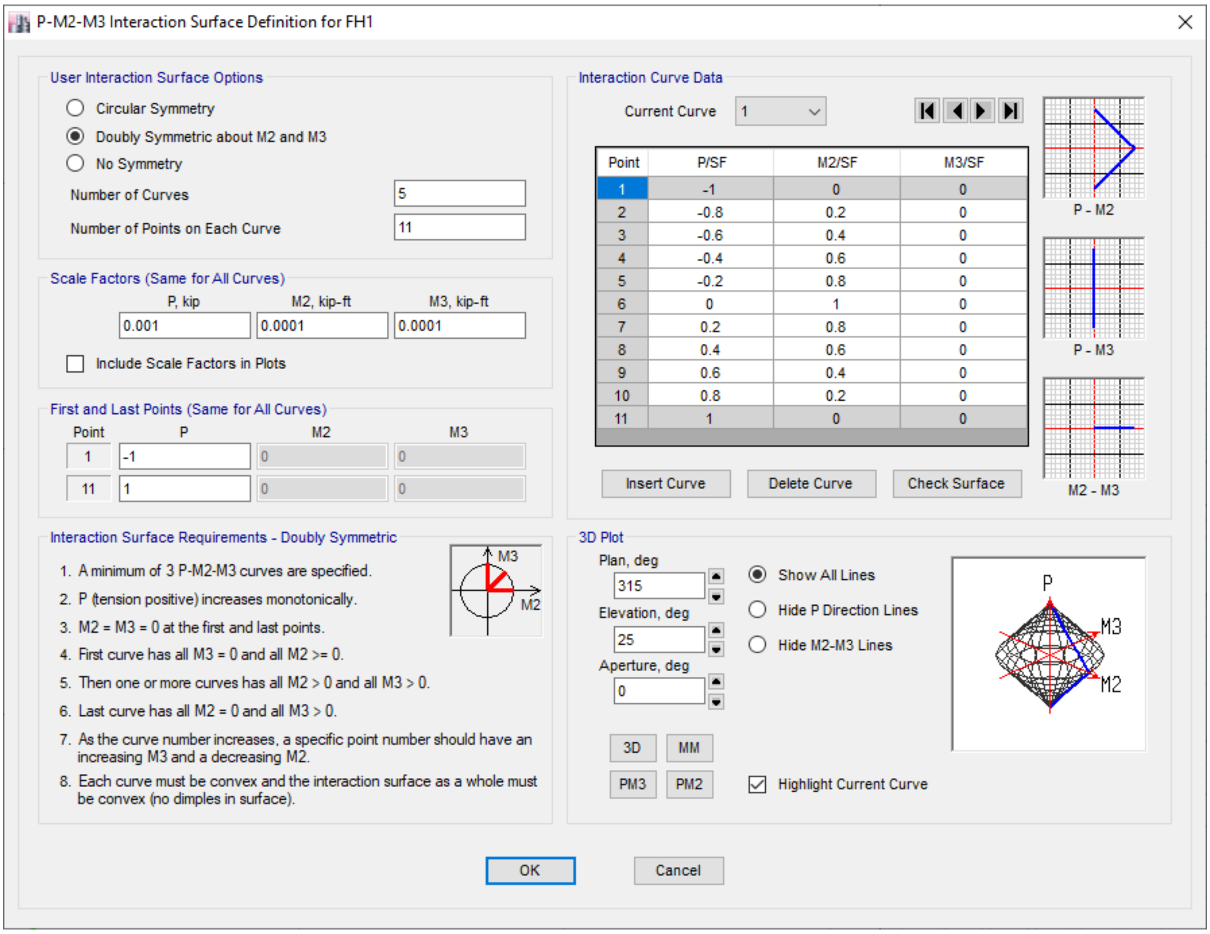


Figure 2-20: CSI ETABS form to define the PMM interaction surface (using the user-defined option) of an interacting P-M2-M3 plastic hinge (concrete type).

2.4.6. Isotropic P-M2-M3 Hinge

This hinge can handle complex and unsymmetrical PMM surfaces and can interpolate between multiple moment-rotation curves. It is limited to isotropic hysteresis, which may not be suitable for some structures.

Three additional coupled hinges are available as subsets of the PMM hinge: P-M2, P-M3, and M2-M3 hinges.

It is important to note that CSI ETABS uses the sign convention where tension is always positive and compression is always negative, regardless of the material being used. This means that for some materials (e.g., concrete) the interaction surface may appear to be upside down.

Interaction (Yield) Surface: For the PMM hinge, you specify an interaction (yield) surface in three-dimensional P-M2-M3 space that represents where yielding first occurs for different combinations of axial force P, minor moment M2, and major moment M3.

The surface is specified as a set of P-M2-M3 curves, where P is the axial force (tension is positive), and M2 and M3 are the moments. For a given curve, these moments may have a fixed ratio, but this is not necessary. The following rules apply:

- All curves must have the same number of points.

- For each curve, the points are ordered from most negative (compressive) value of P to the most positive (tensile).
- The three values P, M2 and M3 for the first point of all curves must be identical, and the same is true for the last point of all curves
- When the M2-M3 plane is viewed from above (looking toward compression), the curves should be defined in a counter-clockwise direction
- The surface must be convex. This means that the plane tangent to the surface at any point must be wholly outside the surface. If you define a surface that is not convex, the program will automatically increase the radius of any points which are “pushed in” so that their tangent planes are outside the surface. A warning will be issued during analysis that this has been done.

You can explicitly define the interaction surface, or let the program calculate it using one of the following formulas:

- Steel, AISC-LRFD Equations H1-1a and H1-1b with phi = 1
- Steel, FEMA-356 Equation 5-4
- Concrete, ACI 318-02 with phi = 1

You may look at the hinge properties for the generated hinge to see the specific surface that was calculated by the program.

Moment-Rotation Curves: For PMM hinges you specify one or more moment/plastic-rotation curves corresponding to different values of P and moment angle θ. The moment angle is measured in the M2-M3 plane, where 0° is the positive M2 axis, and 90° is the positive M3 axis.

You may specify one or more axial loads P and one or more moment angles θ. For each pair (P, θ), the moment-rotation curve should represent the results of the following experiment:

- Apply the fixed axial load P.
- Increase the moments M2 and M3 in a fixed ratio (Cos θ, Sin θ) corresponding to the moment angle θ.
- Measure the plastic rotations Rp2 and Rp3 that occur after yield.
- Calculate the resultant moment $M = M2 \times cos\theta + M3 \times sin\theta$, and the projected plastic rotation $Rp = Rp2 \times cos\theta + Rp3 \times sin\theta$ at each measurement increment.
- Plot M vs. Rp, and supply this data to ETABS.

Note that the measured direction of plastic strain may not be the same as the direction of moment, but the projected value is taken along the direction of the moment. In addition, there may be measured axial plastic strain that is not part of the projection. However, during analysis the program will recalculate the total plastic strain based on the direction of the normal to the interaction (yield) surface.

During analysis, once the hinge yields for the first time, i.e., once the values of P, M2 and M3 first reach the interaction surface, a net moment-rotation curve is interpolated to the yield point from the given curves. This curve is used for the rest of the analysis for that hinge.

If the values of P, M2, and M3 change from the values used to interpolate the curve, the curve is adjusted to provide an energy equivalent moment-rotation curve. This means that the area under the moment-rotation curve is held fixed, so that if the resultant moment is smaller, the ductility is larger. This is consistent with the underlying stress strain curves of axial “fibers” in the cross section.

As plastic deformation occurs, the yield surface changes size according to the shape of the M-Rp curve, depending upon the amount of plastic work that is done. You have the option to specify whether the surface should change in size equally in the P, M2, and M3 directions, or only in the M2 and M3 directions. In the latter case, axial deformation behaves as if it is perfectly plastic with no hardening or collapse. Axial collapse may be more realistic in some hinges, but it is computationally difficult and may re quire nonlinear direct-integration time-history analysis if the structure is not stable enough the redistribute any dropped gravity load.

2.4.7. Parametric P-M2-M3 Hinge

The Parametric P-M2-M3 hinge uses plasticity theory to model P-M2-M3 interaction. This hinge is limited to doubly symmetric section properties and uses a simple parametric definition of the PMM surface. Hysteretic energy degradation can be specified, making it more suitable than the isotropic hinge for extensive cyclic loading. Two versions of the hinge are available, one for steel frame sections, and one for reinforced-concrete frame sections. These two types differ only in the yield surface used. The Parametric Steel P-M2-M3 hinge is intended to model steel sections while the Parametric Concrete P-M2-M3 hinge is intended to model reinforced concrete sections.

2.4.8. Fiber P-M2-M3 Hinge

The Fiber P-M2-M3 (Fiber PMM) hinge models the axial behavior of a number of representative axial “fibers” distributed across the cross section of the frame element. Each fiber has a location, a tributary area, and a stress-strain curve. The axial stresses are integrated over the section to compute the values of P, M2 and M3. Likewise, the axial deformation U1 and the rotations R2 and R3 are used to compute the axial strains in each fiber. Plane sections are assumed to remain planar.

You can define you own fiber hinge, explicitly specifying the location, area, material and its stress strain curve for each fiber, or you can let the program automatically create fiber hinges for circular and rectangular frame sections.

The Fiber PMM hinge is more “natural” than the Isotropic or Parametric PMM hinges described above, since it automatically accounts for interaction, changing moment-rotation curve, and plastic axial strain. However, it is also more computationally intensive, requiring more computer storage and execution time.

Strength loss in a fiber hinge is determined by the strength loss in the underlying stress-strain curves. Because all the fibers in a cross section do not usually fail at the same time, the overall hinges tend to exhibit more gradual strength loss than hinges with directly specified moment-rotation curves. This is especially true if reasonable hinge lengths are used. For this reason, the program does not automatically restrict the negative drop-off slopes of fiber hinges. However, we still recommend that you pay close attention to the modeling of strength loss, and modify the stress-strain curves if necessary.

Figure 2-21 shows the CSI ETABS form to define the properties (using the user-defined option) of an interacting P-M2-M3 fiber hinge (concrete type).

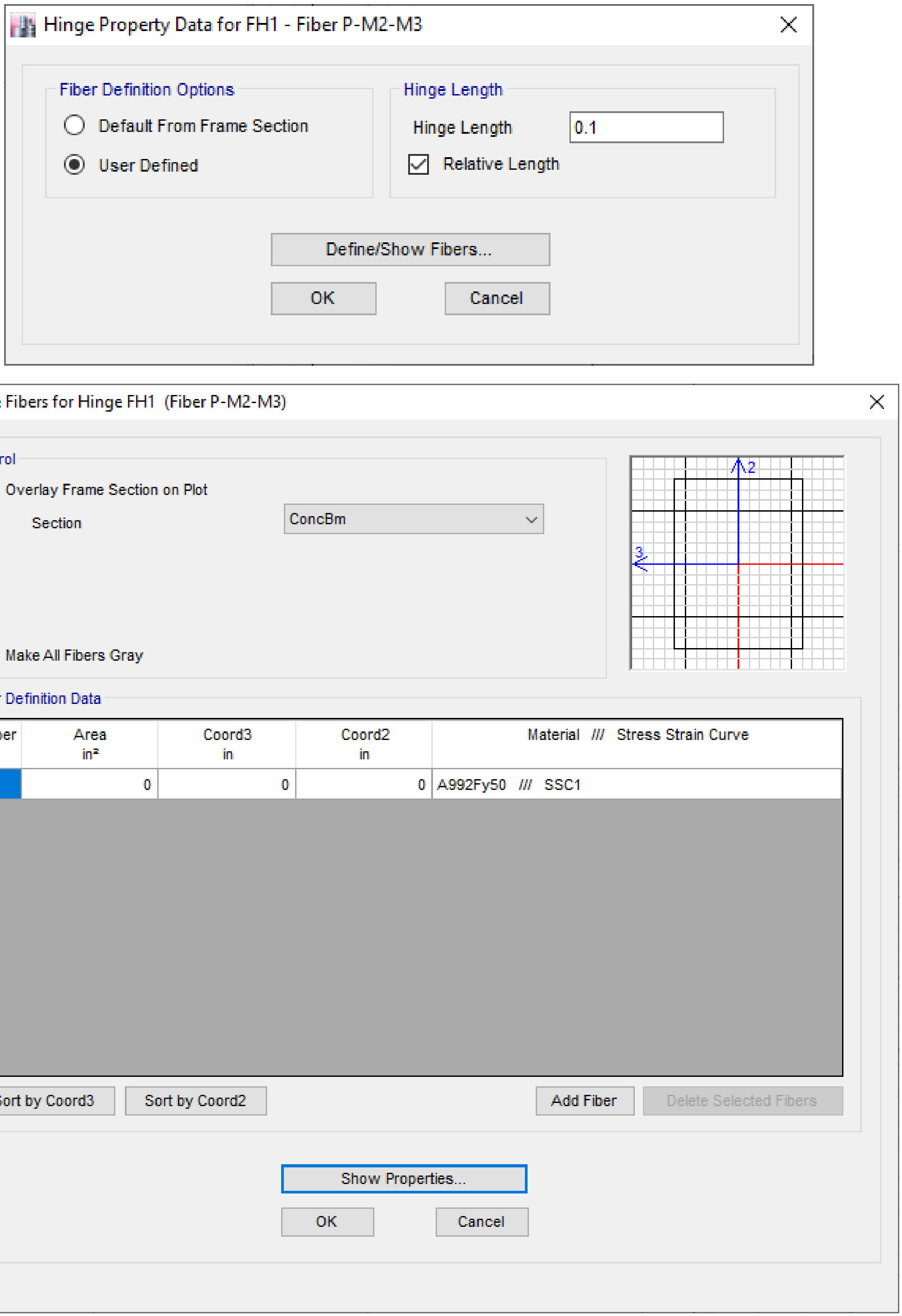


Figure 2-21: CSI ETABS form to define the properties (using the user-defined option) of an interacting P-M2-M3 fiber hinge (concrete type).

2.4.9. Hysteresis Models

The plastic force-deformation or moment-rotation curve defines the nonlinear behavior under monotonic loading. This curve, combined with the elastic behavior of the hinge length in the parent frame element, is also known as the backbone curve for the hinge.

Under load reversal or cyclic loading, the behavior will deviate from the backbone curve. *As mentioned in Section 2.3, several different hysteresis models are available to describe this behavior for different types of materials.*

Hysteresis models are applicable to the different types of hinges as follows:

- Single degree of freedom hinges: All inelastic models (kinematic, degrading, Takeda, pivot, concrete, BRB hardening and isotopic)
- Coupled P-M2-M3, P-M2, P-M3, and M2-M3 hinges: Isotropic model only
- Fiber P-M2-M3 hinges: For each material fiber, all models (elastic, kinematic, degrading, Takeda, pivot, concrete, BRB hardening and isotopic)

Hysteretic behavior may affect nonlinear static and nonlinear time-history load cases that exhibit load reversals and cyclic loading. Monotonic loading is not affected. Note, however, that even static pushover load cases can produce load reversal in some hinges caused by strength loss in other hinges.

2.5. Inelastic Material Properties (Applicable to Fiber Hinges) in ETABS

In CSI ETABS, the Material properties may be defined as *isotropic, orthotropic or anisotropic*. How the properties are actually utilized depends on the element type. Each Material that you define may be used by more than one element or element type. For each element type, the Materials are referenced in directly through the Section properties appropriate for that element type. The following are basic types of properties which can be defined for each material in CSI ETABS.

Basic Elastic Analysis Properties: These are the basic material properties used in the linear elastic analysis of structures. These include mass density, elastic modulus, Poisson's ratio, and coefficient of thermal expansion.

Design-related Properties: These are the basic material properties used in the automated code-based design of structures. You may specify a "design-type" for each Material that indicates how it is to be treated for design by the CSI ETABS graphical user interface. The available design types are:

- Steel: Frame elements made of this material will be designed according to steel design codes.
- Concrete: Frame elements made of this material will be designed according to concrete design codes.
- Aluminum: Frame elements made of this material will be designed according to aluminum design codes.
- Cold-formed: Frame elements made of this material will be designed according to cold-formed steel design codes.
- None: Frame elements made of this material will not be designed.

When you choose a design type, additional material properties may be specified that are used only for design; they do not affect the analysis. These include e.g., concrete's compressive strength, steel's tensile yield strength and ultimate strength etc.

Temperature-dependent properties: All elastic material properties may be temperature dependent. Properties are given at a series of specified temperatures. Properties at other temperatures are obtained by linear interpolation. For a given execution of the program, the properties used by an element are assumed to be constant regardless of any temperature changes experienced by the structure. Each element may be assigned a material temperature that determines the material properties used for the analysis.

Time-dependent properties: These include creep, shrinkage, and age-dependent elasticity. These properties can be activated during a staged-construction analysis, and form the basis for subsequent analyses. For concrete-type materials, you may specify:

- Aging parameters that determine the change in modulus of elasticity with age.
- Shrinkage parameters that determine the decrease in direct strains with time.
- Creep parameters that determine the change in strain with time under the action of stress.

Material Damping Properties: The material damping can be specified for use in dynamic analyses. Different types of damping are available for different types of Load Cases. Material damping is a property of the material and affects all Load Cases of a given type in the same way. You may specify additional damping in each Load Case.

Nonlinear Material Properties: Nonlinear stress-strain curves may be defined for use with fiber hinges in frame elements or nonlinear layers in shell elements.

Nonlinear material behavior is available in certain elements using a directional material model, in which uncoupled stress-strain behavior is modeled for one or more stress-strain components. This is a simple and practical engineering model suitable for many applications such as beams and columns, shear walls, bridge decks, tunnels, retaining walls, and others. You should carefully examine the applicability of this model before using it in a general continuum model where the governing stresses change direction substantially from place to place.

Nonlinear material behavior is currently not temperature-dependent. The behavior specified at the initial (most negative) temperature is used for all material temperatures.

Tension and Compression: For each material you may specify an axial stress-strain curve that is used to represent the direct (tension-compression) stress-strain behavior of the material along any material axis. The nonlinear stress-strain behavior is the same in each direction, even for Orthotropic, and Anisotropic materials. Tension is always positive, regardless of the type of material (steel, concrete, etc.). The tensile and compressive sides of the stress-strain behavior may be different from each other.

Shear: A shear stress-strain curve is computed internally from the direct stress-strain curve. The assumption is made that shearing behavior can be computed from tensile and compressive behavior acting at 45° to the material axes using Mohr's circle in the plane.

Cyclic Behavior: Several hysteresis models are available to define the nonlinear cyclic stress-strain behavior when load is reversed or cycled. For the most part, these models differ in the amount of energy they dissipate in a given cycle of deformation, and how the energy dissipation behavior changes with an increasing amount of deformation.

Nonlinear stress-strain curves are currently used in the following two applications.

a) ***Fiber Hinges:*** As mentioned earlier in Chapter 1, fiber modeling is used to define the coupled axial force and bi-axial bending behavior at locations along the length of a frame element. In ETABS, the term "fiber hinges" is used to define and implement this approach. These hinges can be defined manually, or created automatically for certain types of frame sections, including Section-Designer sections. For each fiber in the cross section at a fiber hinge, the material direct nonlinear stress-strain curve is used to define its behavior. Summing up the behavior of all the fibers at a cross section and multiplying by the hinge length gives the axial force-deformation and biaxial moment-rotation relationships.

b) ***Layered Shell Element:*** The Shell element with the layered section property may consider linear, nonlinear, or mixed material behavior. For each layer, you select a material, a material angle, and whether each of the in-plane stress-strain relationships are linear, nonlinear, or inactive (zero stress).

Modified Darwin-Pecknold Concrete Model: A two-dimensional nonlinear concrete material model is available for use in the layered shell. This model is based on the Darwin-Pecknold model, with consideration of Vecchio-Collins behavior. This model represents the concrete compression, cracking, and shear behavior under both monotonic and cyclic loading, and considers the three direct stress-strain components ($\sigma_{11} - \epsilon_{11}$, $\sigma_{22} - \epsilon_{22}$, and $\sigma_{33} - \epsilon_{33}$). A state of plane stress is assumed. The direction of cracking can change during the loading history, and the shear strength is affected by the tension strain in the material. The axial stress-strain curve specified for the material is simplified to account for initial stiffness, yielding, ultimate plateau, and strength loss due to crushing. Zero tensile strength is assumed. Hysteresis is governed by the concrete hysteresis model described in the previous topic, with the energy dissipation factor $f = 0$. The layered shell allows this material to be used for membrane and/or flexural behavior and to be combined with steel reinforcement placed in arbitrary directions and locations. Transverse (out-of plane) shear is assumed to be elastic and isotropic.

The Figure 2-22 shows the CSI ETABS form for defining the inelastic materials (for use in fiber hinges and layered shell elements).

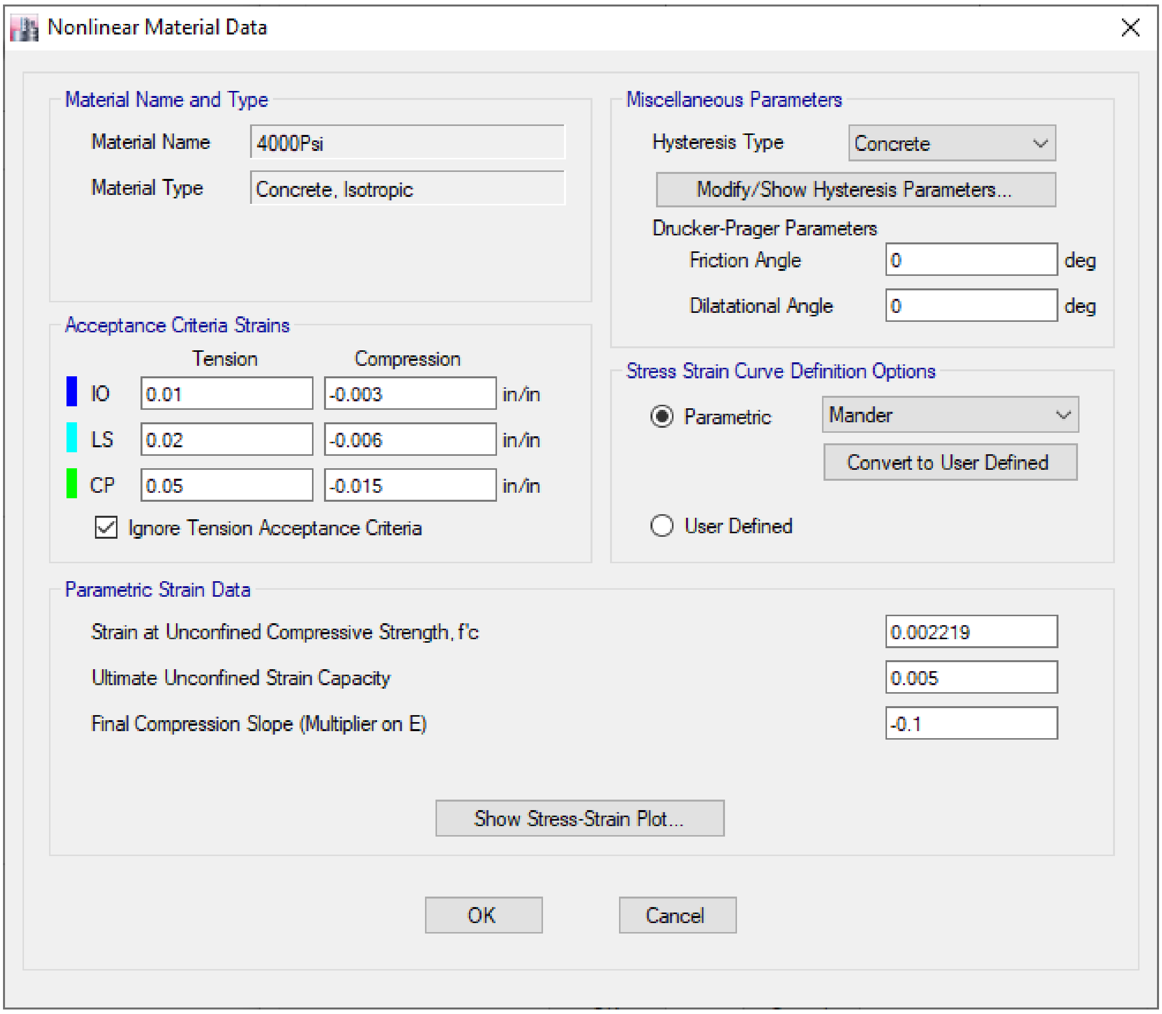


Figure 2-22: CSI ETABS form to define the inelastic materials (for use in fiber hinges and layered shell elements).

2.6. Automatic, User-Defined, and Generated Hinge Properties

There are three types of hinge properties in CSI ETABS:

- Automatic hinge properties
- User-defined hinge properties
- Generated hinge properties

Only automatic hinge properties and user-defined hinge properties can be assigned to frame elements. When automatic or user-defined hinge properties are assigned to a frame element, the program automatically creates a generated hinge property for each and every hinge.

The built-in automatic hinge properties for steel and concrete members are based on tables of ASCE 41-13 and ASCE 41-17. After assigning automatic hinge properties to a frame element, the program generates a hinge property that includes specific information from the frame section geometry, the material, and the length of the element. You should review the generated properties for their applicability to your specific project.

User-defined hinge properties can either be based on a hinge property generated from automatic property, or they can be fully user-defined.

A generated property can be converted to user-defined, and then modified and re-assigned to one or more frame elements. This way you can let the program do much of the work for you using automatic properties, but you can still customize the hinges to suit your needs. However, once you convert a generated hinge to user-defined, it will no longer change if you modify the element, its section or material.

It is the generated hinge properties that are actually used in the analysis. They can be viewed, but they cannot be modified. Generated hinge properties have an automatic naming convention of LabelH#, where Label is the frame element label, H stands for hinge, and # represents the hinge number. The program starts with hinge number 1 and increments the hinge number by one for each consecutive hinge applied to the frame element. For example if a frame element label is F23, the generated hinge property name for the second hinge assigned to the frame element is F23H2.

The main reason for the differentiation between defined properties (in this context, defined means both automatic and user-defined) and generated properties is that typically the hinge properties are section dependent. Thus it would be necessary to define a different set of hinge properties for each different frame section type in the model. This could potentially mean that you would need to define a very large number of hinge properties. To simplify this process, the concept of automatic properties is used in SAP2000. When automatic properties are used, the program combines its built-in default criteria with the defined section properties for each element to generate the final hinge properties. The net effect of this is that you do significantly less work defining the hinge properties because you don't have to define each and every hinge.

2.7. Automatic Hinge Properties

Automatic hinge properties are based upon a simplified set of assumptions that may not be appropriate for all structures. You may want to use automatic properties as a starting point, and then convert the corresponding generated hinges to user-defined and explicitly overwrite calculated values as needed.

Automatic properties require that the program have detailed knowledge of the Frame Section property used by the element that contains the hinge. For this reason, only the following types of automatic hinges are available:

Concrete Beams in Flexure: M2 or M3 hinges can be generated using ASCE 41-17 Table 10-7 for the following shapes:

- Rectangle
- Tee
- Angle
- Section Designer

Concrete Columns in Flexure: M2, M3, M2-M3, P-M2, P-M3, or P-M2-M3 hinges can be generated using ASCE 41-17 Table 10-8 for the following shapes:

- Rectangle
- Circle
- Section Designer

or using Caltrans specifications, for the following shapes:

- Section Designer only

Steel Beams in Flexure: M2 or M3 hinges can be generated using ASCE 41, for the I/Wide-flange shape only.

Steel Columns in Flexure: M2, M3, M2-M3, P-M2, P-M3, or P-M2-M3 hinges can be generated using ASCE 41, for the I/Wide-flange and box shapes only.

Steel Braces in Tension/Compression: P (axial) hinges can be generated using ASCE 41, for the following shapes:

- I/Wide-flange
- Box
- Pipe
- Double channel
- Double angle

Fiber Hinge: P-M2-M3 hinges can be generated for steel or reinforced concrete members using the underlying stress-strain behavior of the material for the rectangular and circular shapes:

Additional Considerations: You must make sure that all required design information is available to the Frame section as follows:

- For concrete Sections, the reinforcing steel must be explicitly defined, or else the section must have already been designed by the program before nonlinear analysis is performed.
- For steel Sections, Auto-select Sections can only be used if they have already been designed so that a specific section has been chosen before nonlinear analysis is performed.

2.8. Analysis Modeling

Hinges are assigned to a Frame or Shell (shear wall) element to represent the nonlinear behavior of their parent element. When the analysis model is created, there are two ways the hinge can be represented:

a) Hinge embedded in the element.
b) Hinge as a separate link element.

The latter method is currently only available in the ETABS Ultimate level, and enables hinge behavior to be considered in nonlinear model time-history (FNA) load cases. As a rule, FNA analysis runs significantly faster than nonlinear direct-integration time-history analysis. Nonlinear static analysis and nonlinear direct-integration time history analysis are available for both types of analysis modeling.

When the hinge is modeled as a link element, the parent Frame element is divided at the hinge location into separate sub-elements, and a zero-length link element is created that contains the hinge property and connects the frame sub-elements. A very small amount of axial mass and rotational inertia are added at the two connecting joints to improve FNA iteration. A similar internal modeling is employed for shear wall elements when the hinge is modeled as a link element.

A second, independent modeling option is available to assign automatic subdivision of Frame elements at hinge locations. Using this assignment, you specify a relative length that is used when creating the analysis model of the hinges for the selected elements. The effect of this depends upon how the hinge is modeled:

- *For the hinge embedded in the element:* The Frame object is subdivided into separate frame elements, with one element containing the hinge that is equal in length to that specified in the assignment.

  This has the advantage of introducing more degrees of freedom into the model that may improve convergence when multiple hinges are failing at the same time, with a possible increase in computation time. In addition, steeper drop-offs are permitted when the hinge curve exhibits strength loss because the element containing the hinge is shorter, and hence stiffer.

  On the other hand, not subdividing the frame element leads to a smaller analysis model, typically requiring less computation time and storage. In addition, stiffness proportional damping for nonlinear direct-integration time-history analysis is better modeled in longer elements.

- *For the hinge as a separate link element:* The subdivision into two frame elements and a zero-length link is not changed. However, the elastic flexibility of the link is changed to be equal to the length of the frame element specified in the assignment, and the corresponding length of the adjacent frame sub elements are made rigid.

  This has the advantage of improving stiffness-proportional damping in nonlinear direct-integration time history analysis, and can be recommended for this reason. On the other hand, this is not necessary for FNA analysis.

The default relative length for automatic subdivision is 0.02. Recommended values typically range from 0.02 to 0.25.

2.9. Computational Considerations

The most important advice is to only add hinges to the model where nonlinear behavior is expected to have a significant effect on the analysis and design. Adding extra hinges increases the time and effort it takes to create the model, to run the analyses, and to interpret the results.

Start with the simplest model possible so that you can make many analysis runs quickly. This helps to better understand the behavior of your structure early in the design process and to correct modeling errors. Add hinges and complexity gradually as you determine where nonlinearity is expected and/or desired.

Adding hinges everywhere to find the nonlinearity is tempting, but this approach usually wastes much more time than incrementally growing the model.

Most models with hinges benefit from using event-to-event stepping for nonlinear static and nonlinear direct-integration time-history load cases. This is particularly true for the parametric P-M2-M3 hinge. However, it may be necessary to turn off event-to-event stepping if the model has a very large number of hinges, or if there is a significant amount of other types of nonlinearity in the structure. This is best determined by running analyses both with and without events to see which is most efficient.

Most nonlinear time-history analysis benefits from the presence of mass at the nonlinear degrees-of freedom. Inertia tends to stabilize iteration when the nonlinear behavior is changing rapidly. This is particularly true for FNA analysis. For ETABS, it is usually best to define the mass source to include vertical mass and to not lump the mass at the story levels for models that have hinges.

For FNA analysis, it is usually most efficient to damp out the very high modes. Some of the Ritz modes needed for FNA analysis can be expected to be of high frequency. An example of how to do this would be to define the load-case damping to be of type "Interpolated by Frequency". Then specify your desired structure damping ratio (say 0.025) for frequencies up to 999 Hz, and a damping ratio of 0.99 for frequencies above 1000 Hz. You can experiment with this cutoff value to see the effect on runtime and results.

2.10. Analysis Results

For each output step in a nonlinear static or nonlinear direct-integration time-history Load Case, you may request analysis results for the hinges. These results include:

- The forces and/or moments carried by the hinge. Degrees of freedom not defined for the hinge will report zero values, even though non-zero values are carried rigidly through the hinge.
- The plastic displacements and/or rotations.
- The most extreme state experienced by the hinge in any degree of freedom.
  This state does not indicate whether it occurred for positive or negative deformation:
  a) A to B
  b) B to C
  c) C to D
  d) D to E
  e) > E
- The most extreme performance status experienced by the hinge in any degree of freedom. This status does not indicate whether it occurred for positive or negative deformation:
  a) A to B
  b) B to IO
  c) IO to LS
  d) LS to CP
  e) > CP

When you display the deflected shape in the graphical user interface for a nonlinear static or nonlinear direct integration time-history Load Case, the hinges are plotted as colored dots indicating their most extreme state or status:

a) B to IO
b) IO to LS

c) LS to CP

d) CP to C

e) C to D

f) D to E

g) > E

The colors used for the different states are indicated on the plot. Hinges that have not experienced any plastic deformation (A to B) are not shown.

Chapter 3

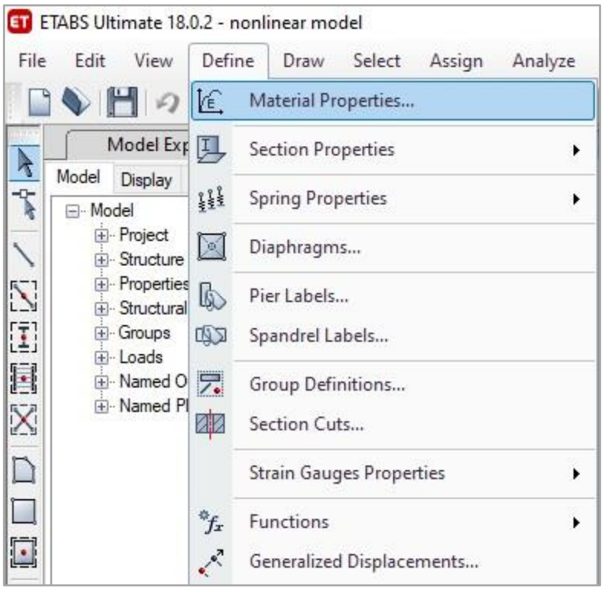
Nonlinear Modeling of Columns using Fiber Modeling Approach

This section describes a step-by-step procedure to model the RC columns in a building using the P-M2-M3 fiber hinges in CSI ETABS.

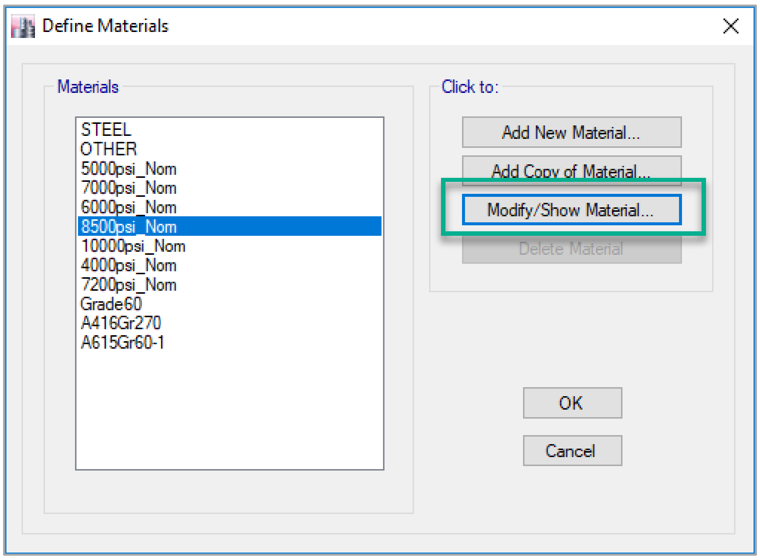
3.1. Definition of Nonlinear Stress-strain Curves (for Material Fibers)

Before defining the P-M2-M3 fiber hinges for column sections, the first step is to define the nonlinear stress-strain curves for all materials. These curves will be assigned to individual material fibers while defining the P-M2-M3 fiber hinges.

→ Click Define > Material Properties.

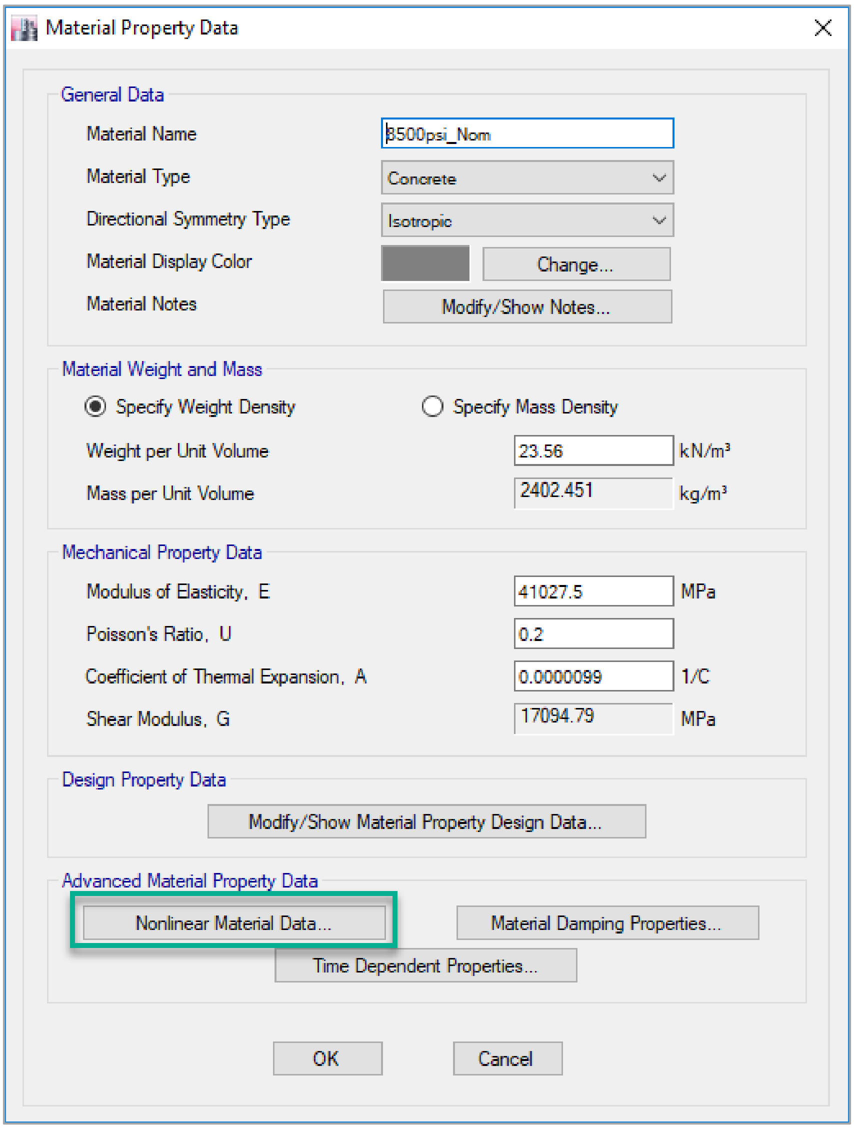


→ The following form will open.



→ Click "Add New Material" for defining the properties of a material. Alternatively, click on a default (or previously defined) material (e.g., concrete or steel) and then click "Modify/Show Material".

→ The following form will open.

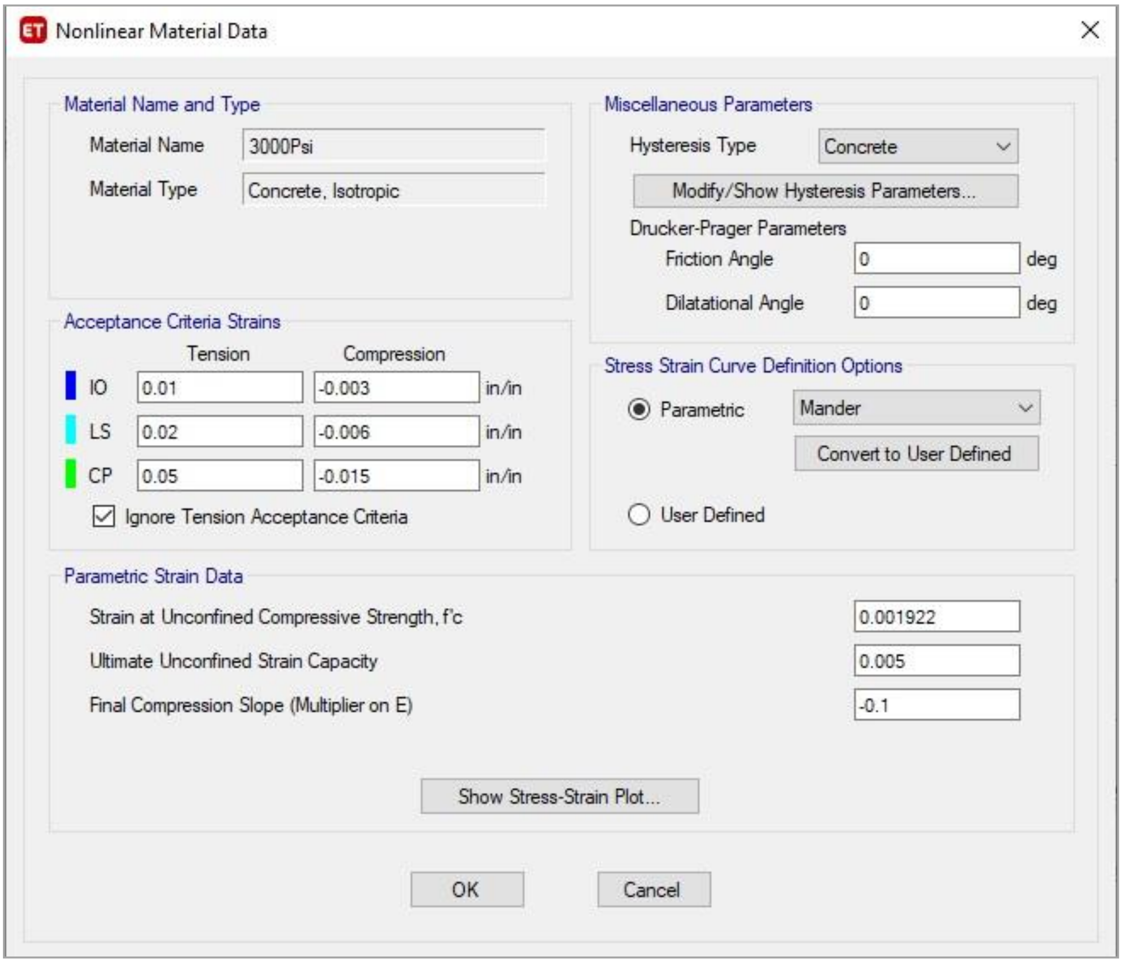


→ The basic material properties can be defined in this form. These include the material density, elastic modulus, Poisson's ratio, coefficient of thermal expansion and shear modulus. The material strength parameters (e.g., f_c' for concrete and f_y for steel) can be provided by clicking "Modify/Show Material Property Design Data".

→ With the objective of accurately simulating structural performance, nonlinear models should be based on the expected properties of materials and components, rather than nominal or minimum specified properties that are otherwise used in design. These properties will generally include the stiffness, strength and deformation characteristics of the components. The term "expected" refers to properties that are defined based on median values from a large population of materials and components that are representative of what occurs in the structure. Use of expected structural properties is important for providing an accurate and unbiased measure of the expected response of the overall system. Equally important is the use of expected values throughout the model to accurately characterize the relative force and deformation demands between components of indeterminate structural systems. The goal is to avoid any systematic bias that could result from the use of nominal instead of expected properties for some components, and not others, in a structure.

The nominal values of f_c' and f_y for concrete and steel materials should be multiplied with amplification factors (recommended by different guidelines) to convert them to "expected" strengths. In the LATBDSC guidelines, an expected strength factors of 1.3 and 1.17 are recommended for concrete and steel, respectively.

→ For defining the stress-strain curve of the material, click on "Nonlinear Material Data". In case of concrete type material, the following form will open.



→ The following three inputs are required to be given in this form.

i) The backbone of the nonlinear force-deformation behavior. In this case, it is the stress-strain curve of the material.

ii) The hysteretic behavior (or rule) for the cyclic force-deformation behavior. In this case, it is the cyclic stress-strain behavior of the material.

iii) The acceptance criteria (i.e., the values of IO, LS and CP) in the units of deformation parameter. In this case, the deformation parameter is the material strain.

→ For defining the backbone of the stress-strain curve, two options ("Parametric" and "User Defined") can be used.

→ The parametric option can directly generate the backbone coordinates using:

i) The "Simple Parabolic" concrete stress-strain model, or
ii) The well-known "Mander" concrete stress-strain model.

The controlling parameters for generating the backbone coordinates using the parametric option can be provided under the "Parametric Strain Data".

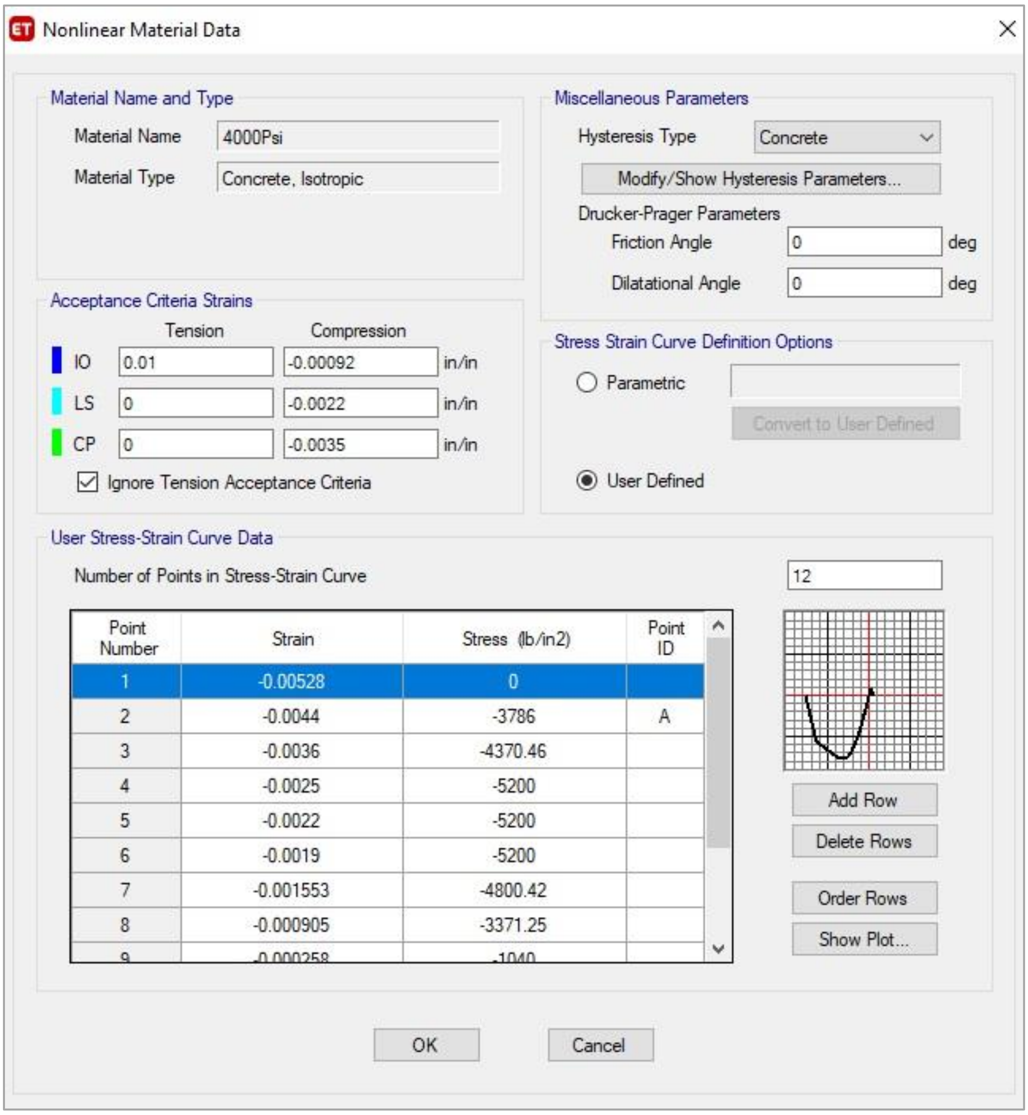
→ The "User-defined" option can be used to point-by-point to enter the coordinates of backbone. The curve is generated automatically.

→ As an example, let's define the stress-strain curve of a concrete with nominal $f_c{}'$ = 4000 psi. The expected compressive strength will be 1.3 × $f_c{}'$ = 5200 Psi. The following default values of "Parametric Strain Data" is used.

Strain at unconfined compressive strength $f_c{}'$ = 0.00192
Ultimate unconfined strain capacity = 0.005
Final compression slope (multiplier on E) = -0.1

With "Mander" option selected, the software will use the given $f_c{}'$ (should be 5200 Psi), initial elastic modulus (given in the basic material properties) and the given "Parametric Strain Data" to determine the complete stress-strain curve using the Mander's (1988) model.

→ The parametrically generated stress-strain curve can be converted to an editable table of coordinates by clicking on "Convert to User Defined". The auto-calculated stress and strain values can be modified conveniently using this option to finally achieve the desired backbone for all materials. In this table, the rows can be added or deleted as desired.

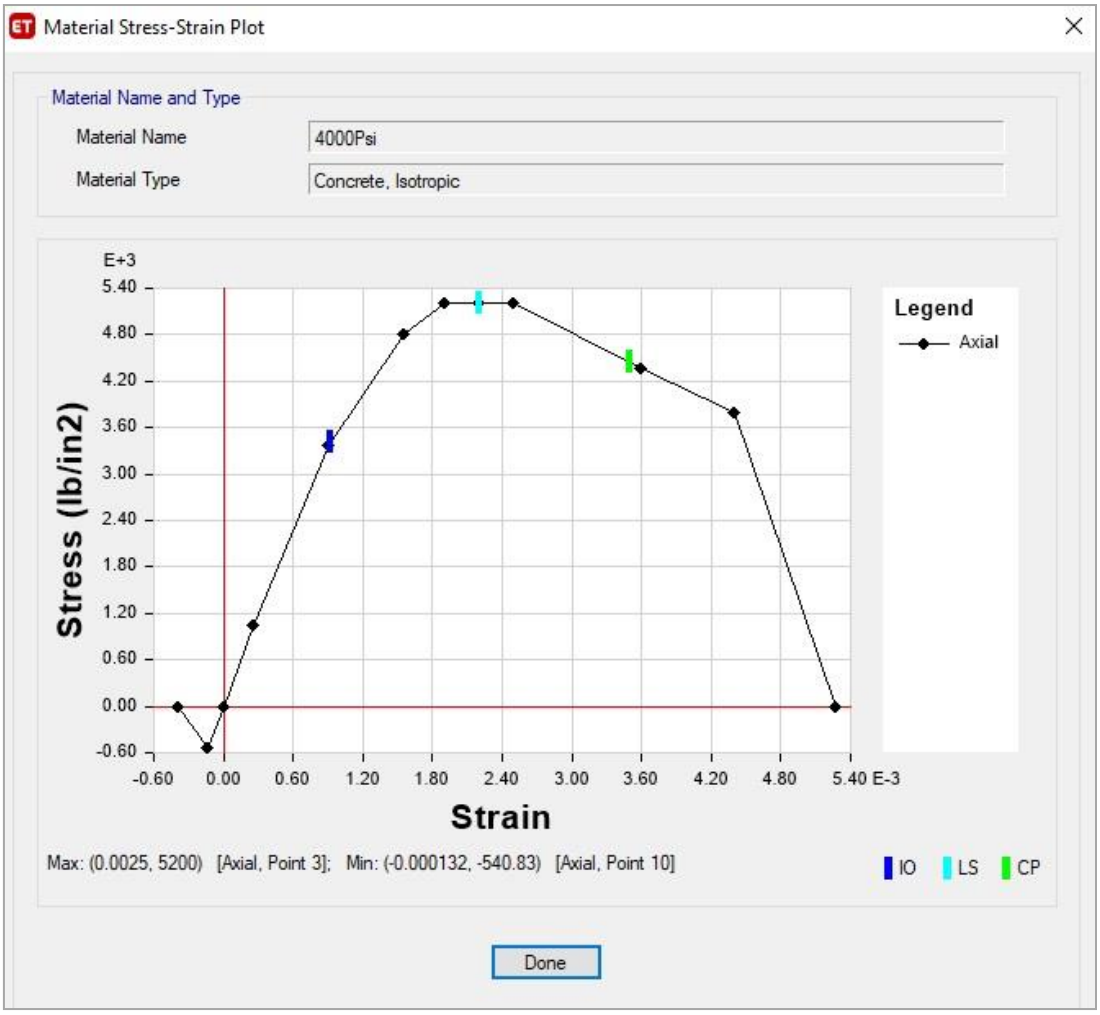


→ An MS Excel sheet is developed by authors to automate the unconfined Mander's model and can be used for guidance. It can be downloaded from the following link.
http://structurespro.info/wp-content/uploads/2021/03/NL-Material-Model.xlsx

→ Set the IO (immediate occupancy), LS (life safety) and CP (collapse prevention) in acceptance criteria strains based on experimental data.

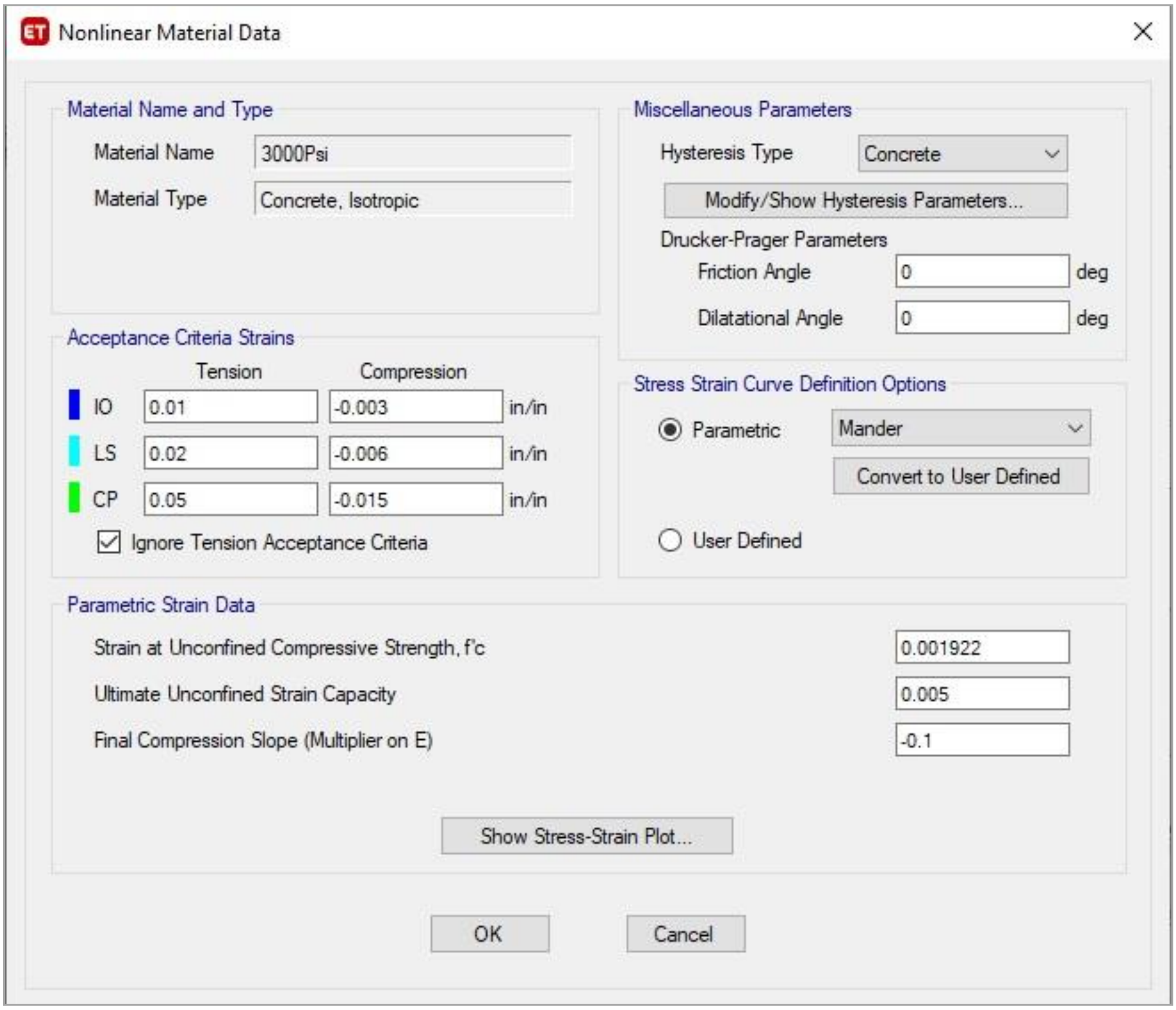
→ To see the stress strain curve of inelastic material, click on the "Show Plot" option.

→ The following window will open. Here in the stress strain backbone curve, compression is shown on positive side and tension is on negative side with acceptance criteria marked on it. In this example, the IO (immediate occupancy) point is set at the onset of first indication of inelasticity (stiffness softening) while the LS (life safety) point is set at the maximum stress level. The CP (collapse prevention) point is set at the onset of significant strength degradation.

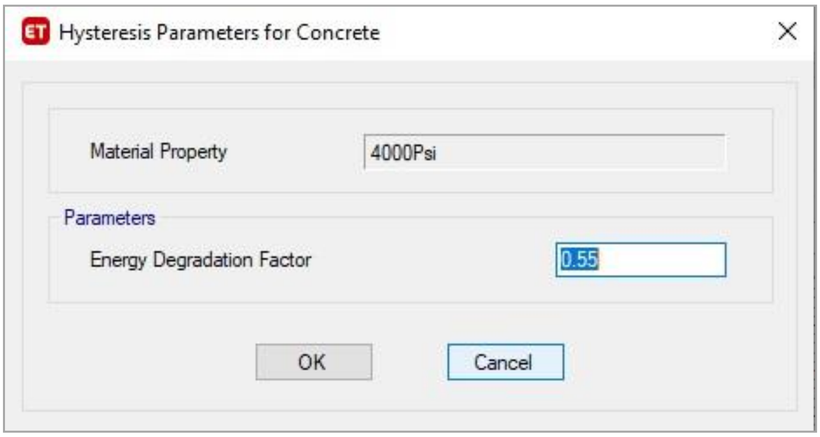


→ To select the cyclic (hysteresis) behavior of materials, return to the nonlinear material data window (as shown below) and select the suitable hysteresis model.

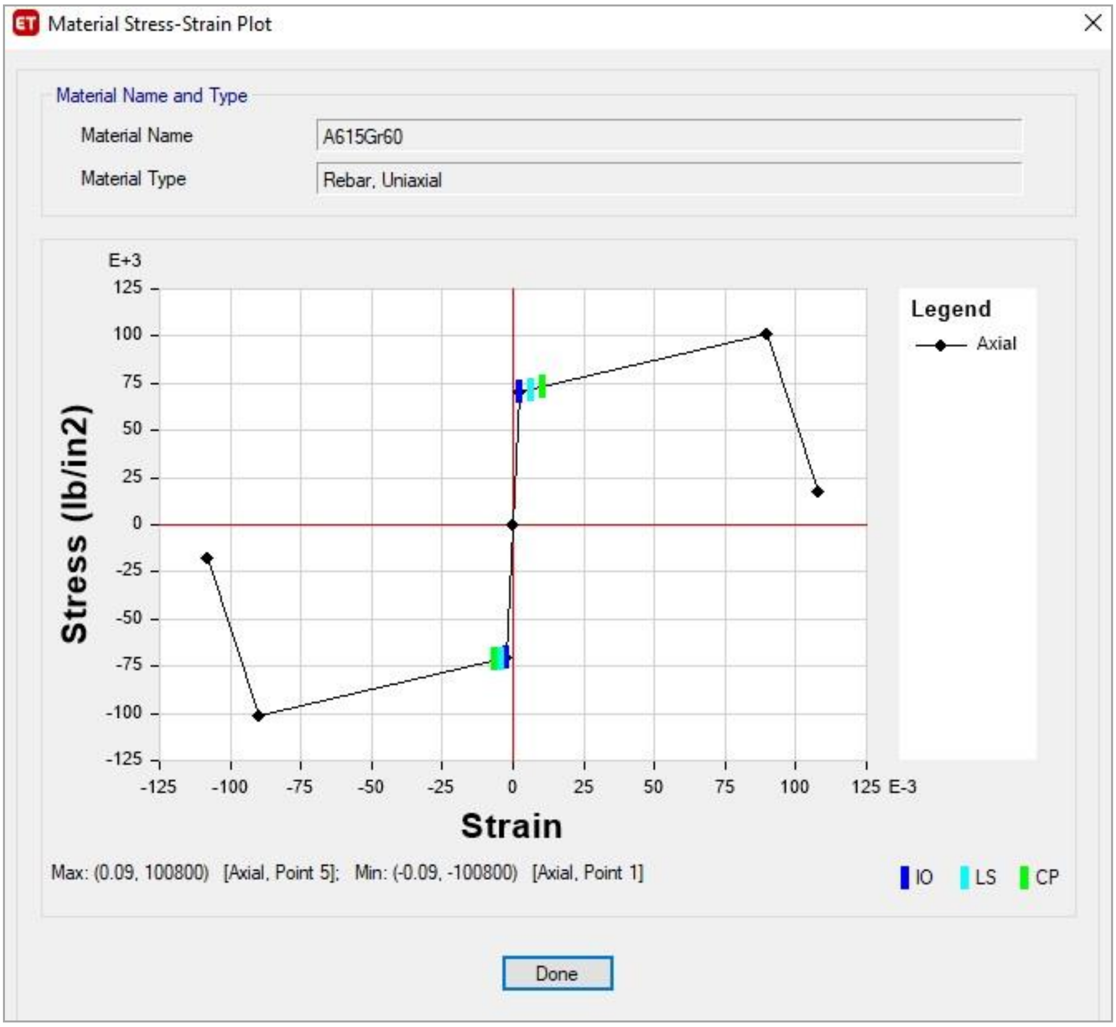
→ In this example, the “Concrete” option is selected as a hysteresis type under the “Miscellaneous Parameters” title. It is a suitable choice for the hysteresis behavior of plain concrete. The Drucker-Prager parameters including friction angle & dilatational angle are usually zero degrees for the concrete.



→ Click on the "Modify/Show Hysteresis Parameters". A new window will appear. Provide the energy degradation factor for the concrete under cyclic loading. By default, its value is zero. However, it can be modified according to the required level of degradation. This number should be based on concrete test results and may vary from 0.4 to 0.7. In this example, a factor of 0.55 is selected. A detailed description about the concrete hysteresis behavior and energy degradation factor can be seen in Section 2.3.



→ The nonlinear stress-strain curve of inelastic steel materials can also be defined in the same way. The bilinear model with or without strain hardening can be used for this purpose. In this example, the Park's model (with strain hardening) is used and shown is below figure. The Kinematic hysteresis model is used for steel. This model does not require any further input parameters.



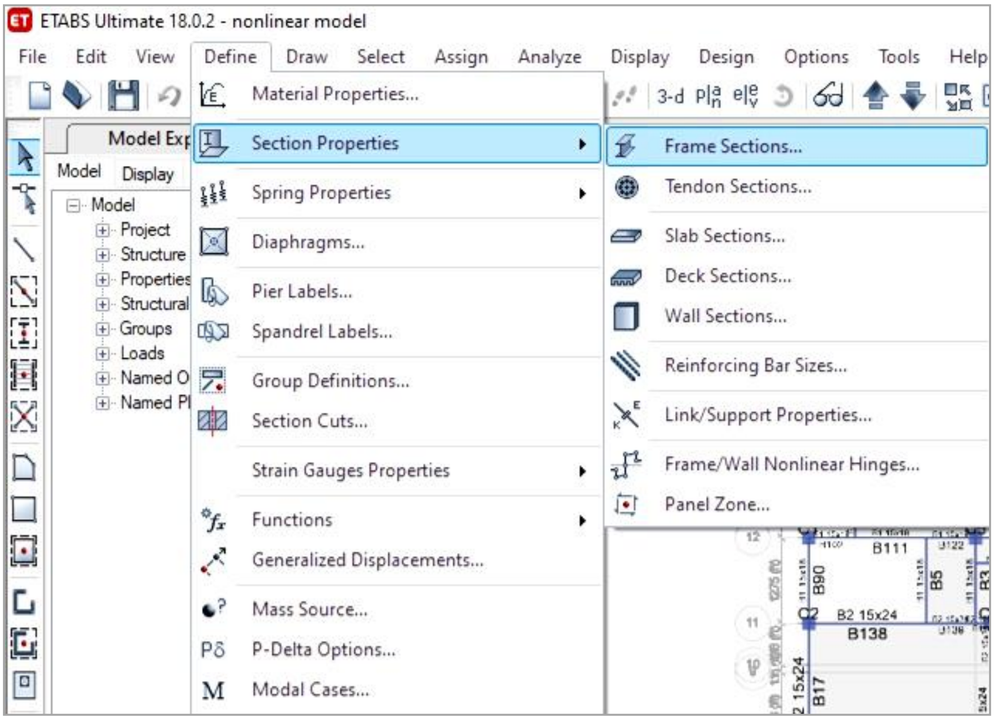
Next, the process of defining the P-M2-M3 fiber hinges for columns is explained. The hinges can either be defined manually or using automatic features of the program. For manual definition of fiber hinges, each cross-section is selected and manually divided into several concrete and steel fibers using the form shown in Figure 2-21. However, in the next Section, the automatic definitions of fiber hinges will be discussed for RC columns. Later in Chapter 5, the manual definition of fiber hinges will also be explained in the context of shear walls.

3.2. Automated Definition of P-M2-M3 Fiber Hinges

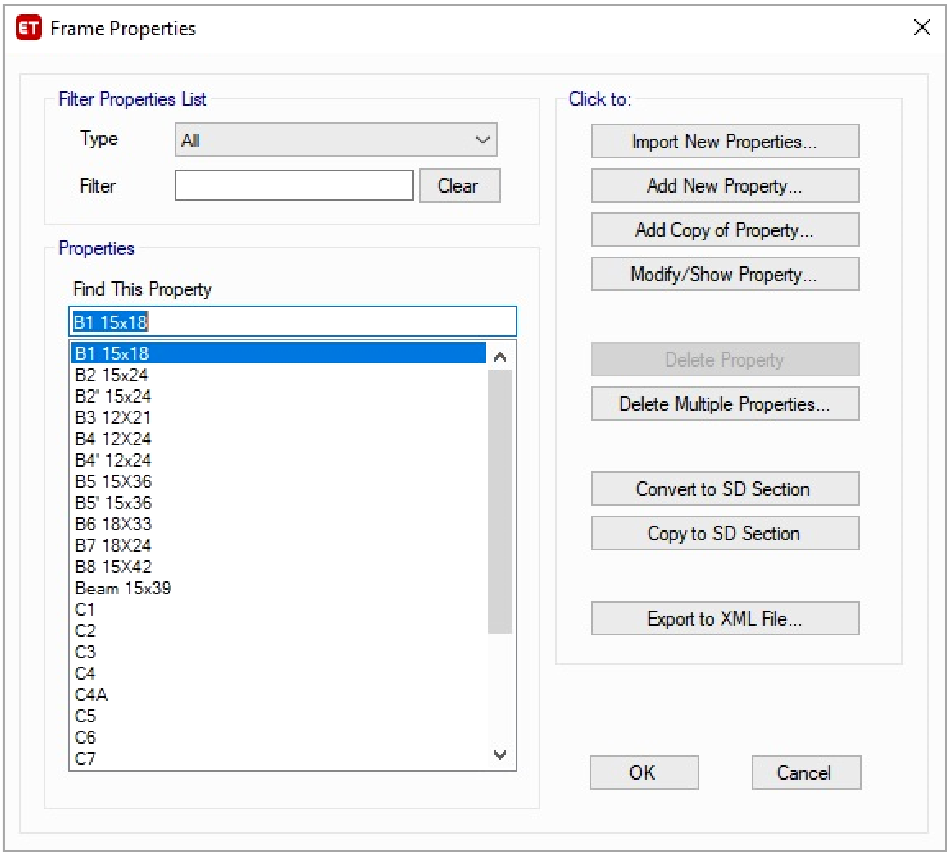
3.2.1. Step 1: Defining the Column Reinforcements

For the automatic definition of P-M2-M3 fiber hinges, first the actual reinforcement should be defined for each column cross-section. *Generally, for linear elastic modeling (for initial analysis and design), the actual column* *reinforcement is either not known or not defined in ETABS. However, this becomes a necessary step if one intends to use the automatic definition of fiber hinges.*

→ To add the design reinforcement in Columns, Click on Define > Section Properties > Frame Sections.

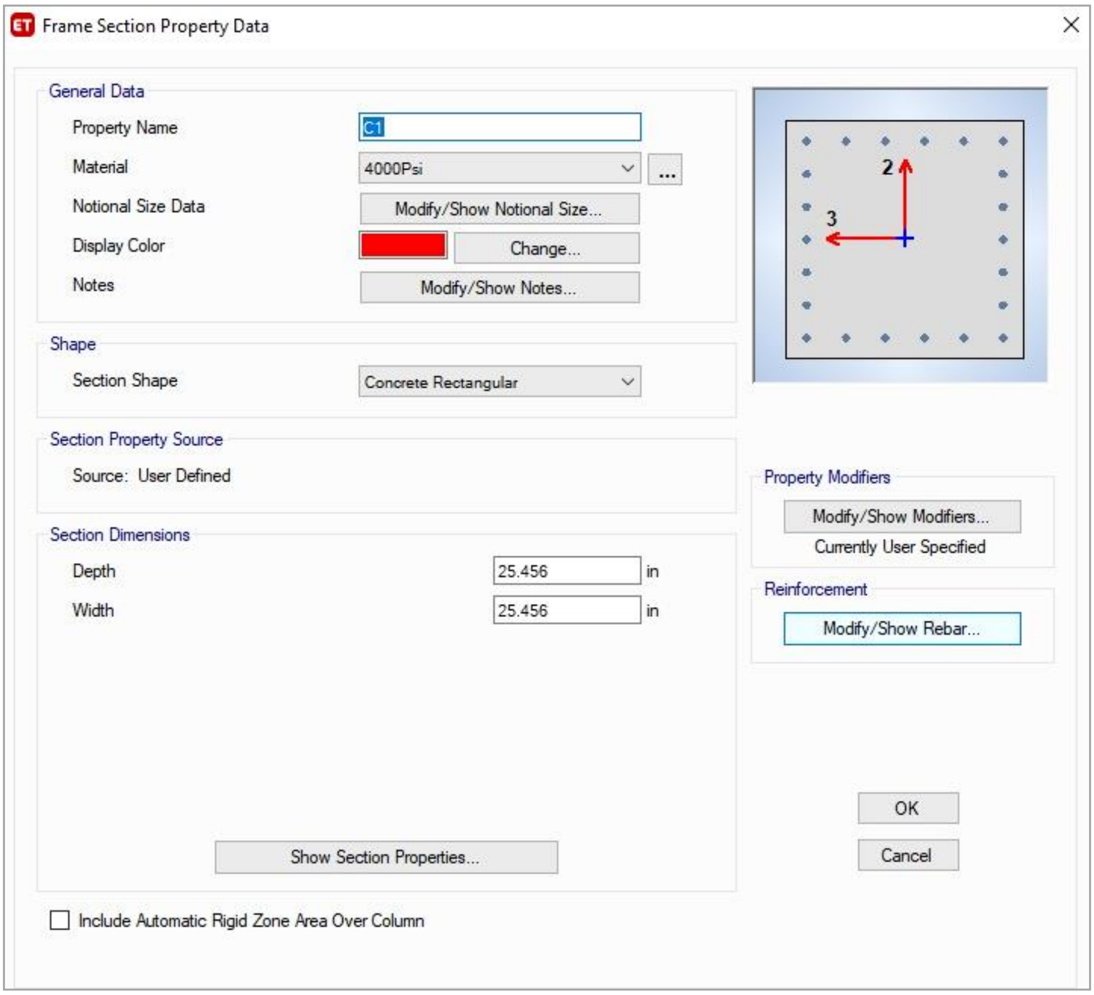


→ The following window will appear.

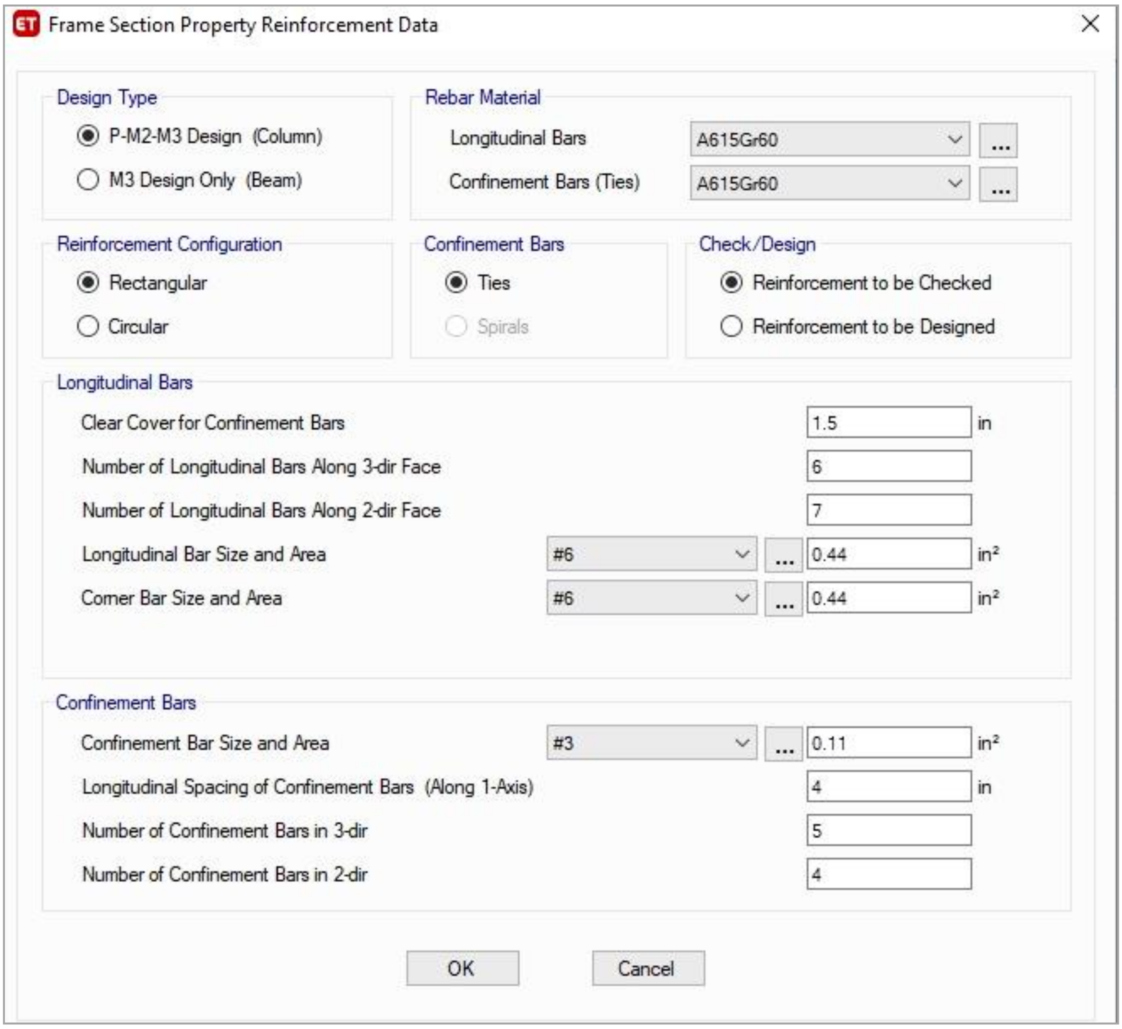


→ In the properties drop down menu, select that frame section property whose reinforcement is to be entered.

→ Click on "Modify/Show Property" which will lead to the following window.



→ In the "Frame Section Property Data" window, click on the "Modify/Show Rebar". The following window will open.



→ In the "Frame Section Property Reinforcement Data" window:

- Select the P-M2-M3 design (Columns) for column sections under the "Design Type".
- Select the grade of longitudinal and confinement bars under the "Rebar Material".
- Select the reinforcement configuration (rectangular or circular) under the "Reinforcement Configuration".
- Select the type of confinement bars i.e., "Ties" or "Spirals" under the "Confinement Bars".
- Since the initial design of frame section is already carried out and currently you are constructing the nonlinear model for performance-based assessment, select the "Reinforcement to be Checked" option under the "Check/Design". The inelastic behavior of this current design will be simulated as a result of intended nonlinear analysis.

→ Under the "Longitudinal Bars", the number of bars in 3-direction and 2-direction faces should be added. Clear cover, longitudinal bar sizes and areas (typical and corner) are provided in this window.

→ Under the "Confinement Bars", no of bars in 3-dir and 2-dir Face should be added depending upon the section reinforcement. Confinement bar size and area & their longitudinal spacing can also be provided in this window.

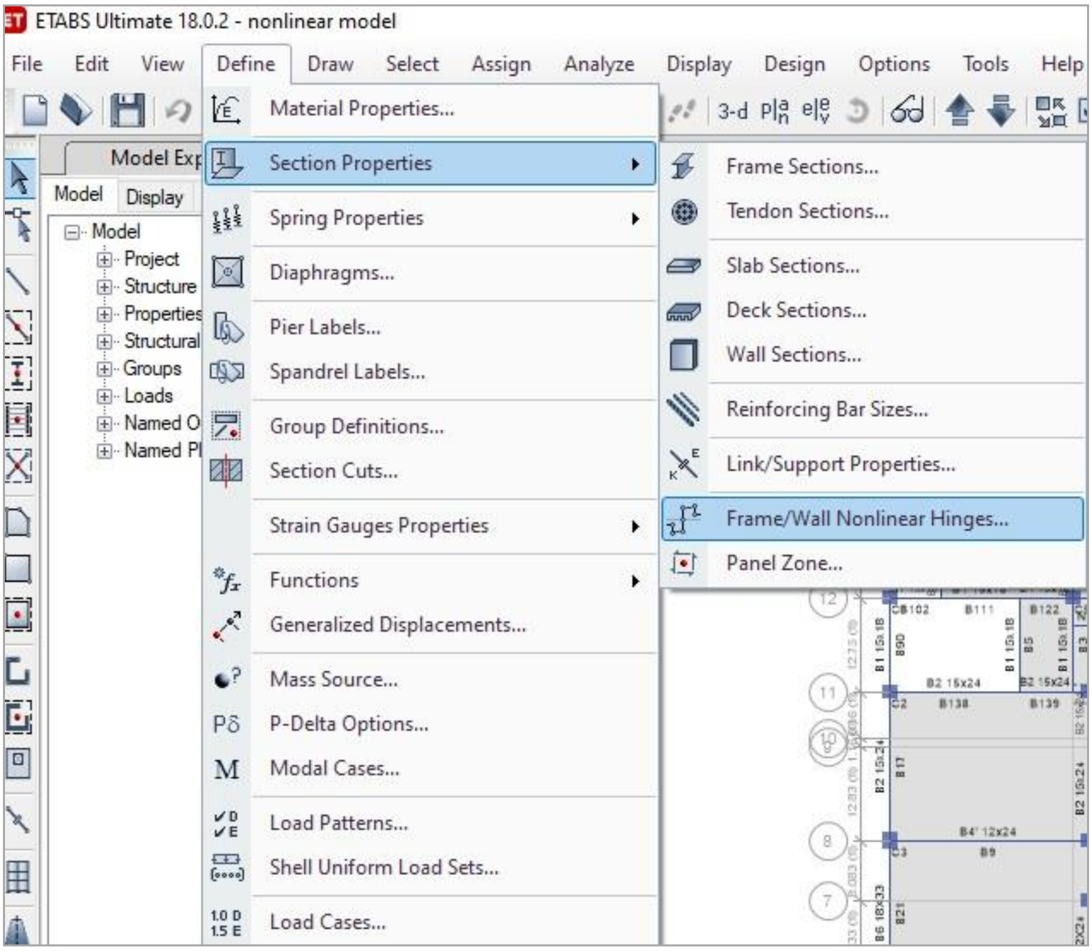
→ Click “OK”. The reinforcement for this column section is defined.

→ The same process can be repeated for all columns (with different material properties, cross-sectional shapes, sizes and amount of reinforcement).

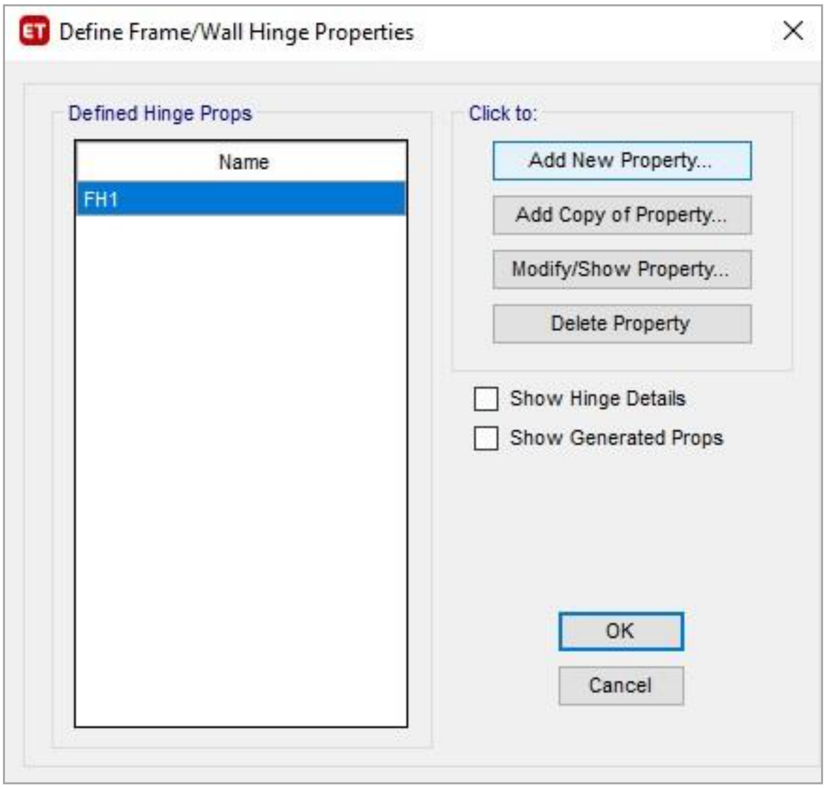
3.2.2. Step 2: Defining the Properties of a Master Hinge

Once the reinforcement is defined for each column section, the fiber hinges can be directly defined and assigned to these sections. The software will automatically define concrete and steel fibers for each section and assign the corresponding stress-strain curves.

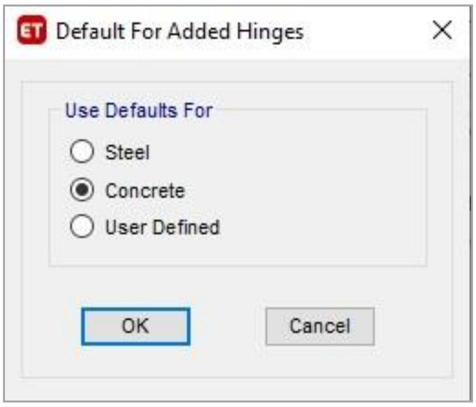
→ To automatically define the fiber hinges, the first step is to define a master fiber hinge which should serve as a template for all other automatically generated fiber hinges. Click Define > Section properties > Frame/Wall Nonlinear Hinges.



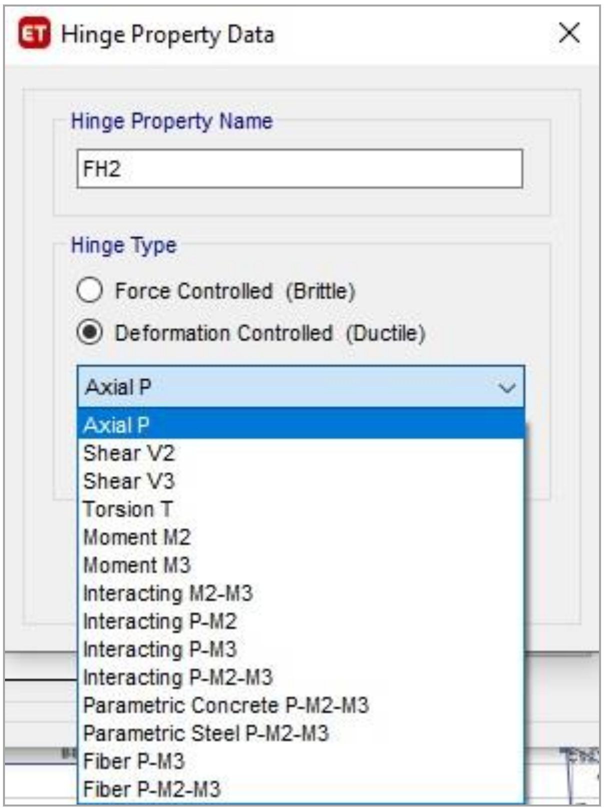
→ In the following window, click the “Add New Property” to define the properties of a hinge.



→ The following form will present the options available for the type of material. For the RC columns, select “Concrete”.

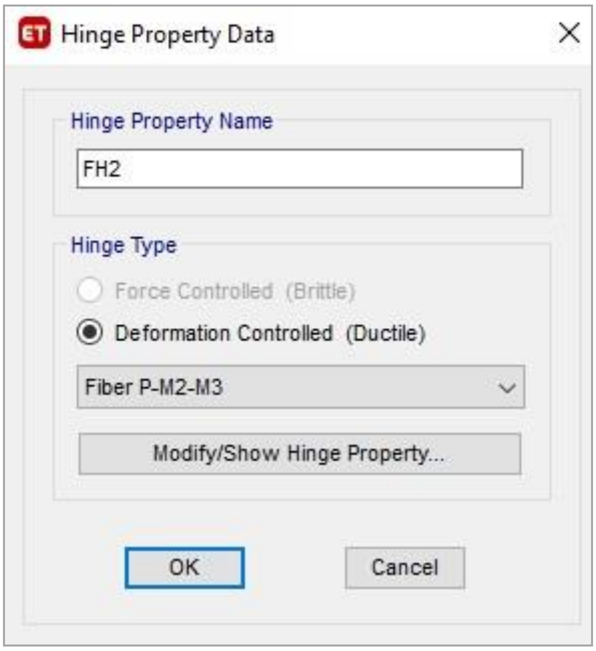


→ Click OK. The following form is used to select the name and basic type of hinge. You can select either a force-controlled or deformation-controlled hinge.

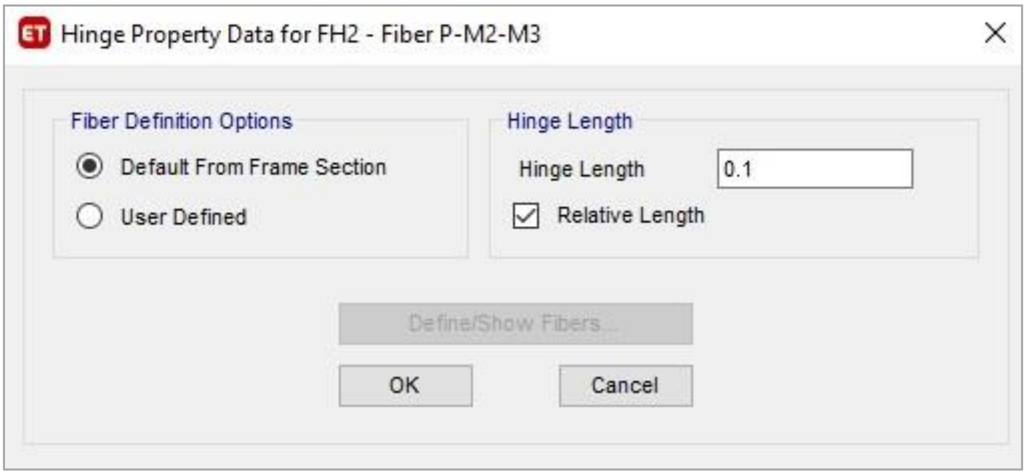


→ Select the “Deformation Controlled (Ductile)” option.

→ Select the fiber P-M2-M3 hinge and click on the “Modify/Show Hinge Property”.



→ The following window will appear.



→ Under the "Fiber Definition Options", one can select one of the following two option.

- "Default from Frame Section": This option is used for the automatic generation of concrete and steel fibers from the frame section properties (i.e., cross-sectional size, material properties and reinforcement).
- "User Defined": This option is used to manually define all fibers for a particular cross-section. For this option, the properties of all fibers (their coordinates, areas and material type) need to be entered manually. Click on "Define/Show Fibers" to manually define the fibers.

→ In the hinge length option, the fiber length can be specified either in terms of absolute or relative length. The check box can be selected if relative length is being entered. At maximum, the length of a fiber hinge can be equal to the length of column. For such case, the relative length equal to 1 will be used (or the absolute height of column). However, using the fiber hinge for the whole column height may significantly increase the analysis time. A convenient option is to use the plastic hinge only at the ends of columns (i.e., the zones of maximum anticipated inelastic action). Therefore, generally the "lumped fiber hinges" with some finite hinge lengths are used at both ends of columns.

→ The plastic zone (or plastic length) has remained an area of detailed research in last 2 to 3 decades. Several guidelines are available to estimate the length or zone where the concentrated inelastic action can be assumed. One such guideline is as follows.

$$L_p = 0.08\,L + 0.022\,f_{ye}\,d_{bl} \geq 0.044\,f_{ye}\,d_{bl} \qquad \text{(MPa)}$$

Where

L_p = plastic hinge length.

L = Distance from the critical section of plastic hinge to the point of contraflexure.

f_{ye} = Expected yield strength of longitudinal reinforcement.

d_{bl} = Diameter of longitudinal reinforcement.

→ Alternatively, a fiber hinge length can be assumed to be a fraction of cross section depth (generally half of the depth).

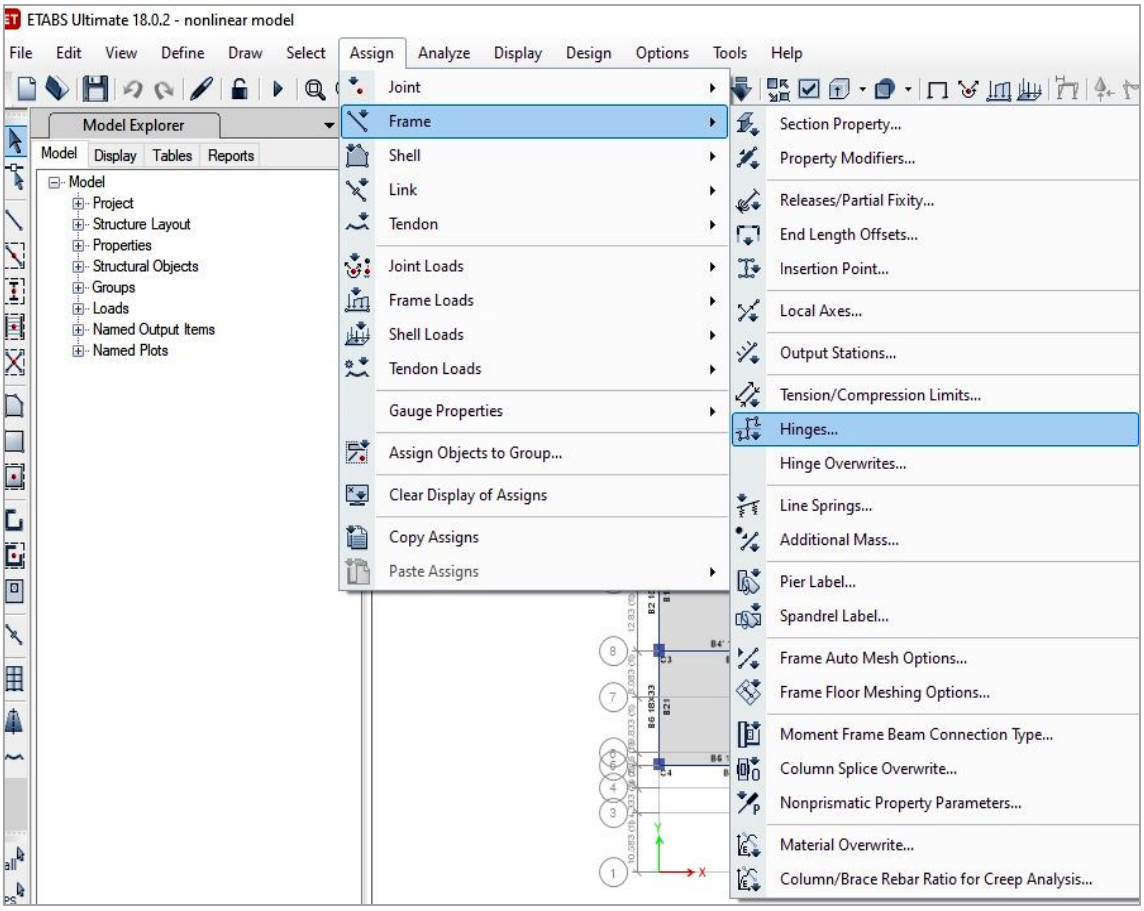
→ In this example, the hinge length is assumed to be the 10% of total length of column. Therefore, the check box should be check with a relative length of 0.1.

→ Click "OK" to complete the process of fiber (P-M2-M3) hinge definition. This master hinge (with automatic generation of material fibers using frame cross-section properties) can now be assigned to all column elements at their both ends.

3.2.3. Automatic Generation and Assignment of P-M2-M3 Hinges to RC Columns

→ To automatically generate and assign the column fiber hinges, select all the frame elements (columns).

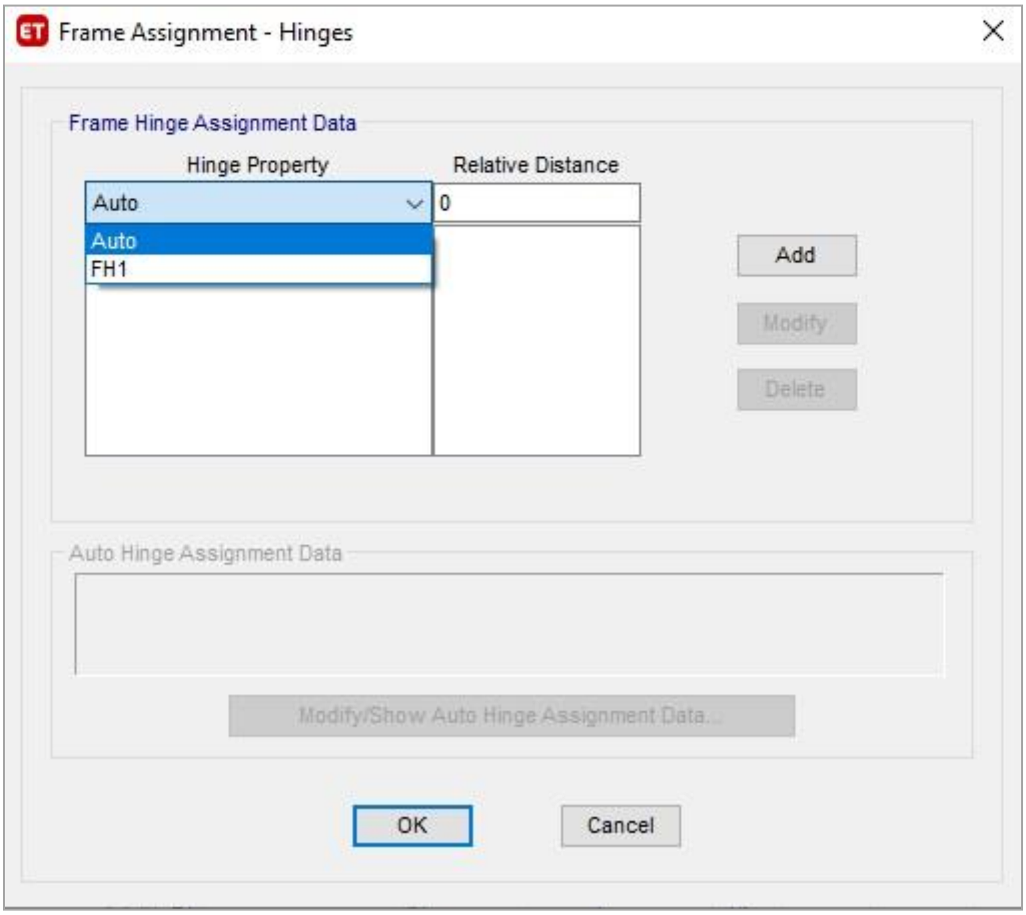
→ Click the Assign > Frame > Hinges to assign the hinges to the selected frame elements.



→ The following window will appear. Select the previously defined master hinge from the dropdown list and enter a relative distance equal to 0. Click "Add".

→ Again, select the previously defined master hinge from the dropdown list and enter a relative distance equal to 1. Click "Add".

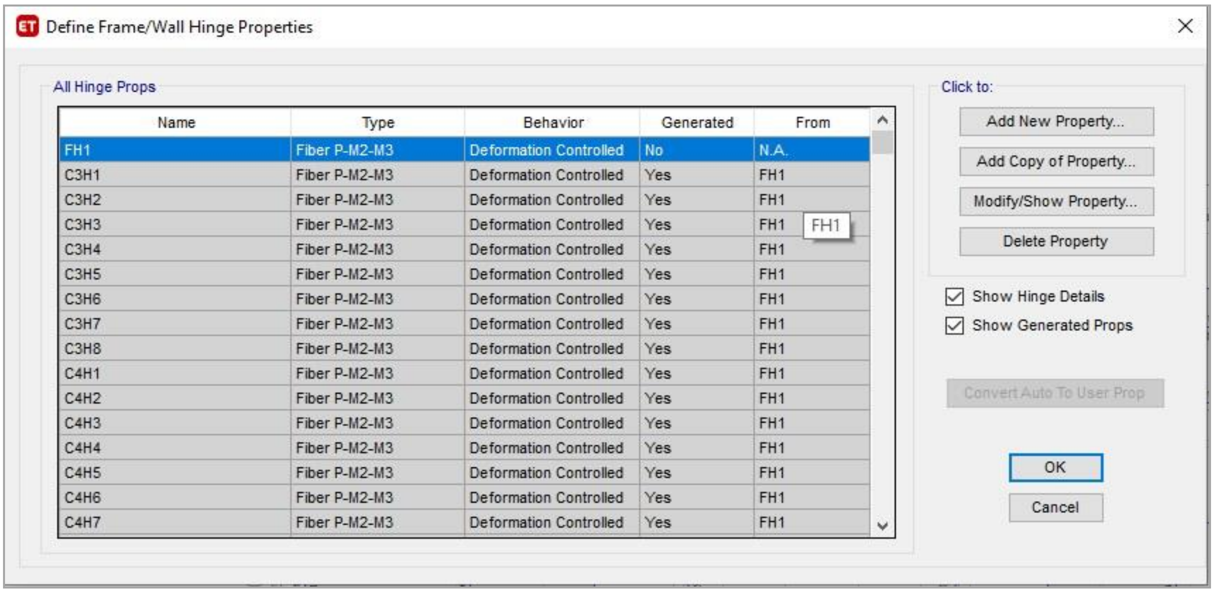
→ The program will generate two new fiber hinges (using the master hinge as a template) and lump them at both ends of the selected columns (with each hinge length equal to 10% of the total column length). The concrete and steel fibers are defined automatically in these generated fiber hinges.



→ All columns can be assigned with the fiber hinges using these automatically generated hinges.

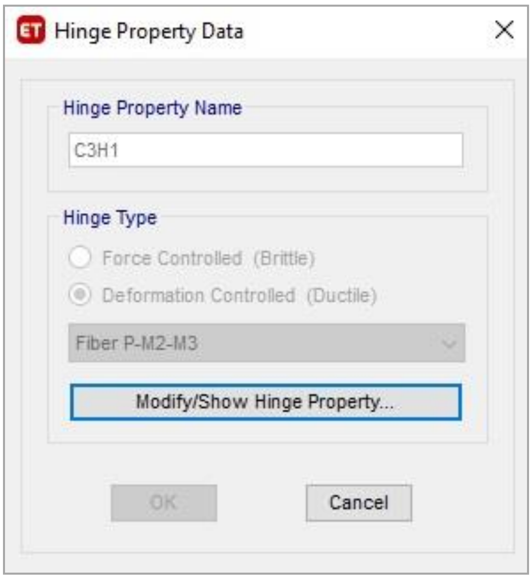
→ All the manually defined and automatically generated hinges can now be seen at Define > Section Properties > Frame/Wall Nonlinear Hinges.

→ Click on the check boxes “Show Hinge Details” & “Show Generated Properties”. The form will be populated will all the manually defined and automatically generated hinges.

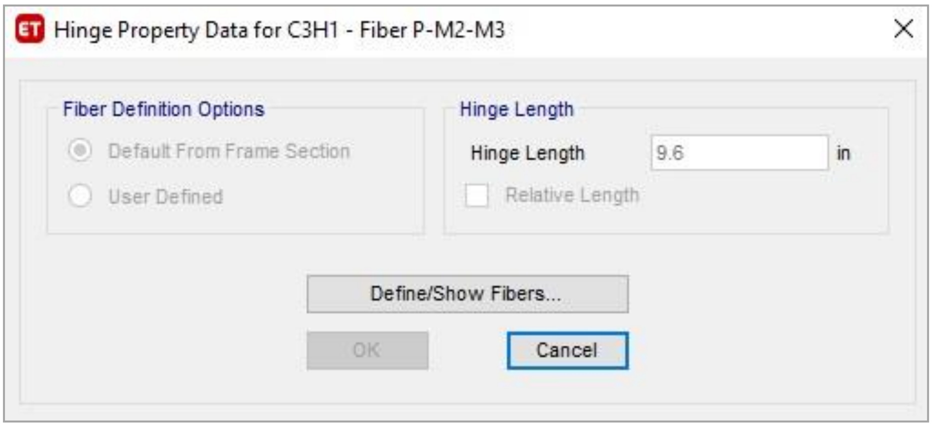


→ In this window, the basic detailed of each generated "Fiber P-M2-M3" hinge can be seen. This include their behavior (Deformation Controlled) and whether it is automatically generated or manually defined. In case of generated hinges, the master hinge is also shown in the "From" column.

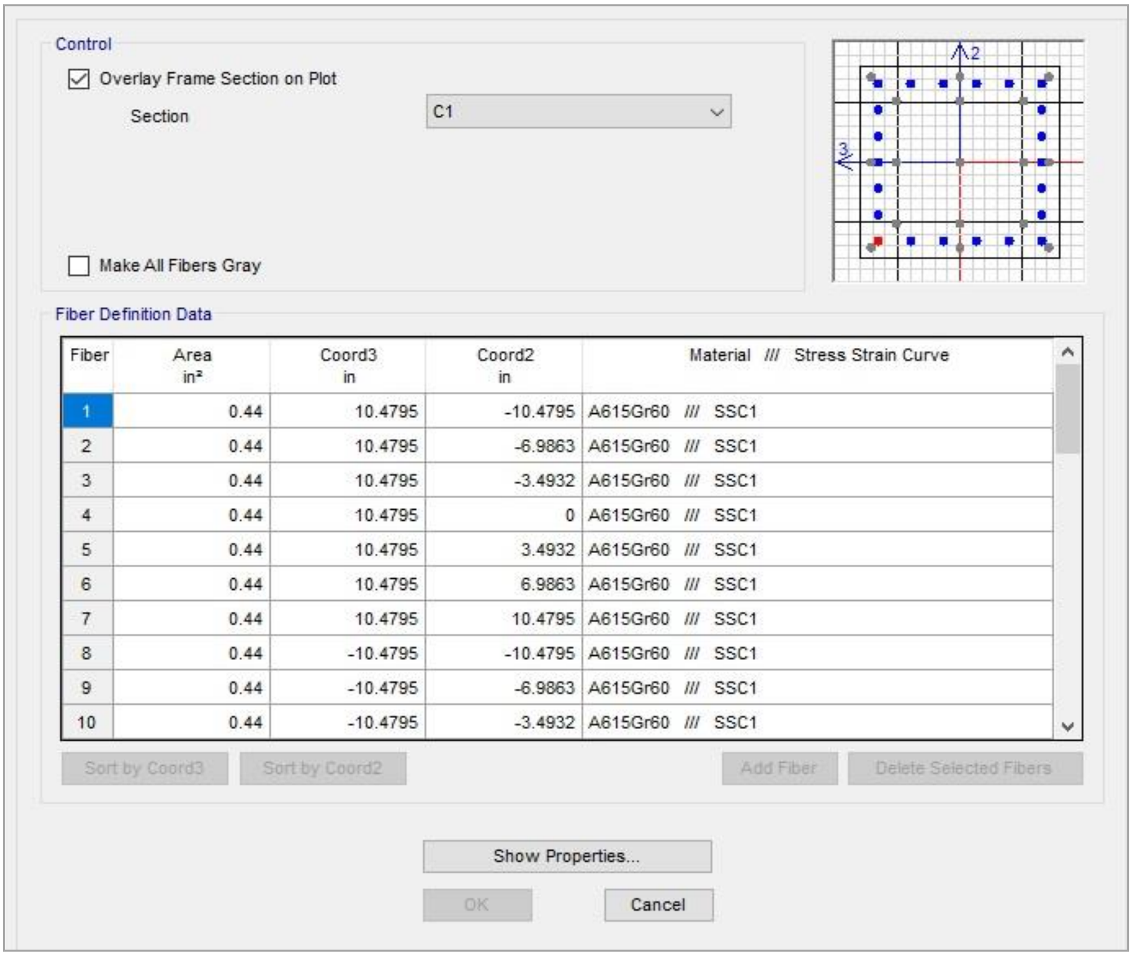
→ Click on any automatically generated fiber hinge and Click "Modify/Show Property". The following window will appear.



→ Click on the "Modify/Show Hinge Property" button to check the properties of this automatically generated hinge.



→ Click on the "Define/Show Fibers" option. The following form will open.



Fiber	Area in²	Coord3 in	Coord2 in	Material /// Stress Strain Curve
1	0.44	10.4795	-10.4795	A615Gr60 /// SSC1
2	0.44	10.4795	-6.9863	A615Gr60 /// SSC1
3	0.44	10.4795	-3.4932	A615Gr60 /// SSC1
4	0.44	10.4795	0	A615Gr60 /// SSC1
5	0.44	10.4795	3.4932	A615Gr60 /// SSC1
6	0.44	10.4795	6.9863	A615Gr60 /// SSC1
7	0.44	10.4795	10.4795	A615Gr60 /// SSC1
8	0.44	-10.4795	-10.4795	A615Gr60 /// SSC1
9	0.44	-10.4795	-6.9863	A615Gr60 /// SSC1
10	0.44	-10.4795	-3.4932	A615Gr60 /// SSC1

→ It can be seen that the list of concrete and steel fibers is automatically populated for this particular column cross-section. The coordinates (locations), areas and material stress-strain curves are also assigned to each fiber. Generally, for each reinforcing bar in the cross-section, one steel fiber is defined. If the total areas of all steel fibers area are summed, it would be equal to the actual longitudinal reinforcement area of the column cross-section. Similarly, the total area of all concrete and steel fibers area should be equal to the cross-sectional area of the column.

Chapter 4

Nonlinear Modeling of RC Beams using the Plastic Hinge Modeling Approach

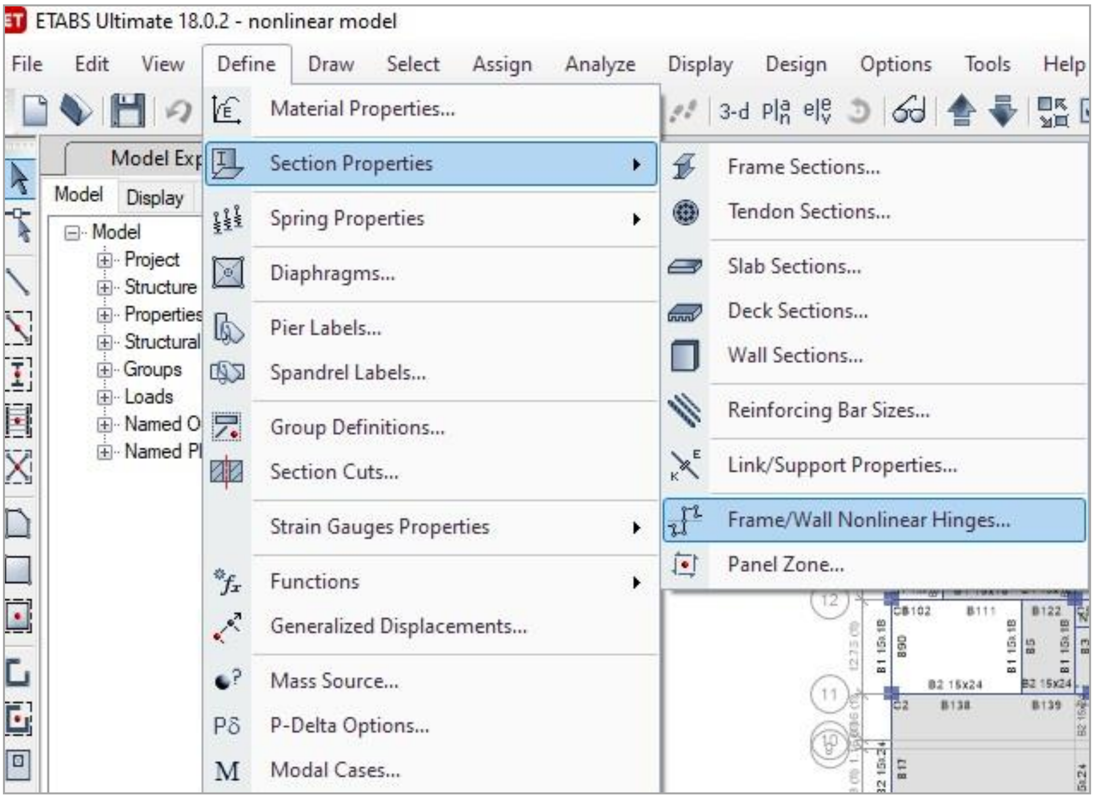
This section describes a step-by-step procedure to model the RC beams in a building using the uncoupled M3 hinges in CSI ETABS.

The uncoupled M3 hinges can either be defined manually or through automated procedure implemented in CSI ETABS. For manual definition, the properties of these hinges should be manually defined and assigned to each beam. For the automatic definition, first the actual reinforcement should be defined for each beam cross-section. *Generally, for linear elastic modeling (for initial analysis and design), the actual beam reinforcement is either not known or not defined in ETABS. However, this becomes a necessary step if one intends to use the automatic definition of hinges.* Alternatively, the program also has the feature to design the beams and use that reinforcement in the automatic generation of plastic hinges.

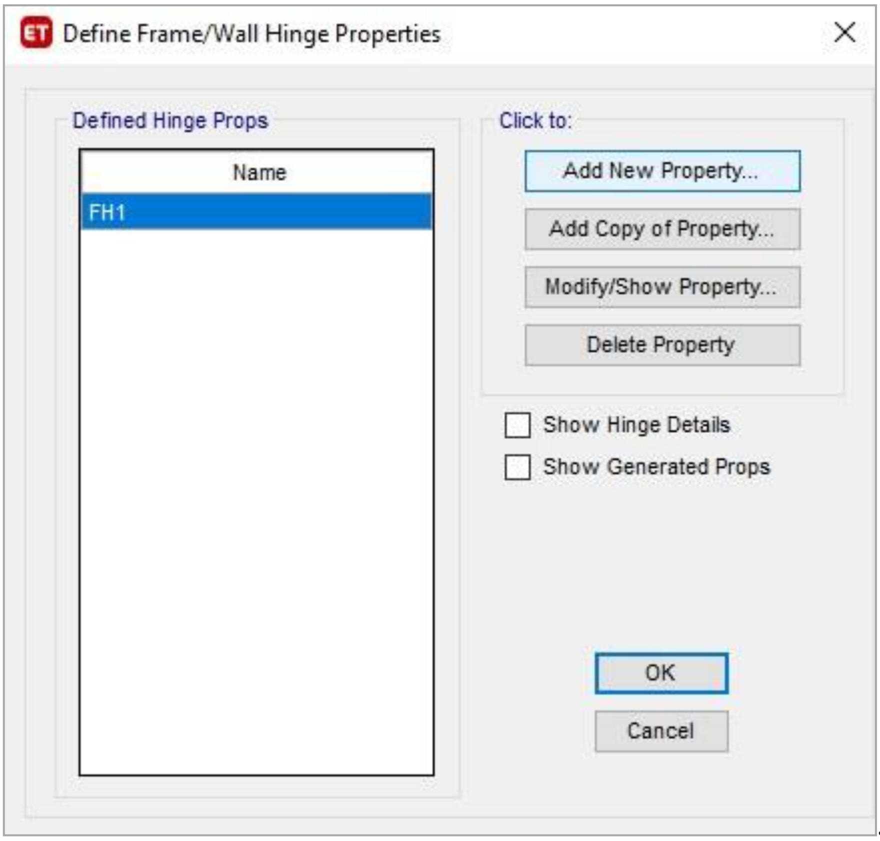
4.1 Manual Definition of M3 Plastic Hinges for RC Beams

4.1.1. Step 1: Defining the Hinge Properties

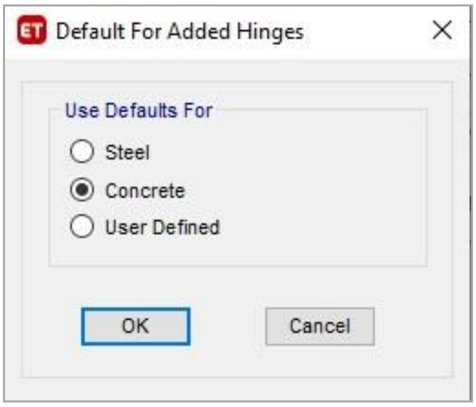
→ To define an M3 hinge, Click Define > Section properties > Frame/Wall Nonlinear Hinges.



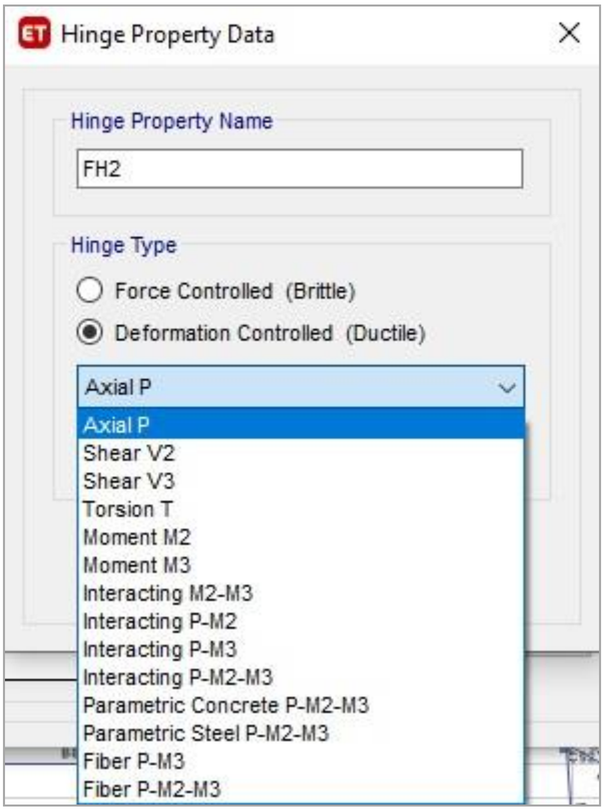
→ In the following window, click the "Add New Property" to define the properties of a hinge.



→ The following form will present the options available for the type of material. For the RC columns, select "Concrete".

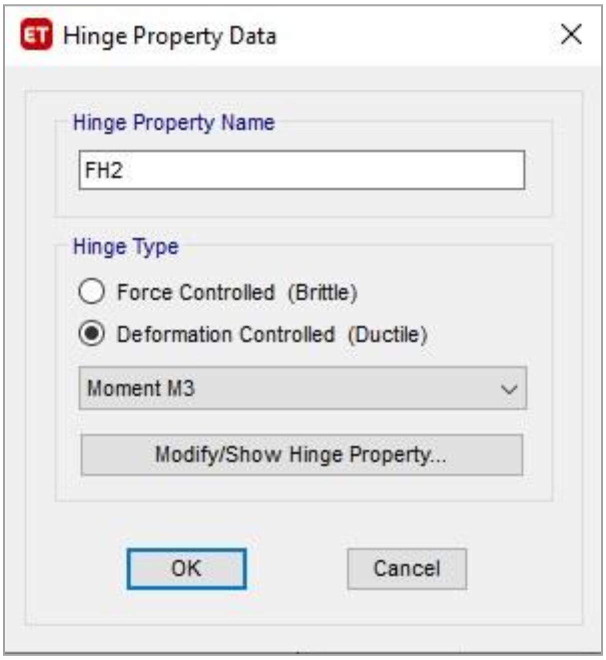


→ Click OK. The following form is used to select the name and basic type of hinge. You can select either a force-controlled or deformation-controlled hinge.

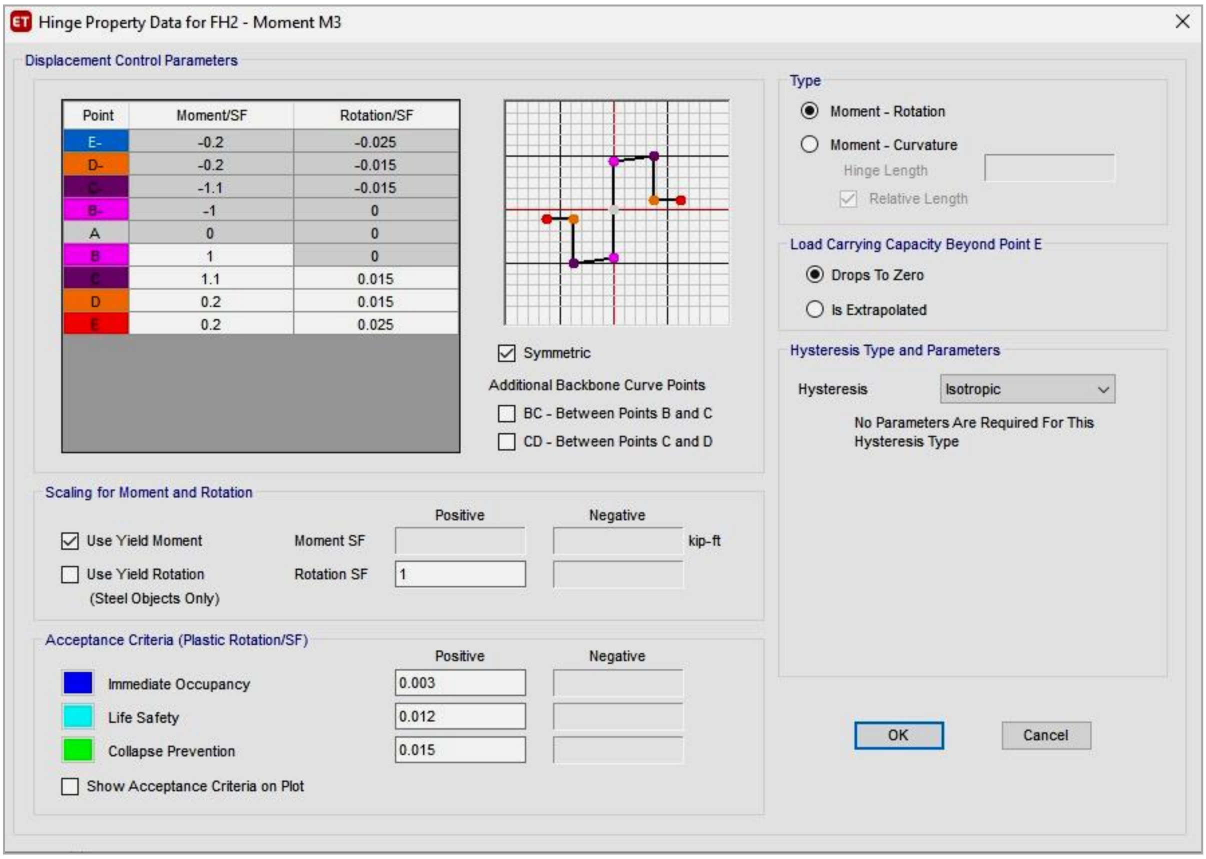


→ Select the "Deformation Controlled (Ductile)" option.

→ Select the Moment M3 hinge and click on the "Modify/Show Hinge Property".



→ Click on the "Modify/Show Property" option, the following window appears.



→ In this form, select the "Moment - Rotation" under the "Type".

→ Enter the coordinates of moment vs. rotation backbone curve. The backbone curve has five points on both positive and negative sides (A, B, C, D and E). Point A is always at the origin, point B is at the yielding. Since the hinge behavior is rigid plastic, no deformation occurs in the hinge up to this point B. When hinge unloads elastically, it does without any plastic deformation following the slope of AB line. Point C represents the ultimate capacity. The post-yield slope is generally assumed to be 10% of the initial slope. Point D represents the residual strength and point E is the total failure point.

→ By default, the values of moment and rotation are normalized by scale factors which can be selected as the yield moment and yield rotation respectively for conveniently defining the normalized backbone. The point B will correspond to a value of 1 (normalized moment).

→ As discussed in Chapter 2, the inelastic force-deformation curves (e.g., moment-rotation curve) can be obtained through experiments, analysis or from guidelines e.g., ASCE 41-17. The ASCE 41-17 prescribes the following force-deformation backbone curves which can be assigned to any inelastic component.

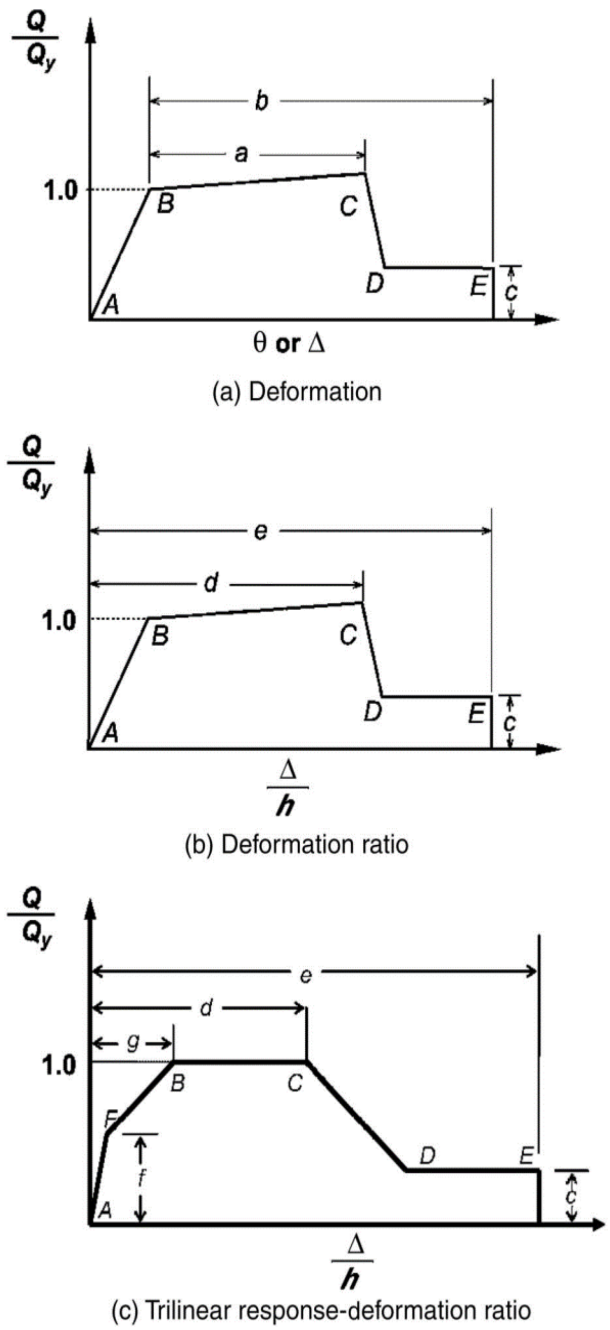


Figure 4-1: Generalized force-deformation relation for concrete elements or components.

→ The Figure (a) shows a general force-deformation behavior. The values a, b & c in this Figure (a) are called the modeling parameters. In case of moment-curvature relationship, the values a and b are the plastic rotations starting from the yield point. (a is the plastic rotation starting from yield to the ultimate capacity point, while b is the plastic rotation starting from yield to the total failure point). The value c is the ratio of residual strength to the ultimate strength.

→ Based on the moment-rotation type plastic hinge modeling approach, ASCE 41 provides the following two sets of values for all important structural components including beams and columns.

- Modeling parameters. i.e., the values of a, b, and c.
- Acceptance criteria. i.e., the values of plastic rotations for three performance levels (IO, LS, and CP).

→ Both sets of values can be used in the manual definition of plastic hinges for different elements.

→ The Table 10-7 in ASEC 41-17 provides these values for reinforced concrete beams and is shown below.

Table 10-7. Modeling Parameters and Numerical Acceptance Criteria for Nonlinear Procedures—Reinforced Concrete Beams

Conditions			Modeling Parameters[a]			Acceptance Criteria[a]		
			Plastic Rotation Angle (radians)		Residual Strength Ratio	Plastic Rotation Angle (radians) — Performance Level		
			a	*b*	*c*	IO	LS	CP
Condition i. Beams controlled by flexure[b]								
$\frac{\rho-\rho'}{\rho_{bal}}$	Transverse reinforcement[c]	$\frac{V^d}{b_w d \sqrt{f'_{cE}}}$						
≤0.0	C	≤3 (0.25)	0.025	0.05	0.2	0.010	0.025	0.05
≤0.0	C	≥6 (0.5)	0.02	0.04	0.2	0.005	0.02	0.04
≥0.5	C	≤3 (0.25)	0.02	0.03	0.2	0.005	0.02	0.03
≥0.5	C	≥6 (0.5)	0.015	0.02	0.2	0.005	0.015	0.02
≤0.0	NC	≤3 (0.25)	0.02	0.03	0.2	0.005	0.02	0.03
≤0.0	NC	≥6 (0.5)	0.01	0.015	0.2	0.0015	0.01	0.015
≥0.5	NC	≤3 (0.25)	0.01	0.015	0.2	0.005	0.01	0.015
≥0.5	NC	≥6 (0.5)	0.005	0.01	0.2	0.0015	0.005	0.01
Condition ii. Beams controlled by shear[b]								
Stirrup spacing ≤ *d*/2			0.0030	0.02	0.2	0.0015	0.01	0.02
Stirrup spacing > *d*/2			0.0030	0.01	0.2	0.0015	0.005	0.01
Condition iii. Beams controlled by inadequate development or splicing along the span[b]								
Stirrup spacing ≤ *d*/2			0.0030	0.02	0.0	0.0015	0.01	0.02
Stirrup spacing > *d*/2			0.0030	0.01	0.0	0.0015	0.005	0.01
Condition iv. Beams controlled by inadequate embedment into beam–column joint[b]								
			0.015	0.03	0.2	0.01	0.02	0.03

Note: f'_{cE} in lb/in.^{2} (MPa) units.
[a] Values between those listed in the table should be determined by linear interpolation.
[b] Where more than one of conditions i, ii, iii, and iv occur for a given component, use the minimum appropriate numerical value from the table.
[c] "C" and "NC" are abbreviations for conforming and nonconforming transverse reinforcement, respectively. Transverse reinforcement is conforming if, within the flexural plastic hinge region, hoops are spaced at ≤ *d*/3, and if, for components of moderate and high ductility demand, the strength provided by the hoops (V_s) is at least 3/4 of the design shear. Otherwise, the transverse reinforcement is considered nonconforming.
[d] *V* is the design shear force from NSP or NDP.

→ In order to obtain the modeling parameters & acceptance criteria for a particular reinforced concrete beam, the following three quantities are required in the Table 10-7 of ASCE 41-17.

(a) $\frac{\rho-\rho'}{\rho_{bal}}$

(b) The information whether the transverse reinforcement is "conforming" or not, and

(c) $\frac{V}{b_w\, d\sqrt{f_{cE}'}}$

Where;

ρ = Tensile reinforcement ratio.

ρ' = Compression reinforcement ratio.

ρ_{bal} = Balanced reinforcement ratio.

f_{cE}' = Expected concrete strength.

V = Design shear force of the beam.

b_w = Width of the beam cross-section.

d = Depth of the beam cross-section.

→ The design longitudinal reinforcements (top and bottom) can be used to determine the factor $\frac{\rho-\rho'}{\rho_{bal}}$.

→ In the transverse reinforcement column of Table 10-7, the symbols "C" and "NC" are abbreviations for conforming and nonconforming transverse reinforcement, respectively. Transverse reinforcement is conforming if, within the flexural plastic hinge region, hoops are spaced at ≤ d/3, and if, for components of moderate and high ductility demand, the strength provided by the hoops (V_s) is at least 3/4 of the design shear. Otherwise, the transverse reinforcement is considered nonconforming. The design shear reinforcement can be checked to confirm whether they are C or NC.

→ V is the design shear from the nonlinear static or dynamic procedures.

→ If the beams are designed according to the capacity design concept, the value of V can be determined using the following procedure. It is taken from NIST GCR 8-917-1 report titled, "Seismic Design of Reinforced Concrete Special Moment Frames – NEHRP Seismic Design Technical Brief 1".

Seismic design shear in plastic hinge regions is associated with maximum inelastic moments that can be develop at the ends of the beams when the longitudinal tension reinforcement is in the strain hardening region (assumed to develop 1.25 f_y).

Figure 4-2 illustrates a free body diagram of the beam is isolated from the frame, and is loaded by factored gravity loads (using the appropriate load combinations defined by ASCE 7) as well as the moments and shears acting at the ends of the beam. Assuming the beam is yielding in flexure, the beam end moments are set equal to the probable moment strengths M_{pr}.

ACI 318 defines the probable moment strength M_{pr} as follows.

Probable moment strength is calculated from conventional flexural theory considering the as-designed cross section, using strength reduction factor ϕ = 1.0, and assuming reinforcement yield strength equal to at least 1.25 f_y. The probable moment strength is used to establish requirements for beam shear strength, beam-column joint strength, and column strength as part of the capacity-design process.

Note: The overstrength factor 1.25 is thought to be a low estimate of the actual overstrength that might occur for a beam. Reinforcement commonly used in the U.S. has an average yield stress about 15 percent higher than the nominal value (f_y), and it is not unusual for the actual tensile strength to be 1.5 times the actual yield stress. Thus, if a reinforcing bar is subjected to large strains during an earthquake, stresses well above 1.25 f_y are likely. The main reason for estimating beam flexural overstrength conservatively is to be certain there is sufficient strength elsewhere in the structure to resist the forces that develop as the beams yield in flexure. The beam overstrength is likely to be offset by overstrength throughout the rest of the building as well. The factor 1.25 in ACI 318 was established recognizing all these effects.

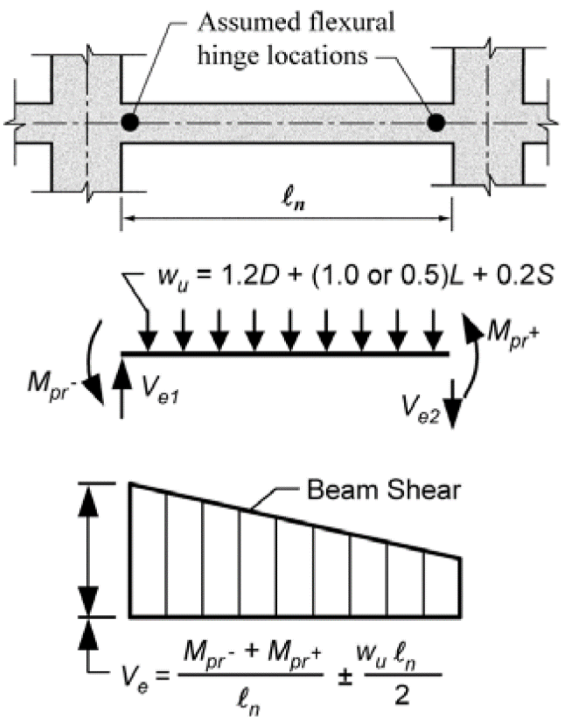


Figure 4-2: Beam shears are calculated based on provided probable moment strengths combined with factored gravity loads.

The values of probable moment strengths at both ends of the beam are calculated. The design shears are then calculated as the shears required to maintain moment equilibrium of the free body (that is, summing moments about one end to obtain the shear at the opposite end). This approach is intended to result in a conservatively high estimate of the design shears.

→ In summary, the value of V can be determined as follows.

$$V = \frac{M_{pr1} + M_{pr2}}{l} + \frac{w_u \times l}{2}$$

Where

l = Span length of the beam.

w_u = Ultimate factored load on that beam.

M_{pr1} = Probable moment at one end of beam.

M_{pr2} = Probable moment at other end of beam.

The ultimate factored load on the beam should be coming from the following load combination:

$$1.2\,D + E_v + (1 \text{ or } 0.5)L + 0.2\,S$$

Where

D = Dead load on the beam.

L = Live load on the beam.

S = Snow load on the beam if any.

E_v = Equivalent static load for the vertical component of earthquake.

The value of E_v depends on the seismic code used for the analysis and design.

For UBC 97, $E_v = 0.5\,C_a\,I\,D$, where C_a is the seismic coefficient and I is the importance factor.

→ Once the values of all three inputs is known for a particular beam element, the Table 10-7 can be used to determine the modeling parameters (a, b and c) & acceptance criteria (plastic rotations for IO, LS and CP).

→ If more than one row in Table 10-7 is applicable (for the inputs), the modeling parameters & acceptance criteria is determined using the linear interpolation.

→ The modeling parameters a, b and c can finally be used to input the moment-rotation behavior as follows [see Figure 4-1(a)].

Point	Moment (normalized to a scale factor) Scale factor = Yield moment)	Rotation (rad) (normalized to a scale factor) Scale factor = 1
E-	$-c$	$-b$
D-	$-c$	$-a$ (or slightly greater than $-a$ to avoid sudden strength loss)
C-	-1.1 to -1.3 (depending upon the assumed amount of post-yield strength)	$-a$
B-	-1	0
A	0	0
B+	1	0
C+	1.1 to 1.3 (depending upon the assumed amount of post-yield strength)	a
D+	c	a (or slightly greater than a to avoid sudden strength loss)
E+	c	b

→ The hysteretic behavior of RC beams can be suitably captured by either "Takeda", "Degrading" or "Pivot" hysteresis models. The Takeda model doesn't require any additional parameters and can be readily used.

→ In this example, let's use the "degrading" hysteresis model to define the cyclic behavior of moment-rotation backbone curve. The following parameters are used for this model.

- Initial energy factor at yield, f_0 = 1
- Energy factor at moderate deformation, f_1 = 0.45
- Energy factor at maximum deformation, f_2 = 0.35
- Moderate deformation level, $x1$, as a multiple of deformation scale factor = 5
- Maximum deformation level, $x2$, as a multiple of deformation scale factor = 10
- Accumulated deformation weighting factor, a = 0
- Stiffness degradation weighting factor, s = 1
- Larger-smaller weighting factor, w = 0.0

Check the option "Parameters are Symmetric" so that the same values are used for both compression and tension sides of the hysteresis model.

→ The option "Load carrying capacity beyond Point E" can be set as "Dropped to Zero".

→ The scale factor for moment is the yield moment of the beam cross-section. It can either be provided manually or determined automatically by the program based on the design reinforcement (if already provided in the beam cross-section). To use the automatic determination of yield moment, check the option of "Use Yield Moment" in the main window. The actual longitudinal reinforcement must be provided in the cross-section definition prior to the use of this option. Alternatively, the yield moment in both positive and negative bending directions can be manually calculated and provided as an input. Therefore, the role of design longitudinal reinforcement is vital in determining all input parameters for the plastic hinge including the modeling parameters, acceptance criteria the yield moment etc.

→ The scale factor for rotation is set to 1.

→ Enter the acceptance criteria determined from the Table 10-7 (in the form of plastic rotations in both positive and negative sides). It can be seen on the force-deformation curve by checking the "Show Acceptance Criteria on Plot" option.

→ If the section is symmetrical and reinforcement on both sides is equal, check the "Symmetry" option available below the force deformation-curve. The modeling parameters and acceptance criteria can be entered only for one side and the same will be used for the other side.

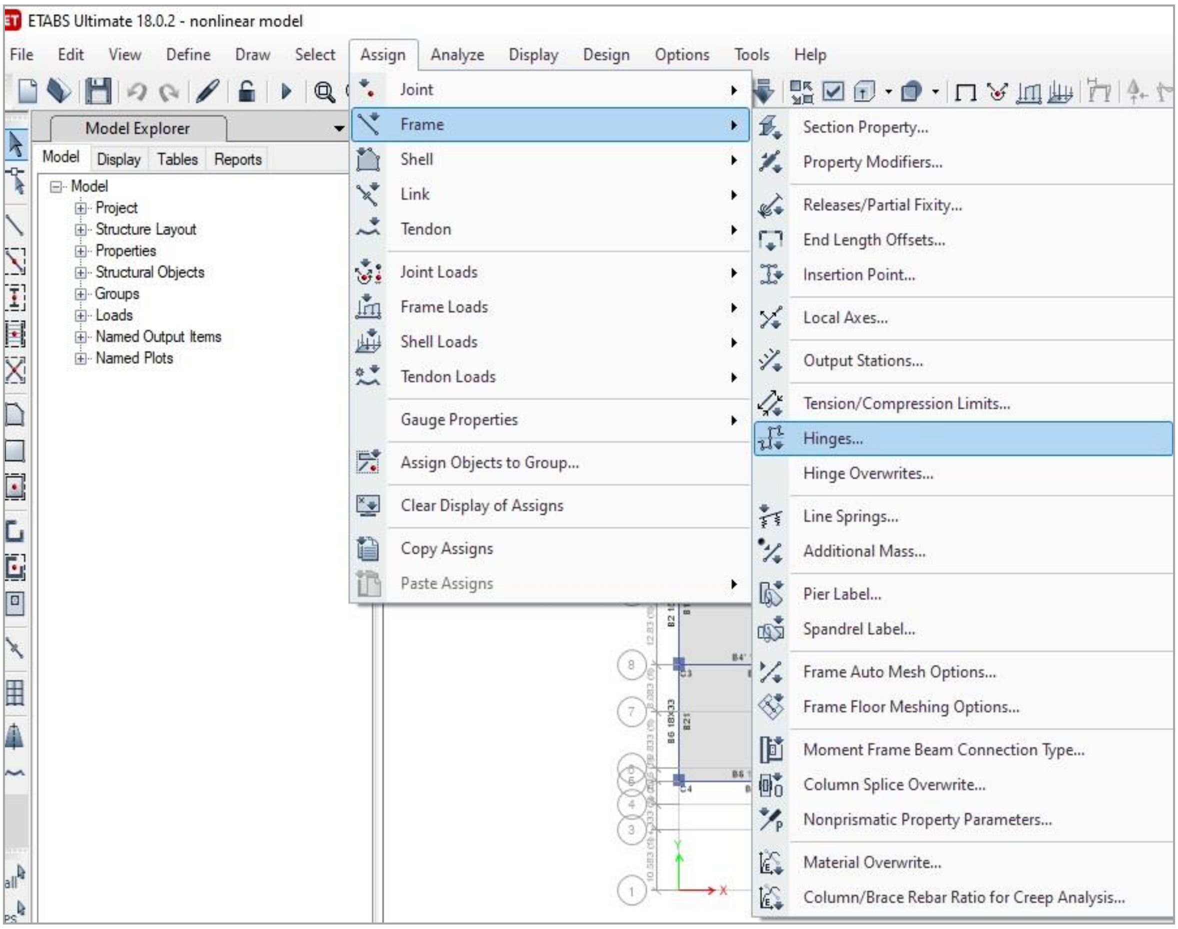
→ Click "OK" to complete the manual definition of moment M3 plastic hinge for a particular beam element.

→ Repeat this process for all beams.

4.1.2. Assigning M3 Plastic Hinges to RC Beams

→ To assign the hinges, select a beam element (or a group of beam elements to which the same hinge is to be assigned).

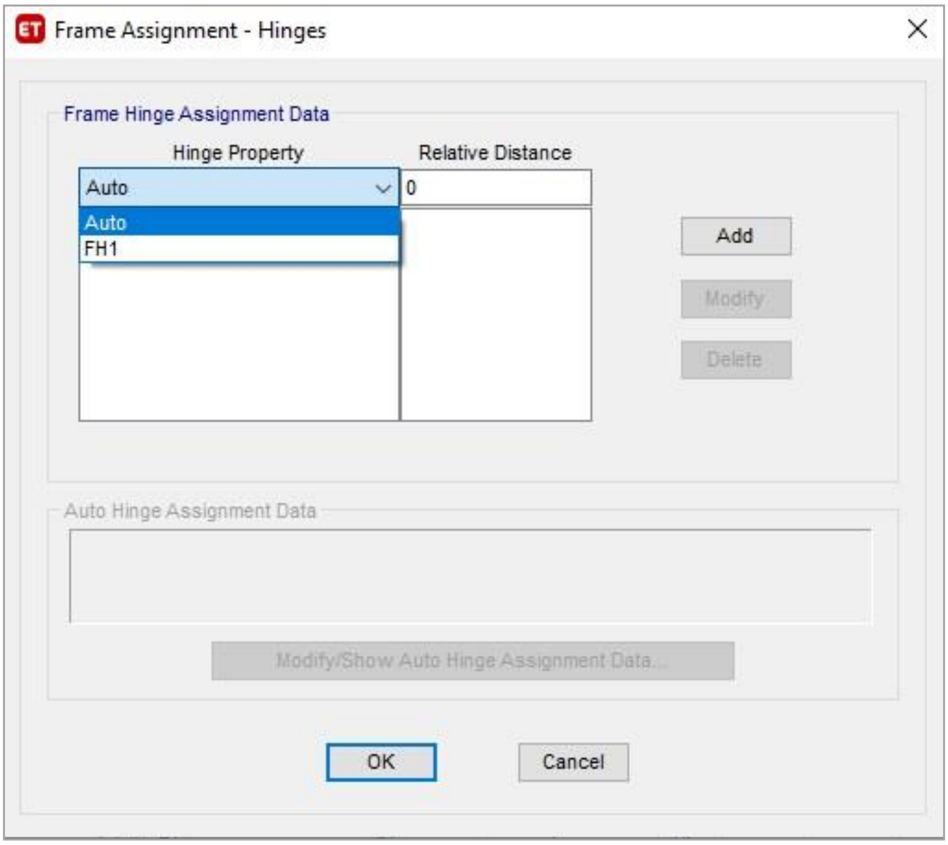
→ Click the Assign > Frame > Hinges to assign the hinges to the selected frame elements.



→ The following window will appear. Here, the desired hinge property can be assigned to the selected member from the list of defined hinges. Select the previously defined hinge (for the selected beam element) from the dropdown list and enter a relative distance equal to 0. Click “Add”.

→ Again, select the previously defined hinge from the dropdown list and enter a relative distance equal to 1. Click “Add”.

→ The program will generate two new M3 hinges (using the defined hinge as a template) and include them at both ends of the selected beams.



→ All beams can be assigned with the previously defined M3 hinges using this procedure.

→ All the manually defined and automatically generated hinges can now be seen at Define > Section Properties > Frame/Wall Nonlinear Hinges.

→ Click on the check boxes “Show Hinge Details” & “Show Generated Properties”. The form will be populated will all the manually defined and automatically generated hinges.

→ In this window, the basic detailed of each generated M3 hinge can be seen. This include their behavior (Deformation Controlled) and whether it is automatically generated or manually defined. In case of generated hinges, the master hinge is also shown in the “From” column.

→ Click on any automatically generated M3 hinge and Click “Modify/Show Property”. The following window will appear.

→ Click on the “Modify/Show Hinge Property” button to check the properties of this automatically generated M3 hinge.

4.2 Automated Definition of Plastic Hinges

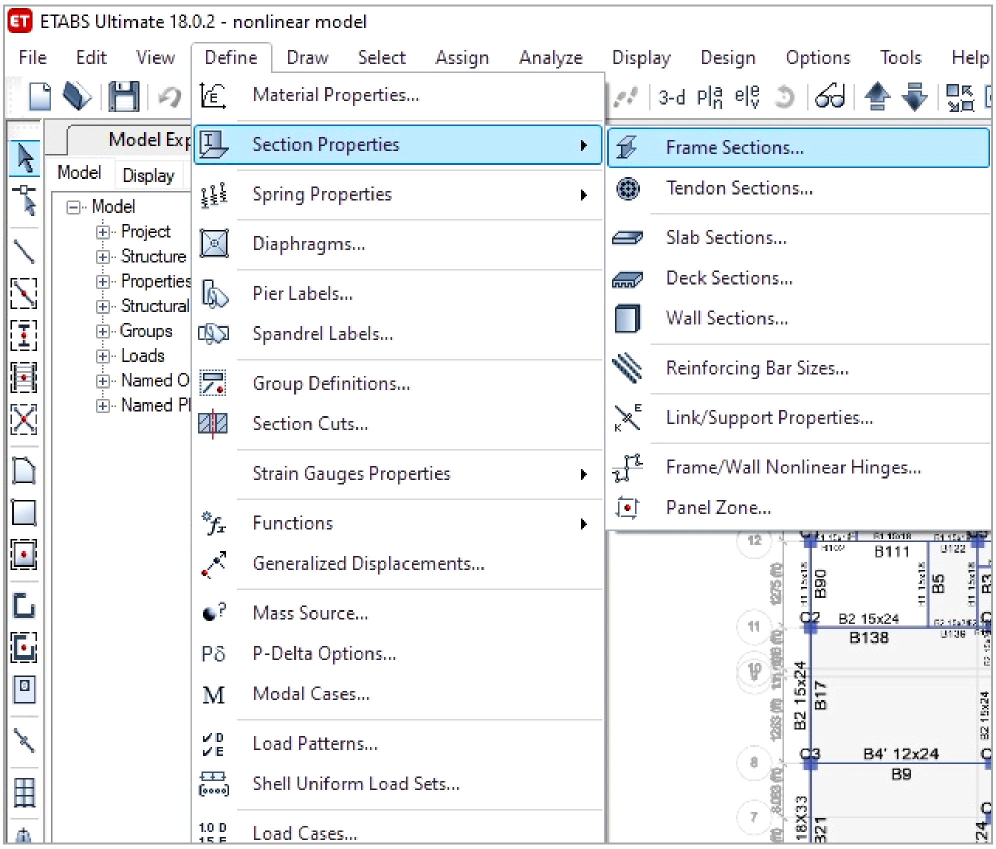
4.2.1. Step 1: Defining the Beam Reinforcements

CSI ETABS also allows the user to apply the auto hinges directly to the beam or column elements. By doing so, the software will automatically get the modeling parameters and acceptance criteria from the relevant tables of ASCE 41-13 and ASCE 41-17 (Table 10-7 of ASCE 41-17 in case of beams). In order to use the option of auto moment M3 hinges, first the design reinforcement need to be provided to the program. This can be provided through the "Reinforcement Overrides for Ductile Beams" option available while defining the beam cross-section. These reinforcement overrides are specified areas of longitudinal reinforcing steel that occur at the Top and Bottom of the left and right ends of the beam. These overrides are used by the program as follows:

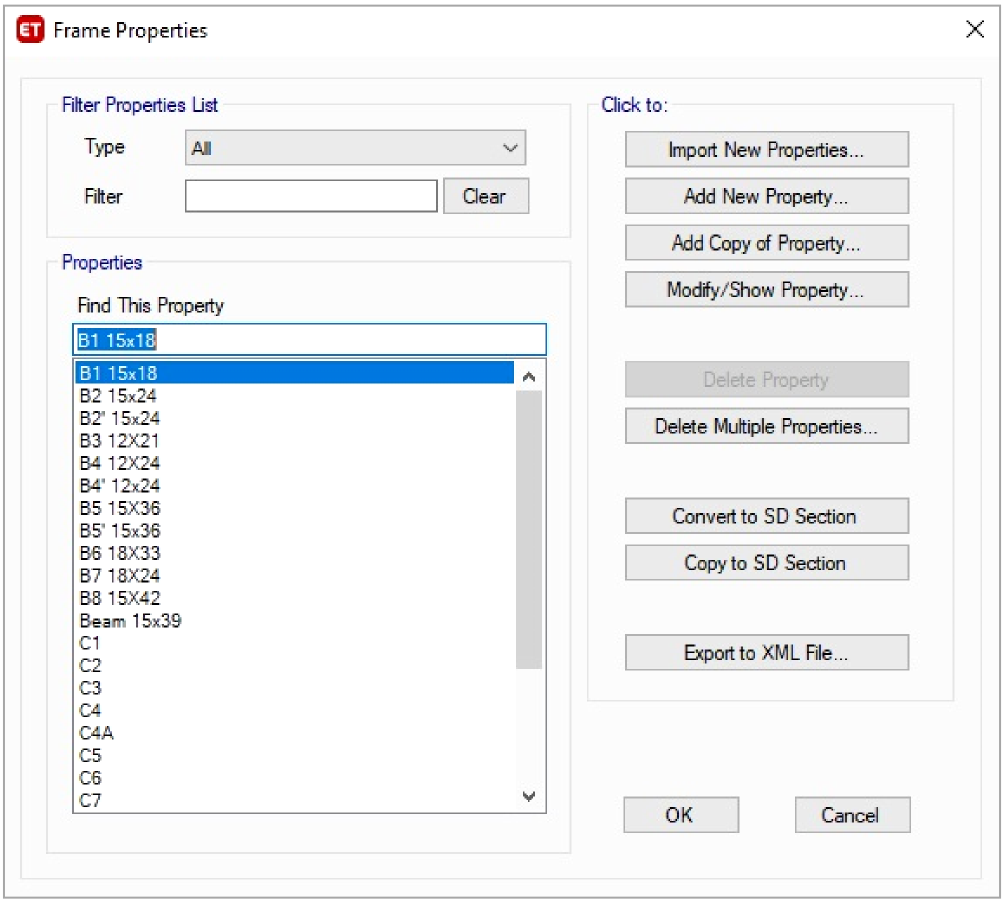
a) In the Concrete Frame Design postprocessor:
    - When the design shear in a concrete beam is to be based on provided longitudinal reinforcement (that is, the shear design is based on the moment capacity of the beam), the program compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity on which the shear design is based.
    - When the minimum reinforcing in the middle of a beam is to be based on some percentage of the reinforcing at the ends of the beam, the program compares the calculated required reinforcement at the ends of the beam with that specified in the reinforcement overrides and uses the larger value to determine the minimum reinforcing in the middle of the beam.
    - When the shear design of columns is to be based on the maximum moment that the beams can deliver to the columns, the program compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity of the beam.
b) For any degree of freedom in the frame nonlinear hinge properties assigned to a concrete member that is specified as default, the program calculates the hinge force-deformation properties based on the larger of the calculated required reinforcement at the ends of the beam (assuming that run the design has been run through the Concrete Frame Design postprocessor) and the specified reinforcement overrides.

Therefore, in order to use the automatic definition of hinges, the longitudinal reinforcement can be provided as reinforcement overrides while defining the beam cross-sections. The following procedure can be followed.

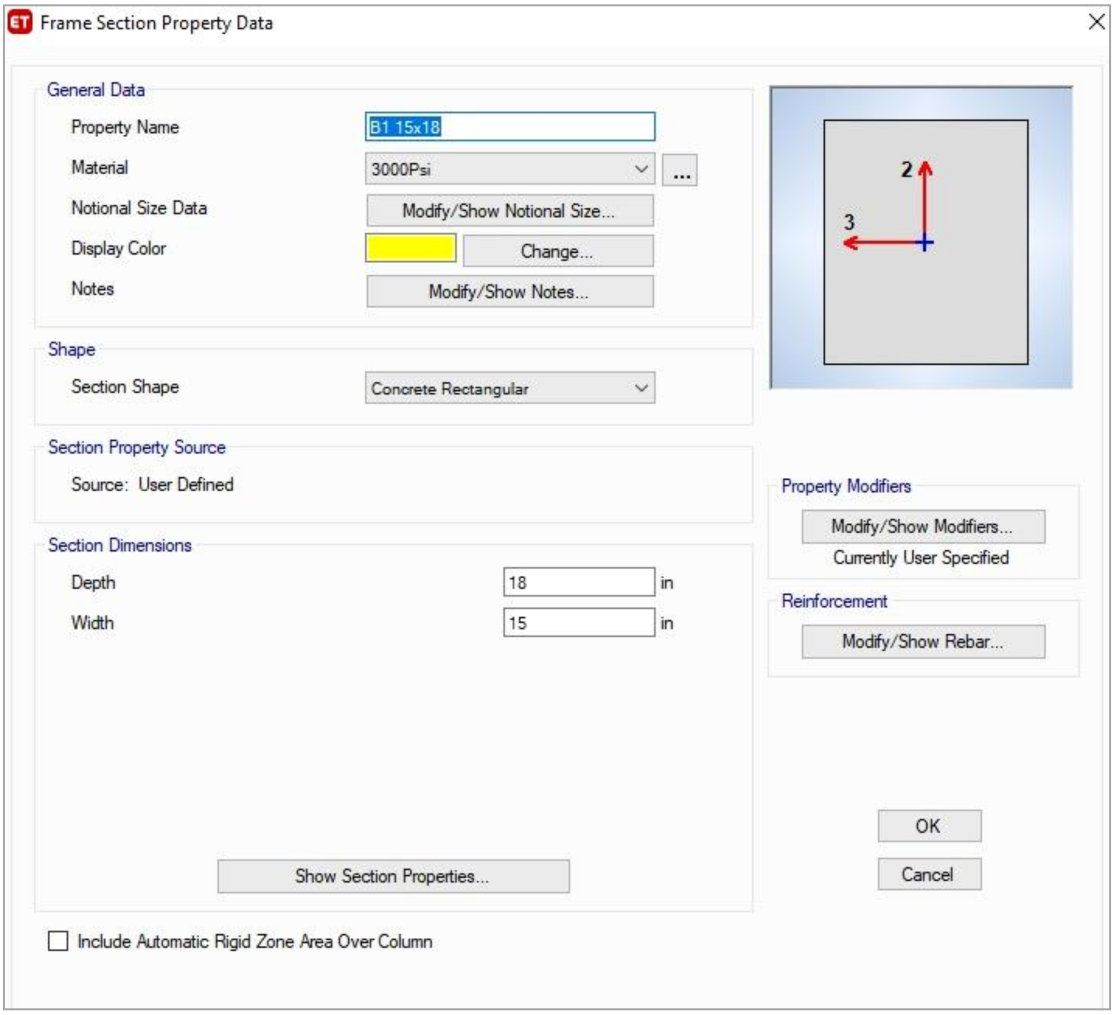
→ Click on Define > Section Properties > Frame Sections.



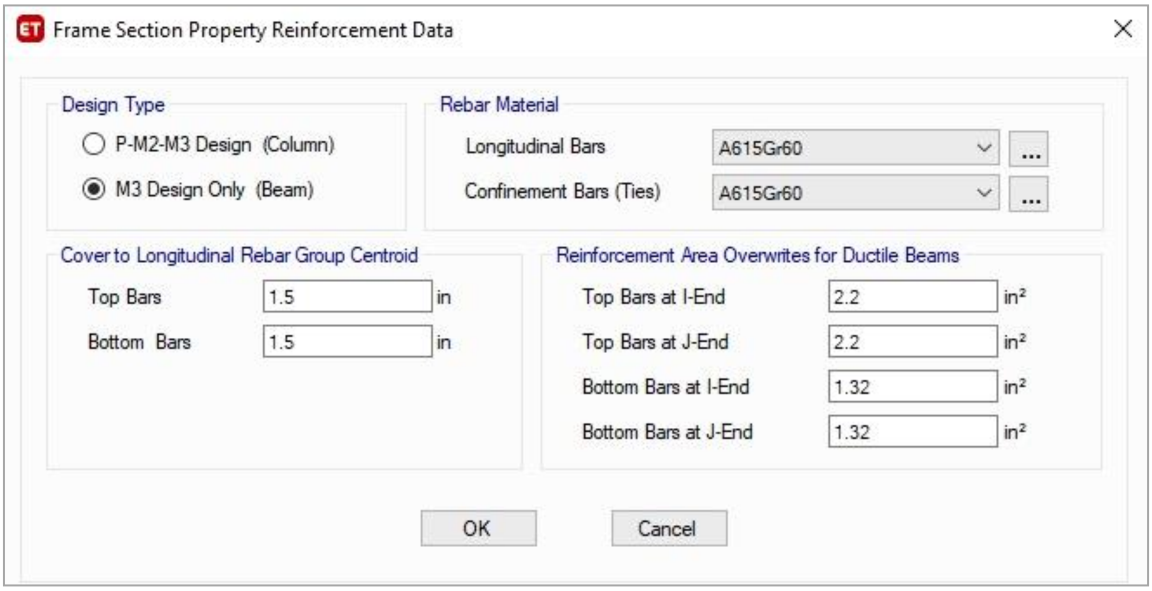
→ The following window will appear.



→ In the properties drop down menu, select that beam section property whose reinforcement should be entered for analysis purposes. Click on “Modify/Show Property” which will lead to the following window.



→ In the “Frame Section Property Data” window, click on the “Modify/Show Rebar”.



→ In the "Frame Section Property Reinforcement Data" window, select the M3 design only (Beams) for beam sections under the "Design Type".

→ Select the material of longitudinal and confinement bars under the "Rebar Material".

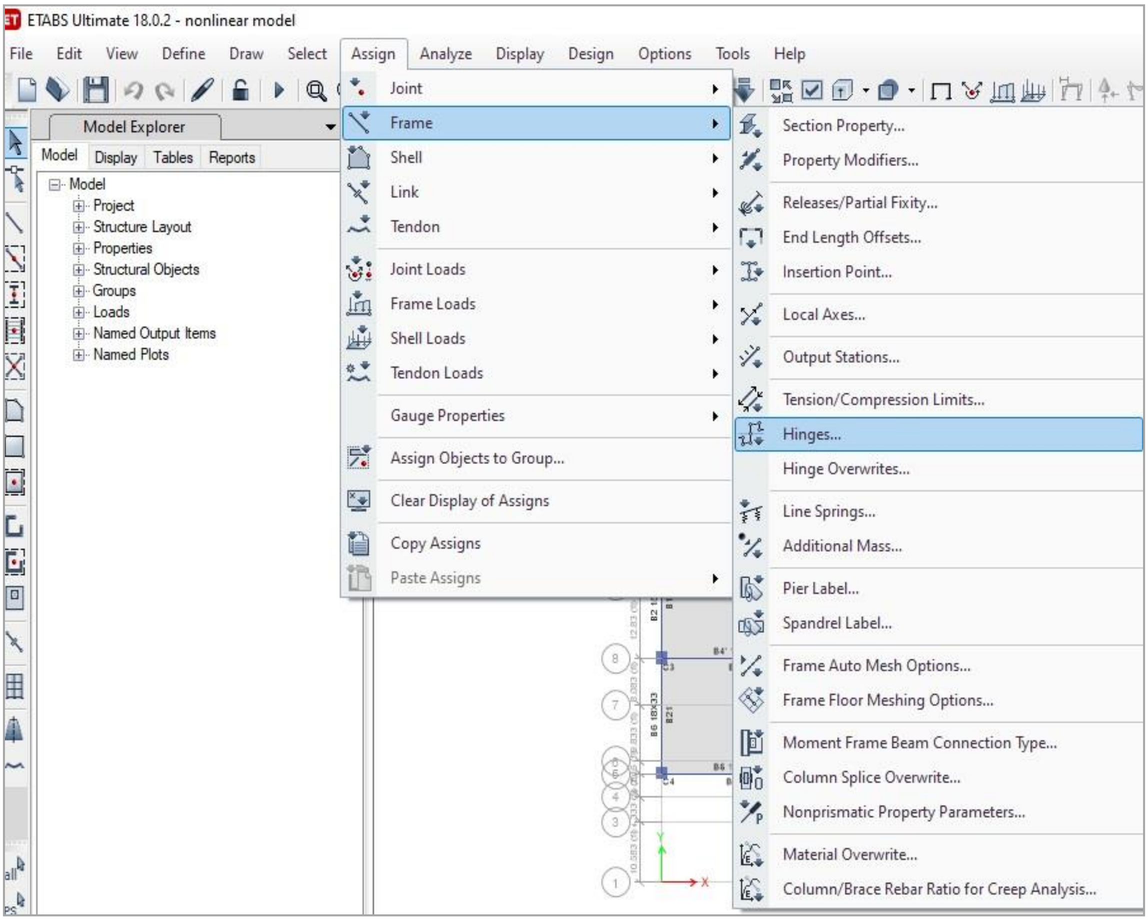
→ The clear cover from center of bars (for both top and bottom bars) can be provided under the "Cover to Longitudinal Rebar group Centroid".

→ Reinforcement area in beams can be specified by providing top and bottom bars at I-end and J-end in "Reinforcement Area Overwrites for Ductile Beams".

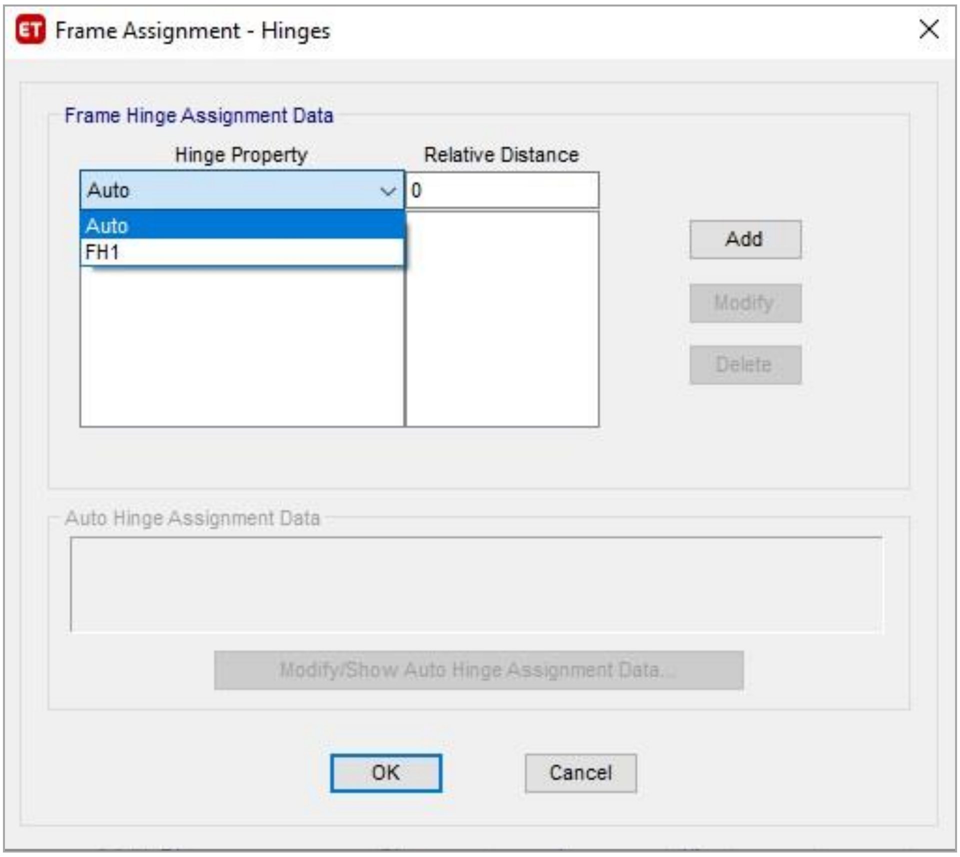
→ By clicking on "OK", the reinforcement for this frame section property is defined.

4.2.2. Step 2: Assigning the M3 Hinges to Beams

→ Select the beam elements and go to the Assign > Frame > Hinges as shown below.

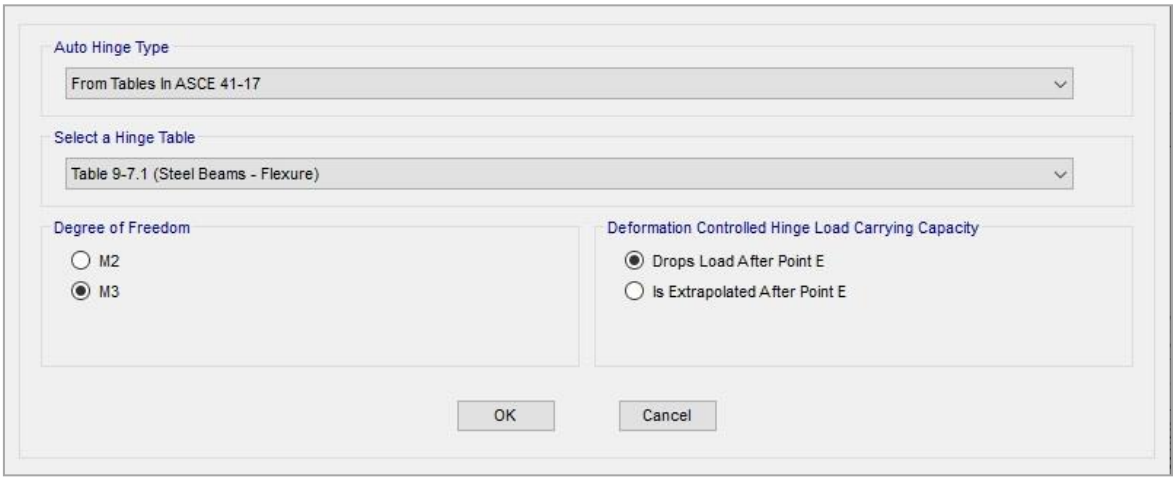


→ The following window will appear.



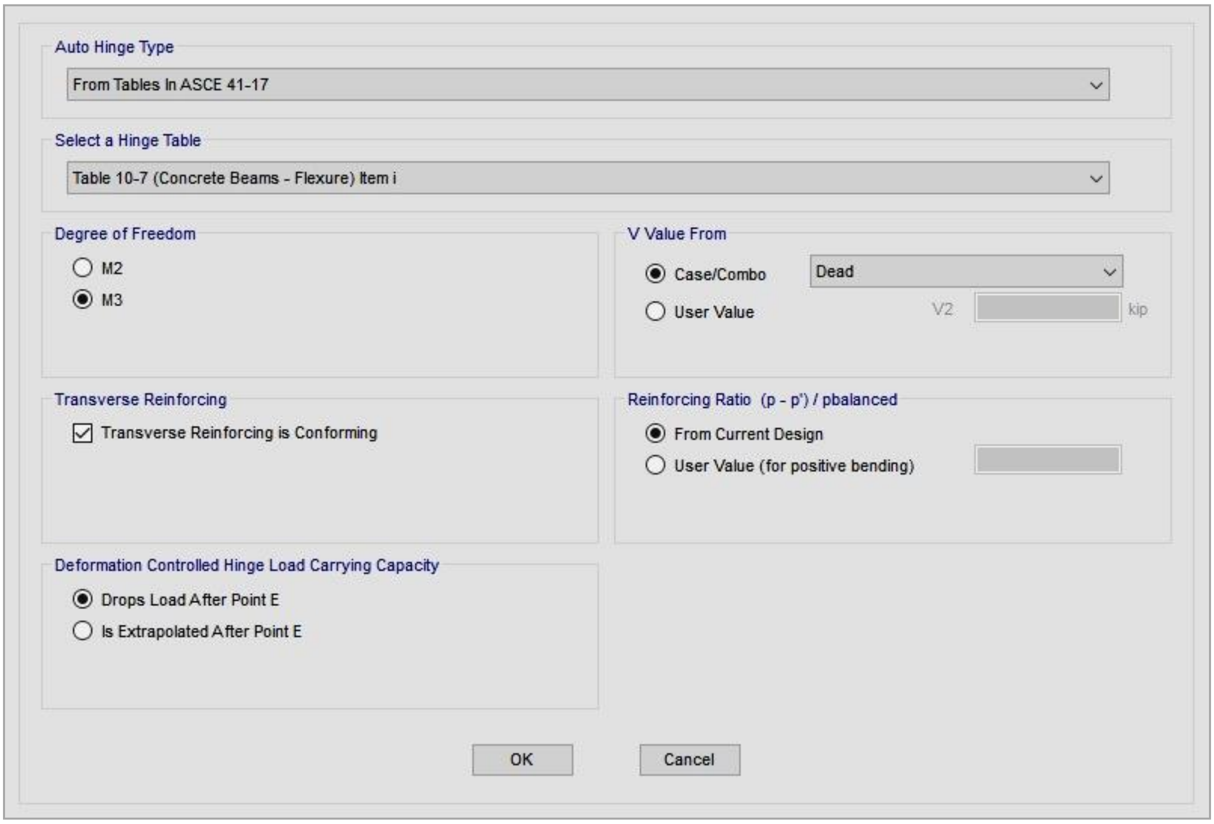
→ Then select the "Auto" option from the hinge property dropdown menu and add it at the 0 and 1 relative distance to apply plastic hinges on both ends of frame elements.

→ Click on the "Modify/Show Auto Hinge Assignment Data". The following window will appear.



→ Here, in the "Auto Hinge Type" select the "From Tables in ASCE 41-17" option.

→ Under the "Select a Hinge Table" select the "Table 10-7 (Concrete beams - Flexure) item i" option. The following window will appear.



→ Select the M3 option under "Degree of Freedom" option.

→ The shear value (V) can be automatically taken either from the analysis results of a particular load case/combination or a user-defined value can be provided based on manual calculations.

→ Similarly, the factor $\frac{\rho-\rho'}{\rho_{bal}}$ can either be automatically obtained by the program using the "From Current Design" option or it can be manually entered for positive bending.

→ For the transverse reinforcement, the check box can be checked if it is conforming.

→ Under the "Deformation Controlled Hinge Carrying Capacity", the option for "Drops Load After Point E" can be selected.

→ Click "OK" and the auto hinges will be assigned to the selected frame elements.

→ Note that in the auto hinge option, the hysteresis behavior cannot be selected and by default, the isotropic hysteresis behavior is assigned to all automatically generated hinges. Similarly, the user will not have control over the post-yield stiffness used in the backbone curve and other important hinge properties.

→ All the automatically generated hinges can now be seen at Define > Section Properties > Frame/Wall Nonlinear Hinges.

→ Click on the check boxes "Show Hinge Details" & "Show Generated Properties". The form will be populated will all the manually defined and automatically generated hinges.

→ In this window, the basic detailed of each generated M3 hinge can be seen. This include their behavior (Deformation Controlled) and whether it is automatically generated or manually defined.

→ Click on any automatically generated M3 hinge and Click "Modify/Show Property". The following window will appear.

→ Click on the "Modify/Show Hinge Property" button to check the properties of this automatically generated M3 hinge.

Chapter 5

Nonlinear Modeling of Shear Walls using Fiber Modeling Approach

In Chapter 3, the interacting P-M2-M3 fiber hinges were used to model the biaxially loaded columns. The inelastic behavior was lumped at both the ends of columns by using a finite plastic hinge length.

For shear walls, an adequate approach would be to model both membrane and bending behaviors as nonlinear. In such case, the fibers of shear walls should be made in both the directions (similar to columns). However, this approach is complex, time consuming and may not be practical. For the sake of simplicity, the out-of-place actions in shear walls can be assumed to remain elastic and the inelastic action is generally considered only in one direction (i.e., the in-place direction). With this in mind, a practical shear wall model is presented. The interaction of moment (in only one direction) with the axial loading is modeled using the interacting P-M3 fiber hinges. Moreover, in this example, the fiber hinges will be defined for the full length of shear walls since the inelastic action can occur at any location along the height of building. Such a model may be suitable for taller shear walls where column-like behavior governs.

Where the actions are assumed to remain linear, the cracked stiffness modifiers should be applied to account for cracking. Generally, the property modifiers for the shear walls are “0.7” for the bending in m11, m22 and m12 directions (plate bending).

5.1. Definition of Nonlinear Stress-strain Curves (for Material Fibers)

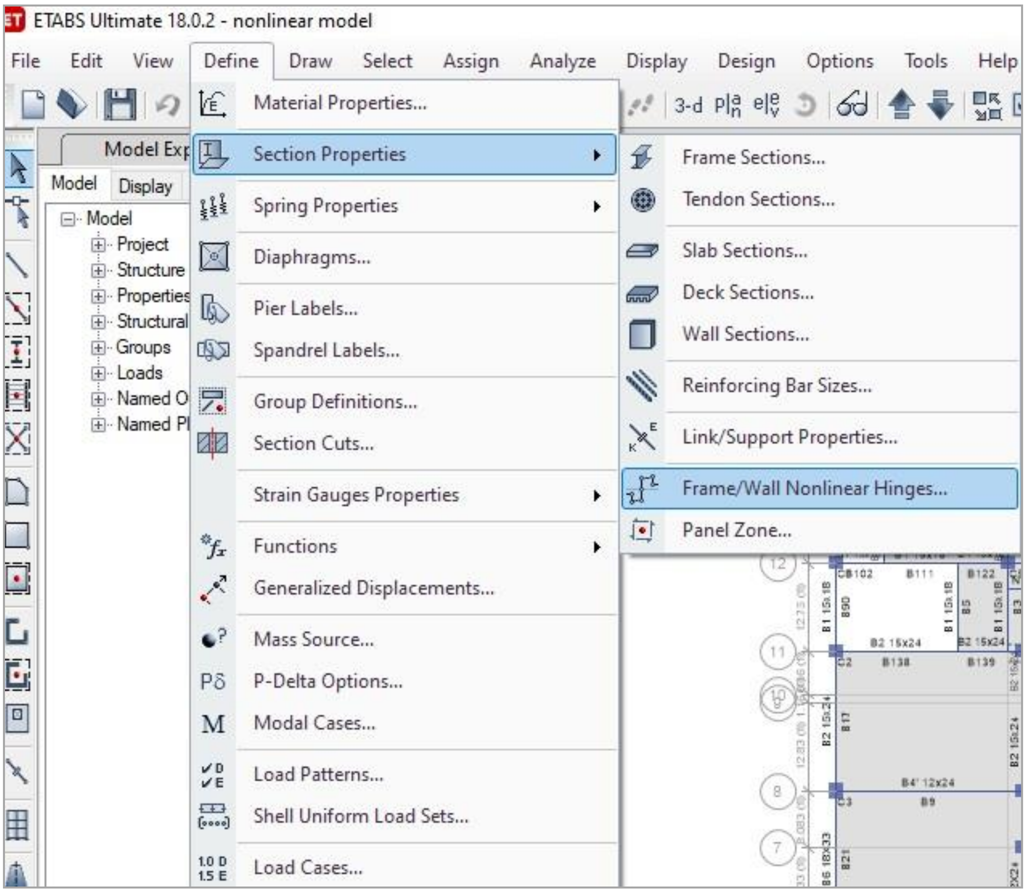
Before defining the P-M3 fiber hinges for shear wall sections, the first step is to define the nonlinear stress-strain curves for all materials. These curves will be assigned to individual material fibers while defining the P-M3 fiber hinges.

The procedure given in Section 3.1 can be followed to define the nonlinear stress-strain curves of all materials used in the shear wall sections.

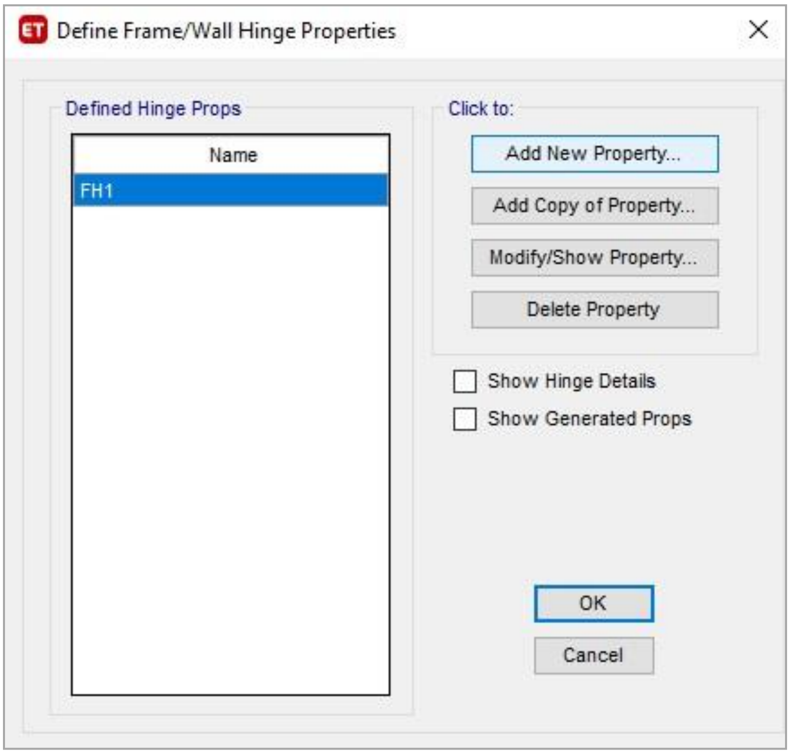
5.2. Manual Definition of P-M3 Fiber Hinges

5.2.1. Step 1: Defining the Hinge Properties

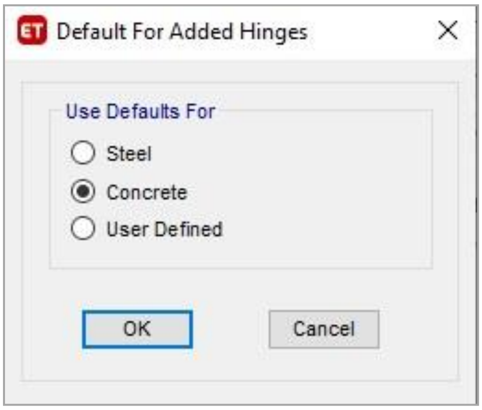
→ To define a P-M3 hinge, Click Define > Section properties > Frame/Wall Nonlinear Hinges.



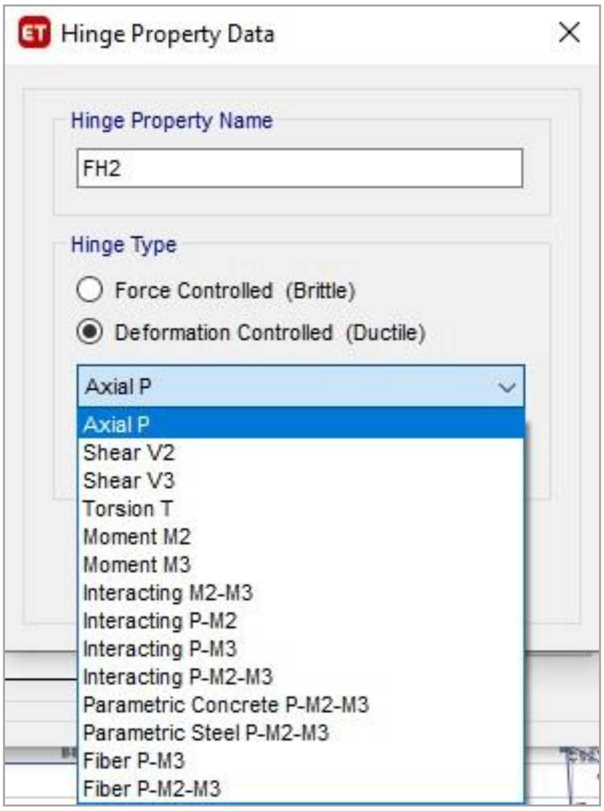
→ In the following window, click the "Add New Property" to define the properties of a hinge.



→ The following form will present the options available for the type of material. For the RC columns, select "Concrete".

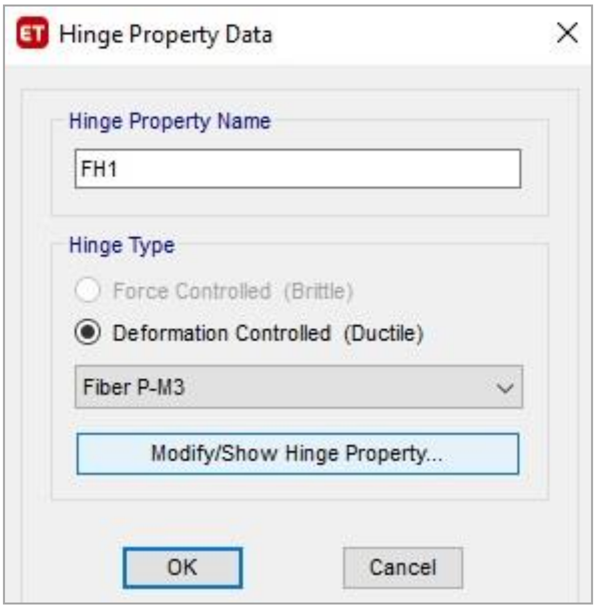


→ Click OK. The following form is used to select the name and basic type of hinge. You can select either a force-controlled (brittle) or deformation-controlled (ductile) hinge.

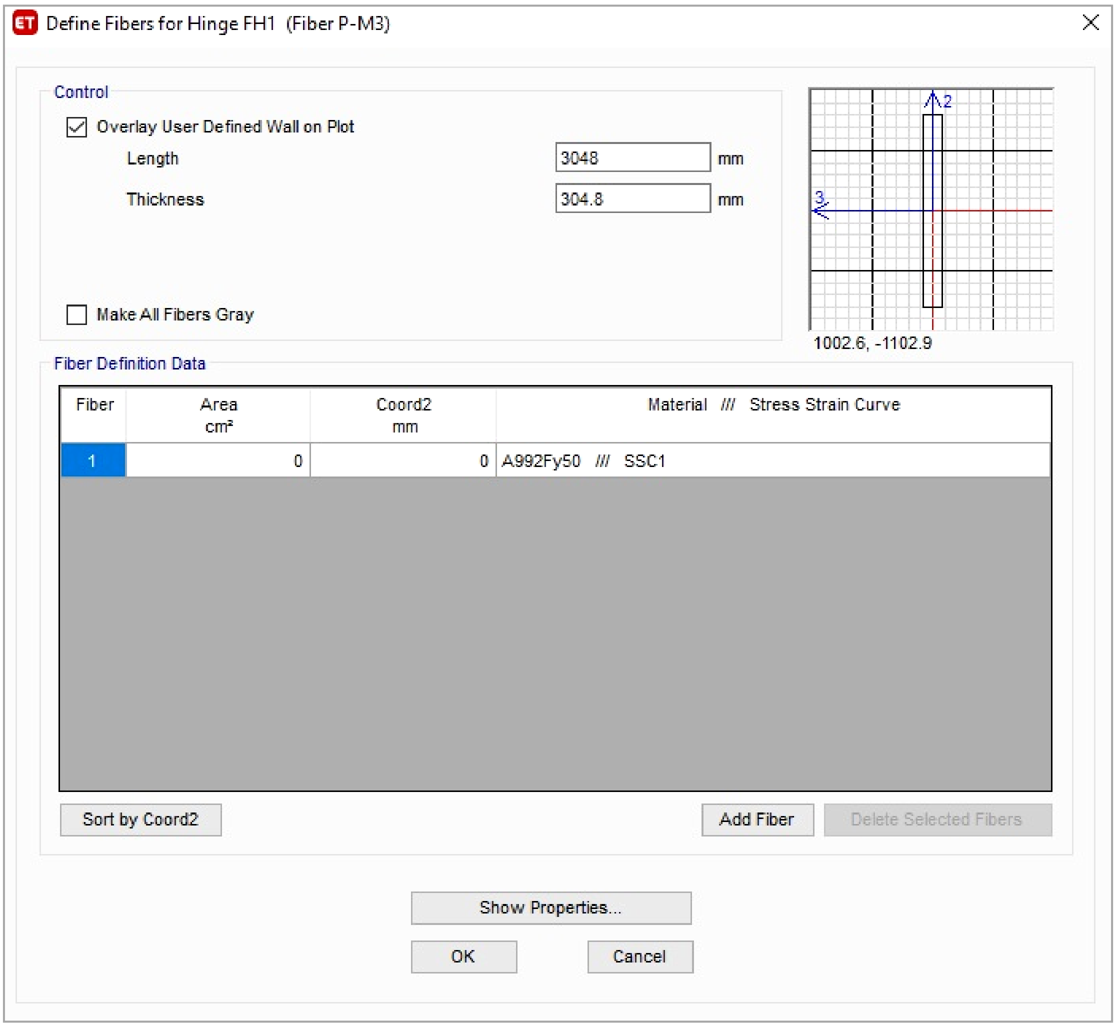


→ Select the "Deformation Controlled (Ductile)" option.

→ Select the fiber P-M3 hinge for the shear wall legs, click on the "Modify/Show Hinge Property".



→ A following window will appear.



→ In this window, for a particular leg of shear wall, all concrete and steel fibers should be added manually. The following information is added to this form.

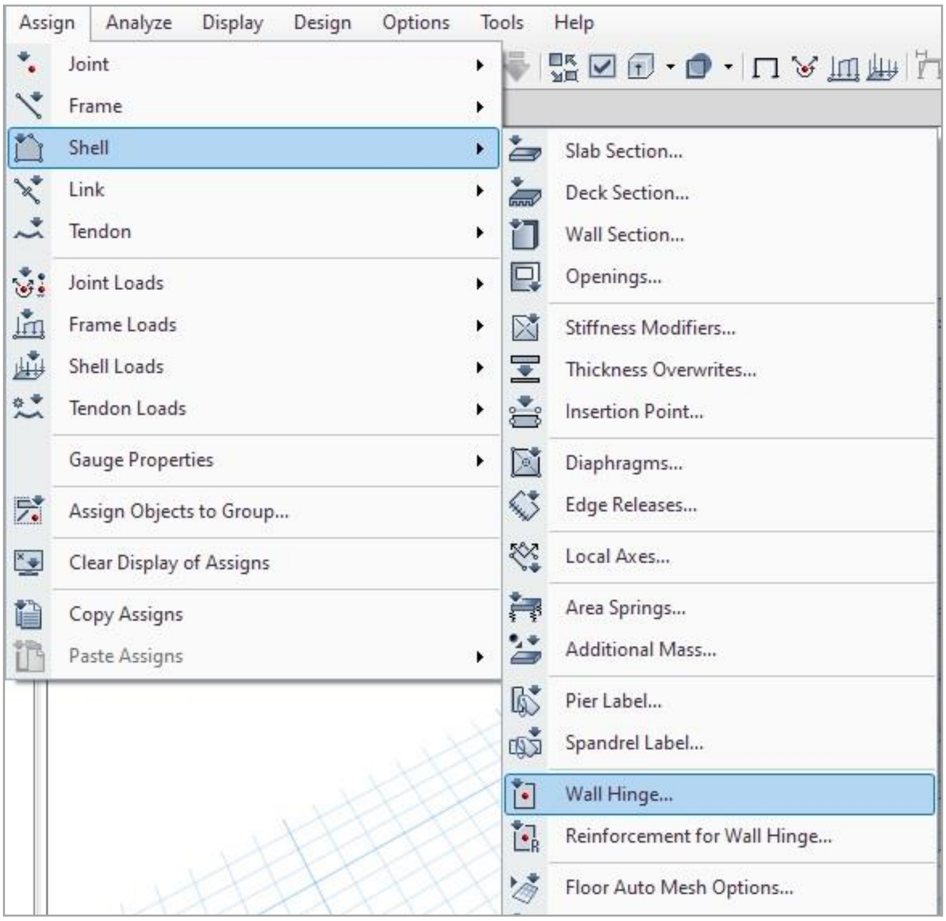
- The length and thickness of that particular shear wall leg is defined.
- Each concrete or steel fiber is added by providing the following three parameters
  - The area of fiber.
  - The coordinate 2 of fiber for its location in the cross-section (since the fibers are only in one direction).
  - The material (stress-strain curve) of the fiber (i.e., steel or concrete).

→ Once all the fibers for a particular shear wall leg are added to the list, Click OK. This will complete the manual definition of a P-M3 fiber hinge.

5.2.2. Assigning M3 Plastic Hinges to RC Beams

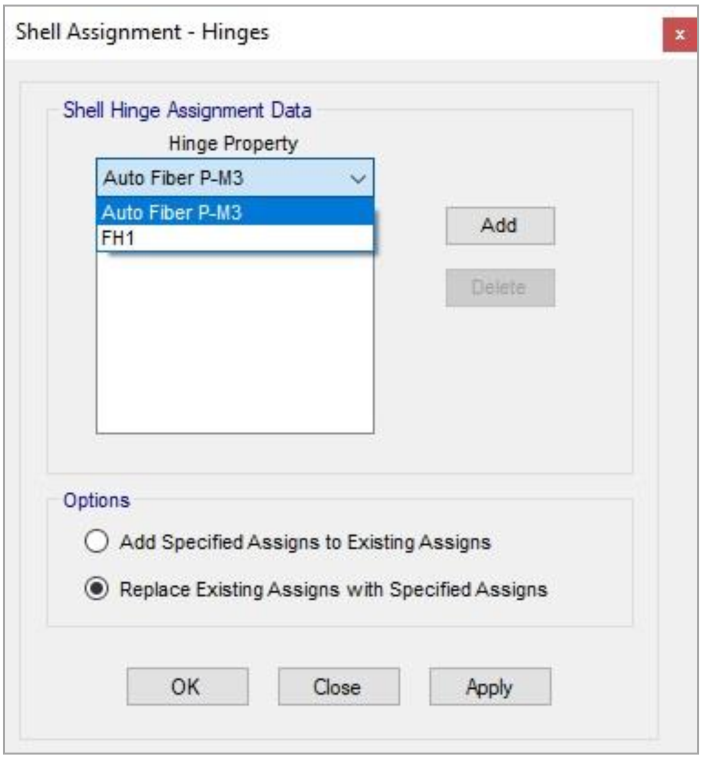
→ To assign the P-M3 hinges, select a particular shear wall leg (to which the fiber hinge is to be assigned).

→ Go to Assign > Shell > Wall Hinge.



→ The following window will appear.

→ Under the "Hinge Property" dropdown menu, select the previously defined P-M3 hinge and click "OK" option to assign this hinge to selected shear wall panel.



→ Under the "Hinge Property" dropdown menu, select the previously defined P-M3 hinge and click "OK" option to assign this hinge to selected shear wall panel.

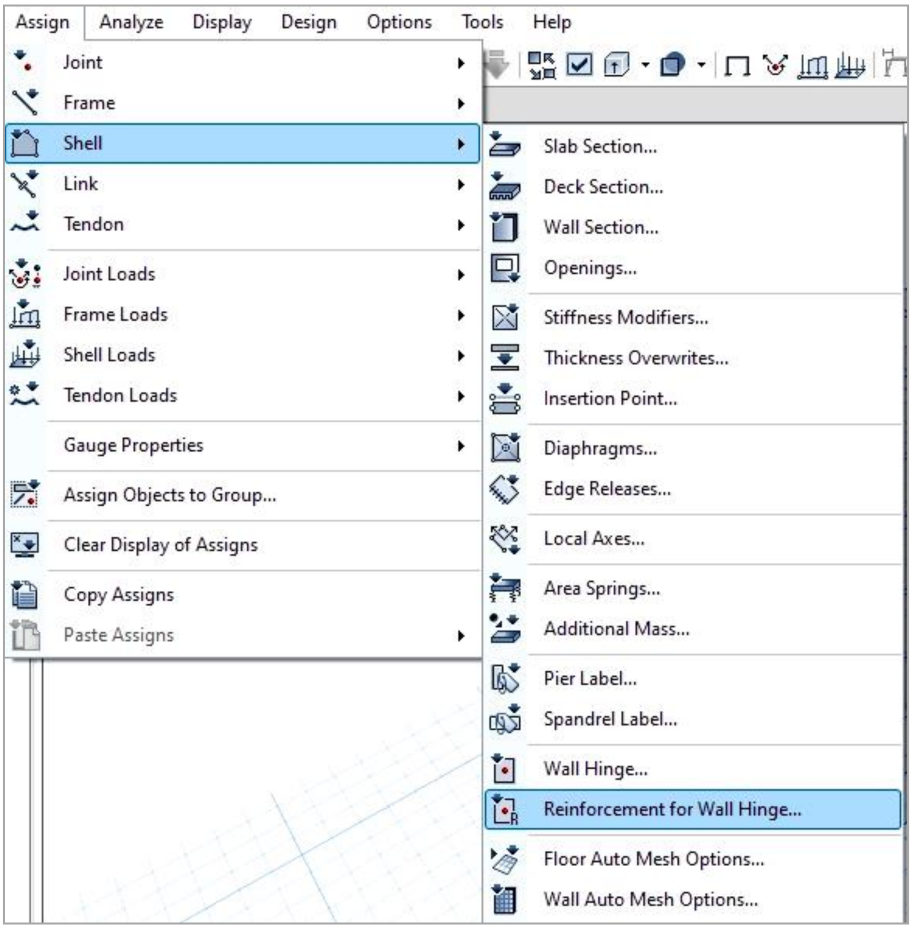
→ The process can be repeated for all shear wall legs to manually define and assign the P-M3 fiber hinges.

5.3. Automated Definition of P-M3 Fiber Hinges

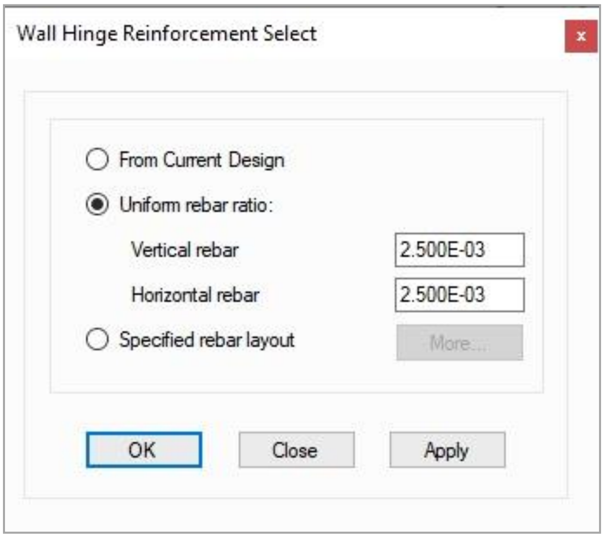
5.3.1. Step 1: Defining the Shear Wall Reinforcements

Before using the automatic definition of P-M3 fiber hinges for shear wall sections, the first step is to define the actual design reinforcement for each wall leg. The program will use this reinforcement to determine the properties of P-M3 fiber hinges. The following procedure can be followed.

→ Select the shear wall legs with same amount of reinforcement and Click on Assign > Shell > Reinforcement for Wall Hinges.



→ The following window will appear.



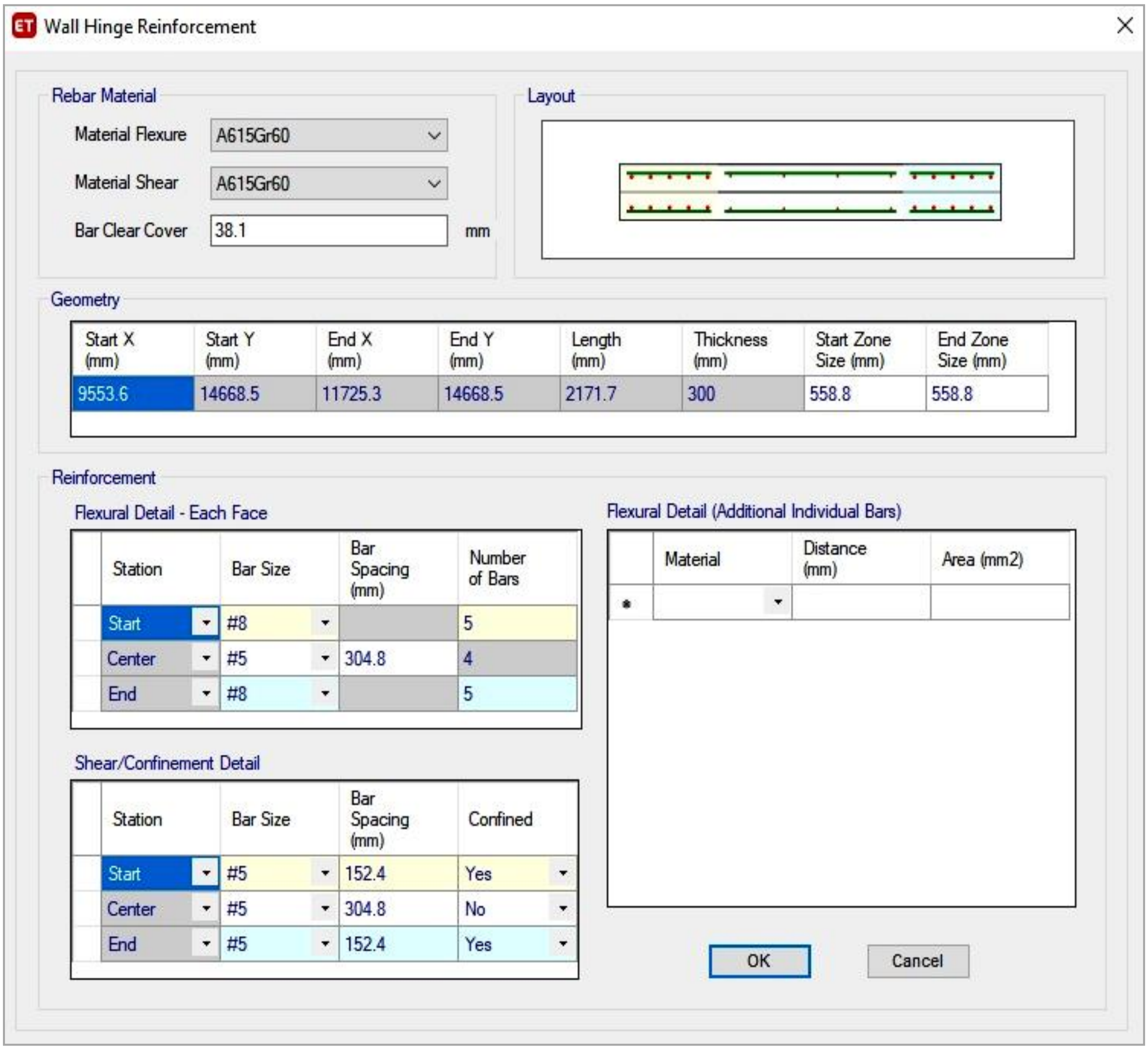
→ This window presents three available options for defining the reinforcement in selected shear walls.

- Option 1: “From Current Design” → ETABS will take reinforcement automatically from the shear wall design output.
- Option 2: “Uniform rebar ratio” → The user will define a uniform reinforcement (constant spacing throughout the height and breadth of shear wall leg).

- Option 3: "Specified rebar layout" → The user will define the specified reinforcement amount and its layout.

→ If the spacing of shear wall design reinforcement is not uniform (which is the most common case), the Option 3 should be selected. Click "Specified rebar layout" and then click "More" to proceed further.

→ The following window will open.



→ In this form, the detailed layout of the shear wall design reinforcement can be defined.

→ Under the "Rebar Material", specify the steel material and clear cover of bars.

→ Under the "Geometry", the location (in X & Y coordinates) of the selected shear wall leg is shown.

→ Enter the amount of flexural reinforcement in the panel by specifying the bar size, bar spacing & number of bars on each face.

→ Similarly, specify the amount of shear reinforcement by specifying the bar size, bar spacing and whether bars are confined or non-confined.

→ This reinforcement layout can also be seen under the “layout”.

→ Additional flexural individual bars can also be provided in panels by specifying their distance, area and material.

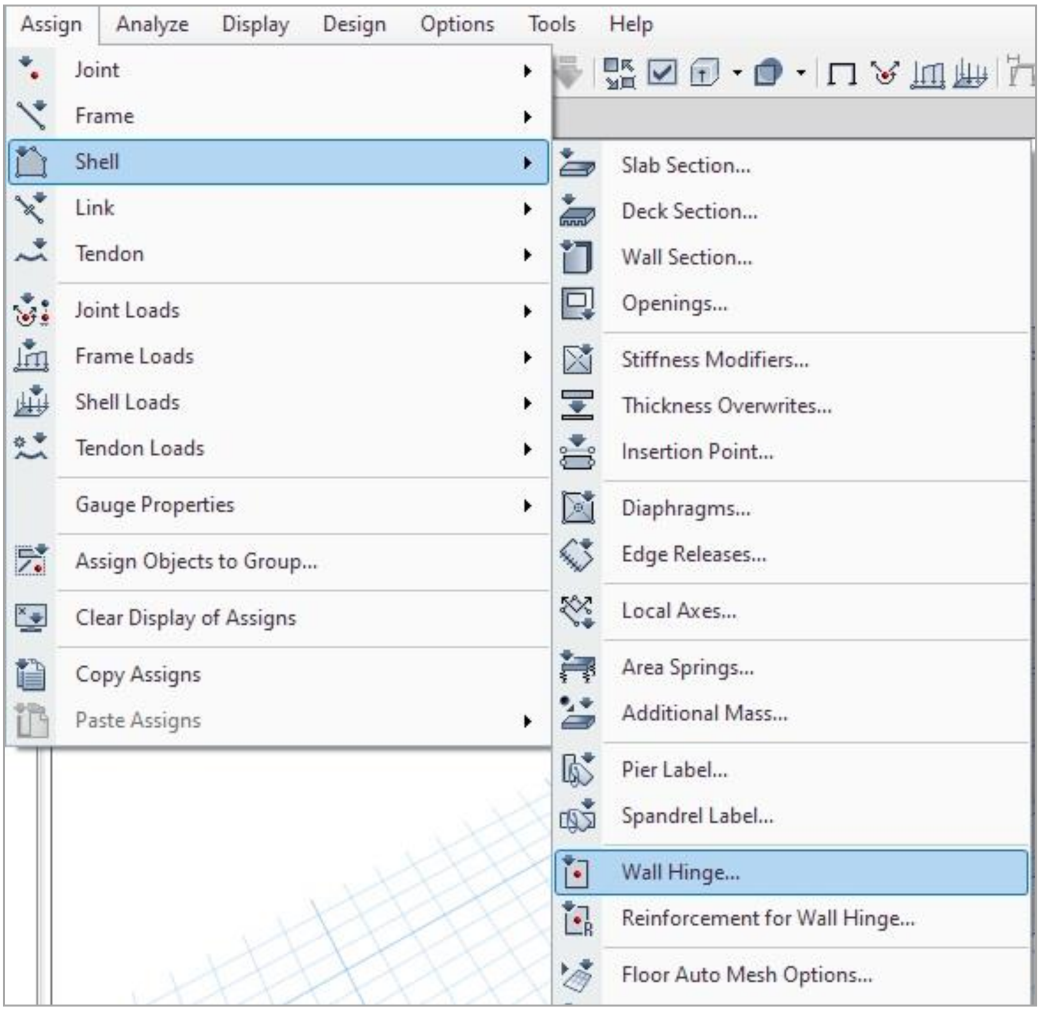
→ Click “OK” to complete the definition of reinforcement for the selected shear wall leg. This reinforcement will now be used by the program to automatically determine the properties of P-M3 fiber hinges.

→ The process should be repeated for all shear wall legs (having different amount and layout of reinforcement).

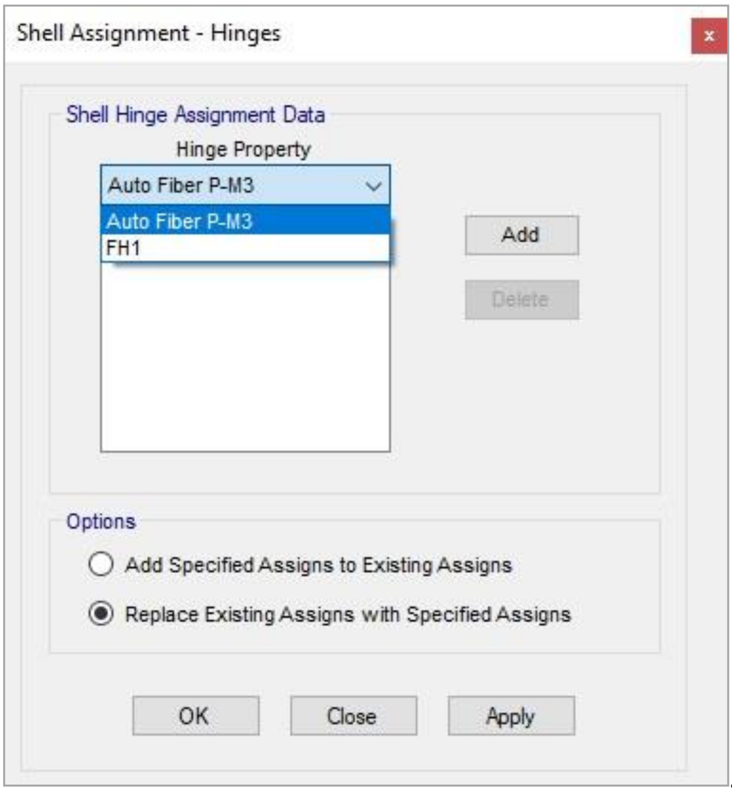
5.3.2. Step 2: Assigning the Hinges to Shear Walls

→ Select the Shear Walls.

→ Click the Assign > Shells > Wall Hinge.



→ The following window will appear.



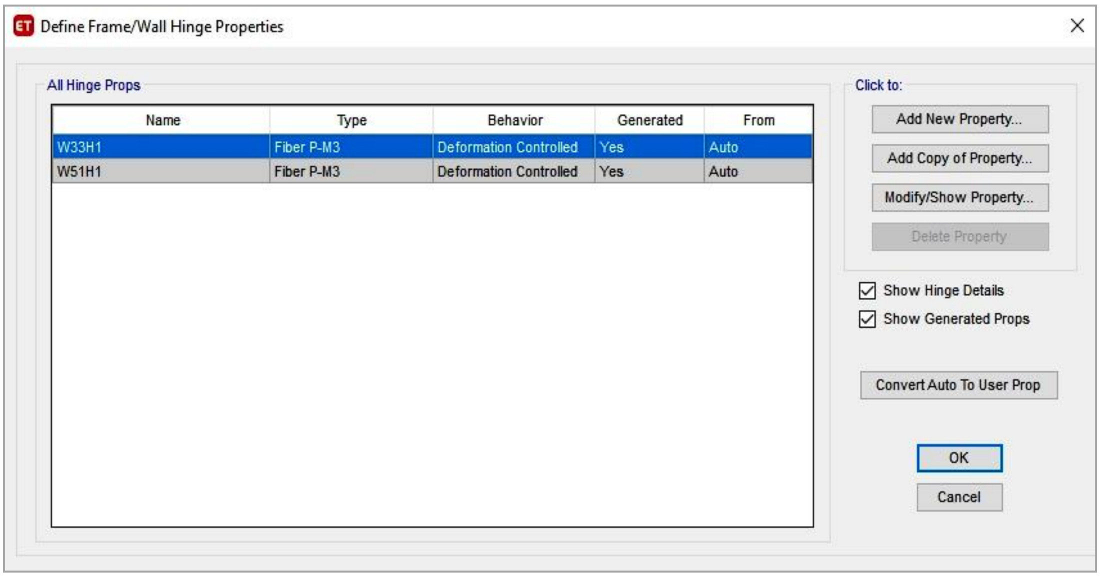
→ Here, in the "Hinge Property" dropdown menu, select the "Auto Fiber P-M3" and click "OK".

→ The program will automatically generate the P-M3 fiber hinges for the selected shear walls. The concrete and steel fibers are defined automatically in these generated fiber hinges.

→ The process can be repeated for all shear walls. In fact, once the reinforcement is defined separately for all legs, the hinge assignment can be performed by selecting all shear walls together.

→ All the automatically generated hinges can now be seen at Define > Section Properties > Frame/Wall Nonlinear Hinges.

→ The following window will appear.



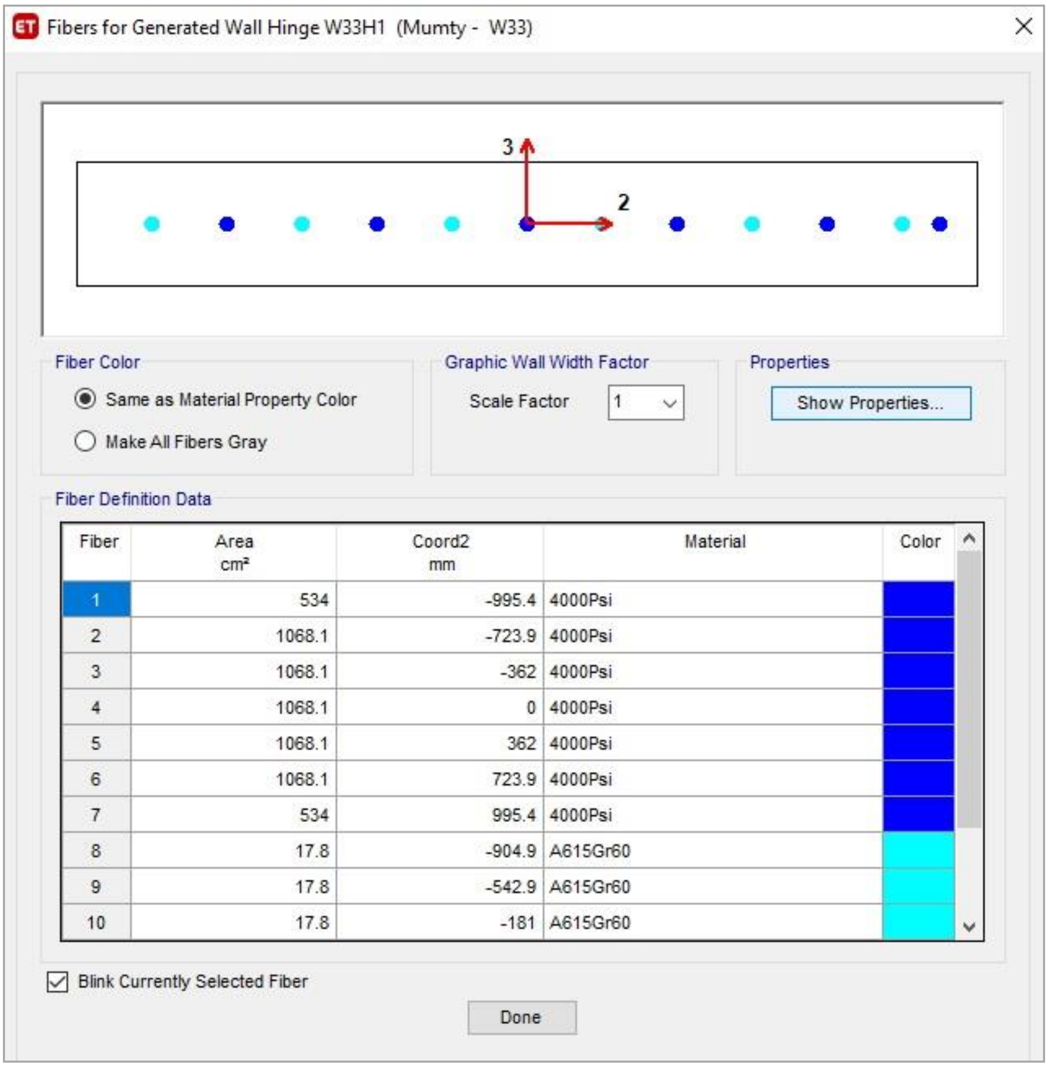
→ Click on the check boxes “Show Hinge Details” & “Show Generated Properties”. The form will be populated will all the manually defined and automatically generated hinges.

→ In this window, the basic detailed of each generated “Fiber P-M3” hinge can be seen. This include their behavior (Deformation Controlled) and whether it is automatically generated or manually defined. In case of generated hinges, the master hinge is also shown in the “From” column.

→ Click on any automatically generated fiber hinge and Click "Modify/Show Property". The following window will appear.

→ Click on the "Modify/Show Hinge Property" button to check the properties of this automatically generated hinge.

→ Click on the "Define/Show Fibers" option. The following form will open.



→ It can be seen that the list of concrete and steel fibers is automatically populated for this particular column cross-section. The coordinates (locations), areas and material stress-strain curves are also assigned to each fiber. Generally, for each reinforcing bar in the cross-section, one steel fiber is defined. If the total areas of all steel fibers area are summed, it would be equal to the actual longitudinal reinforcement area of this particular shear wall leg. Similarly, the total area of all concrete and steel fibers area should be equal to the cross-sectional area of this particular shear wall leg.

Chapter 6

Nonlinear Seismic Analysis Procedures

The discussion is concluded with few brief comments on the nonlinear static and dynamic analysis procedures.

5.1 The Evolution of Seismic Design Philosophy over Past Few Decades

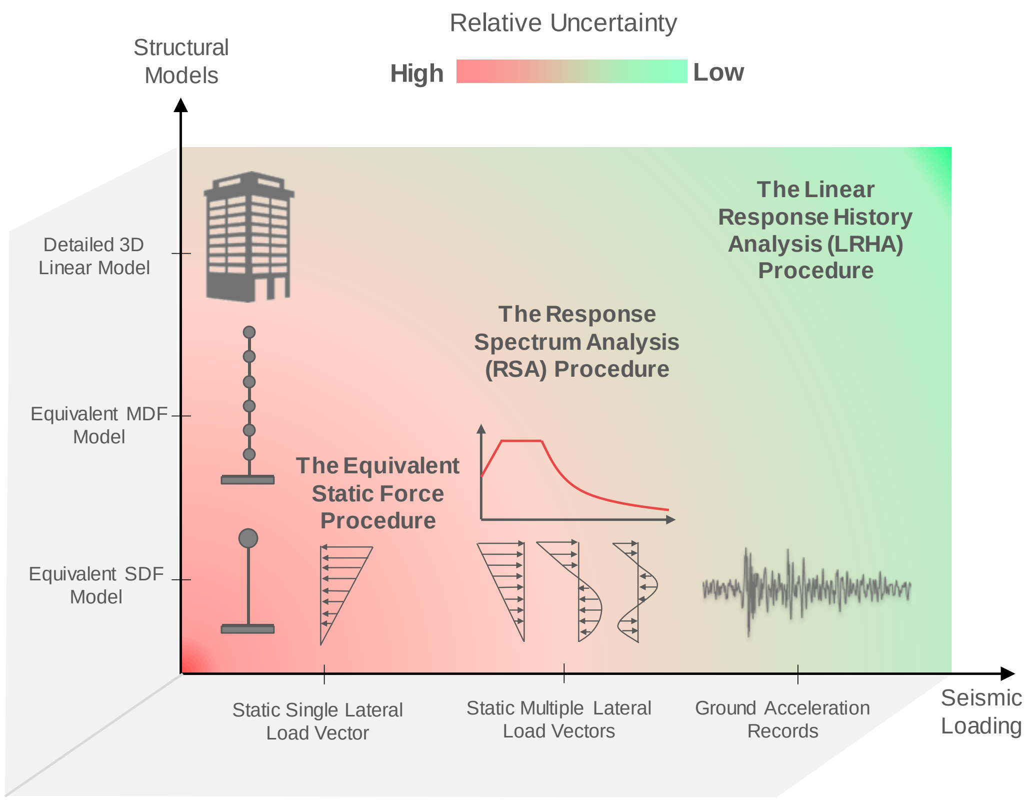
The analysis procedures in early 20th century were essentially based on the use of simplified structural models subjected to simplified loading types. For example, for the purpose of structural design, the seismic load was historically idealized as a simple mass-proportional lateral static loading. Later, with the increasing applications of modal analysis and the formulation of the response spectrum analysis (RSA) procedure, the role of vibration modes and natural periods in understanding and controlling the seismic demands was recognized. With the advent of computer programs and dynamic analysis solvers in mid-1960s and 1970s, and with the increasing availability of more ground motion records, the use of detailed dynamic analysis procedures based on the direct integration solution of the governing dynamic equations of motion was established. This also started the use of nonlinear modeling for a relatively better structural idealization as compared to the linear elastic models.

In late 1980s and 1990s, the importance of nonlinear modeling and analysis increased significantly with the emergence of performance-based seismic engineering (PBSE) as a well-accepted methodology for the seismic evaluation and design of building structures (ATC 40, 1996). This methodology uses the predicted structural performance to equip the decision-makers with the key information regarding structural safety and risk. The performance is primarily characterized in terms of expected damage to various structural and nonstructural components and building contents. Since the structural damage implies an inelastic behavior, the traditional design and analysis procedures that are based on linear elastic behavior can only implicitly predict the performance. By contrast, the objective of nonlinear seismic analysis procedures is to directly estimate the magnitude of inelastic seismic demands.

The ***performance-based design (PBD)*** approach is a recent shift in our understanding of structural design. It provides a systematic and flexible methodology for assessing the structural performance of a building, system or any component, as opposed to the cookbook type design methods prescribed in building codes. The building code design procedures are intended to result in buildings capable of providing certain levels of performance, however, the actual performance of individual building design cannot be assessed as part of the traditional code design process. The performance-based evaluation explicitly evaluates how a building is likely to perform; given the potential hazard it is likely to experience, considering uncertainties inherent in the quantification of potential hazard and uncertainties in assessment of the actual building response. This methodology explicitly evaluates the response of the buildings under the potential seismic hazard while considering different probable site-specific seismic demand levels (Service Level Earthquake (SLE) and Maximum Considered Earthquake (MCE)). For this purpose, various state-of-the-art nonlinear analysis procedures and latest computer modeling tools are used to accurately determine the nonlinear seismic demands of whole structure and its individual components.

The generic procedure for the determination of nonlinear seismic demands involves a number of key steps. The engineer is first required to set up a computer model which is expected to mimic the behavior of the actual structure. The anticipated seismic shaking is characterized after a seismic hazard analysis while accounting for various site-specific phenomenon. A suitable analysis procedure is then applied to the structural model considering all important loading scenarios. This results in the predictions of engineering demand parameters (EDPs) which can be subsequently compared with an acceptance criteria (generally prescribed by the seismic evaluation guidelines) to determine the seismic performance of the building. The EDPs normally comprise of global displacements (e.g. at roof or at any other reference point), inter-story drifts, story forces, component distortions, and component forces (FEMA 440, 2005). The level of complexity in this overall process may vary widely depending upon the choice of modeling scheme and analysis procedure, as well as the required degree of accuracy.

Although the seismic evaluation guidelines allow the use of approximate analysis procedures for conventional low- to mid-rise structures, the detailed NLRHA procedure is still recommended as a final check especially for the structures having extraordinary importance or those with special features.



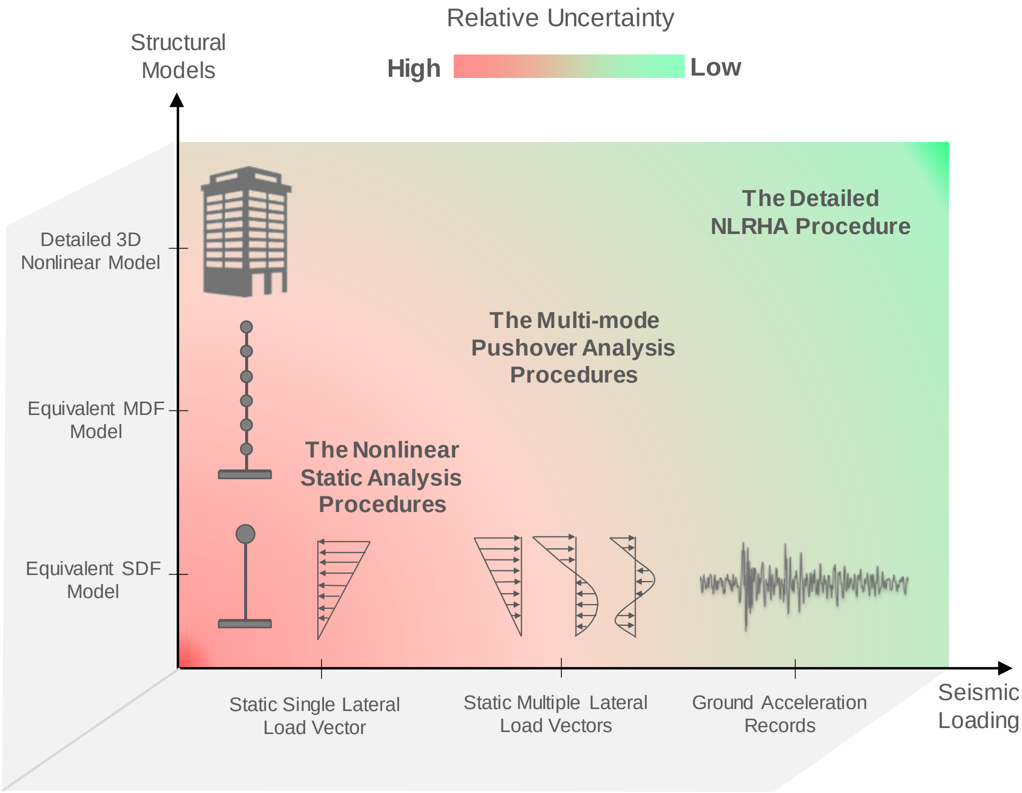


Figure 5-1: The relative modeling complexities and uncertainties of major linear and nonlinear seismic analysis procedures.

5.2 Prescriptive vs. Performance-based Seismic Design – A New Front

The performance-based seismic design and assessment methodology is a relatively recent shift in our understanding of structures. It is an approach in which structural design criteria are expressed in terms of achieving a set of performance objectives or levels (and not in terms of passing some code-prescribed checks). This approach explicitly links the structural performance with earthquake hazard and ensures that the structure reaches specified demand level in both service and strength design levels.

The Essence of PBD

A “decision-maker” states a desire that a building be able to “perform” in a certain way, e.g. protect life safety, minimize potential repair costs, minimize disruption of use, etc.

The “engineer” uses his or her skill to provide a design that will be capable of achieving these objectives.

The primary motivation for performance-based seismic design originates from the non-suitability of traditional building codes to design high-rise buildings with new structural systems and innovative shapes and materials.

The design codes doesn't specify any explicit verification of the building performance. They can't answer about what level of structural damage is expected in case of future ground shakings of different levels. On the other hand, the building owners, residents and public at large see the process of structural design and construction with a different view. In 1980s and 1990s (in the prevailing era of building codes), the building owners in US began to question how their buildings would perform in future earthquakes and demand that engineers should design (or upgrade) their buildings to perform better. The building owners and public does not care about the code, or theories or procedures. All they care about is "building safety" and "structural performance". They usually express their desires in terms of a series of performance objectives. These objectives can be certain levels of safety and damage, operational discontinuity, repair cost or any other indicators. The performance-based design and assessment methodology emerged as a response to all these needs. This methodology requires the designer to assess how a building is likely to perform under extreme events. A correct application of this methodology helps to identify unsafe designs. It enables arbitrary restrictions to be lifted and provides scope for the development of innovative, safer and more cost-effective solutions.

Are All Buildings Codes Correct ?

- Building codes differ in seismic analysis and design philosophies.
- If they differ, can all of them be correct ?
- Did we inform the structures to follow which code when an earthquake or thunderstorm strikes?
- Codes change every 3 or 5 years, should we upgrade our structures every 3 or 5 years to conform?
- Codes intend for "Life Safety", not damage limits or cost implications.

Prescriptive Building Codes – A Shelter

Public: Will the building be safe?

Owner: Will the building collapse/ will it be damaged ?

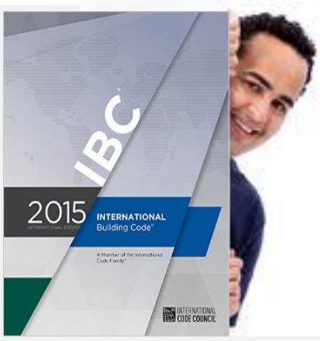
Can I use the building after a given earthquake?

How much will repair cost?

How long will it take to repair?

Can I make building that will not be damaged and will not collapse?

Structural Engineer: Not sure, but I did follow the "Code"



- As long as engineers follow the code, they can be sheltered by its provisions.
- The building codes "implicitly" ensures that the performance of structure will be acceptable if its rules are followed.
- The performance may not be acceptable in certain cases.

So, we end up in changing the rules every three years, or invent new rules.

The performance-based seismic design and assessment methodology is developed in response to the cook book-type prescriptive building codes. This methodology encourages the designer to go beyond the code to determine the actual inelastic response of the structure under several levels of anticipated ground motions. Therefore, this methodology cannot be simply coded under a traditional prescriptive framework of building codes. However, the code provisions can be systematically made more "performance-oriented" by employing some of the concepts considered in this methodology. Several attempts are being made to bring some elements of performance-based design (e.g., the probabilistic framework to define different levels of seismic hazard etc.) in the traditional codes.

5.3 Linear Time History Analysis (LTHA) Procedure

The time history analysis (also referred to as the response history analysis) is the rigorous dynamic analysis procedure. In this procedure, the complete governing dynamic equation of motion is solved for the structure subjected to a ground motion. Beside the restoring lateral stiffness effects, the inertial and damping effects of the structure's mass and energy dissipation respectively, are also included in this analysis procedure. From structure's side, the primary inputs include the definition of detailed geometry, its mass and the amount of energy dissipation (often represented by damping coefficients). From loading's side, the required input includes a ground motion shaking function (often represented by a plot between horizontal ground acceleration and time) which represents the future earthquake shaking. Since the prediction of an actual future earthquake is not possible, we are left with the following three options to develop this ground acceleration function (or time history) for this detailed dynamic analysis procedure.

a) Select the data of past earthquakes recorded on sites with similar properties (soil type etc.) as the structure's site. These selected past earthquakes should be produced by the similar seismic sources (and in similar seismo-tectonic environment) as the actual site of the structure. The source-to-site distance, faulting mechanism and other hazard governing parameters of the selected ground motions should match with those of the actual site of the structure. The as-recorded hazard level and other characteristics of previously recoded ground motions never exactly match with those of a future earthquake expected at a particular site. Therefore, before using in the time history analysis, these selected ground motions are also modified in accordance with the hazard level (defined by the response spectrum curve) of that site.

b) Use Green's functions and other techniques to develop synthetic time histories. These are the artificial (hypothetical) ground acceleration functions which can be used to perform the dynamic analysis of the structure. This approach, although nowadays available in several structural analysis software, is not popular among the structural engineering community.

c) Physics-based predictions of time histories. This technique includes simulating the actual rupture of the governing seismic source and the propagation of seismic waves to the site. This approach is very rare and is not yet practical for general structural engineering applications.

The structural model required for time history analysis procedure is generally more sophisticated compared to the equivalent static force procedure and the response spectrum analysis (RSA) procedure. Apart from lateral stiffness effects, it should also be able to represent and capture the inertial and damping effects of the structure. Similarly, the selection of seismic loading (time histories) also requires more expertise and skills in the areas of engineering seismology and hazard assessment. Moreover, the latest analysis guidelines require to use a large number of ground motions records representing the anticipated seismic hazard at the building site. This process of selecting representative ground motions, performing the detailed dynamic analysis and post-processing of results may cost a significant amount of time. An ordinary design office may not have necessary expertise and resources to undergo this complete process for each project. For most practical cases, the equivalent static force procedure or the RSA procedure may serve the purpose of estimating design demands within their required degree of accuracy. Due to these reasons, until recently, the time history analysis is not commonly used for the purpose of structural design, although several new codes and studies are now emphasizing on its use instead of the RSA procedure.

The UBC 97 (and BCP-2007) prescribes that the time history analysis should be performed with pairs of appropriate horizontal ground-motion time history components that are selected and scaled from not less than three recorded earthquake events. Appropriate time histories should have magnitudes, fault distances and source mechanisms that are consistent with those that control the design-basis earthquake (or maximum capable earthquake) at building's site. Where three appropriate recorded ground-motion time-history pairs are not available, appropriate simulated ground-motion time-history pairs may be used to make up the total number required. The following guidelines are also prescribed for the modification of selected ground motion records.

a) For each pair of horizontal ground motion components, the square root of the sum of the squares (SRSS) of the 5 percent-damped site-specific spectrum of the scaled horizontal components shall be constructed.

b) The ground motion records shall be scaled such that the average value of their SRSS spectra does not fall below 1.4 times the 5 percent-damped code-prescribed or site-specific spectrum of the design-basis earthquake for periods from 0.2T second to 1.5T seconds (Where T is the fundamental natural period of the structure in the direction under consideration). The code-prescribed or the site-specific spectrum can also be referred to as the "target spectrum" in this case.

The latest strong motion recording stations are capable of simultaneously recording 3 components (two horizontal components 90^0 apart and one vertical component) of each ground motion. The time history analysis is generally performed only in two horizontal directions. In some special cases (e.g., for structures with significant irregularities or with special features e.g., the base isolation mechanism etc.), the vertical time history analysis can also be performed. In equivalent static force procedure and the RSA procedure, the effect of vertical ground motions can simply be added linearly to the gravity loads (using amplification factors dependent on that seismic hazard parameters) in design load combinations.

The BCP-2007 prescribes that the record pair of time histories (representative of the actual future earthquake motions) shall be applied simultaneously to the model considering torsional effects. The parameter of interest shall be calculated for each time history analysis. For the purpose of structural design, all response modifications prescribed for the RSA analysis are also applicable to the linear time history analysis procedure. If three time histories analyses are performed, then the maximum response of the parameter of interest shall beused for design. If seven or more time-history analyses are performed, then the average value ofthe response parameter of interest may be used for design.

Moreover, for the structures(regular or irregular) located on soil profile type $S_F$and having a time period greater than 0.7 second, the analysis shall should also the effects of the soils at the site and should also conform to the following requirements.

a) Either the RSA procedure should be conducted using a site-specific response spectrum, or the linear time history analysis should be conducted using the ground motions developed and modified in accordance with site-specific response spectrum.

b) The possible amplification of building response due to the effects of soil-structure interaction and lengthening of building period caused by inelastic behavior should be considered in the analysis.

5.4 Nonlinear Time History Analysis (NLTHA) Procedure

Since nonlinear modeling of structures requires significant expertise and understanding of the true inelastic behavior of individual structural components, the nonlinear time history analysis procedure (NLTHA) is seldom used in ordinary design offices for conventional structural analysis. With a recent advent of the performance-based seismic design philosophy, the role of nonlinear response history analysis is also included in the structural design practice. Conventionally, this procedure is only used for detailed design review or settlement of design disputes etc.

The UBC 97 (and BCP-2007) allows the use of alternative lateral-force procedures (instead of those discussed in earlier sections) using rational analyses based on well-established principles of mechanics. The use of nonlinear time history analysis (NLTHA) also falls in this category and therefore should meet the requirements set by the code for the selection of time histories. The results of nonlinear time history analysis are a true measure of the real behavior of the structure and need no modification with the response modification factor, displacement amplification factor or over-strength factor.

When nonlinear time-history analysis is used to justify a structural design, a design review of the lateral force resisting system should be performed by an independent engineering team, including persons licensed in the appropriate disciplines and experienced in seismic analysis methods. The lateral-force-resisting system design review shall include, but not be limited to, the following:

a) Reviewing the development of site-specific spectra and ground-motion time histories.

b) Reviewing the preliminary design of the lateral-force-resisting system.

c) Reviewing the final design of the lateral-force-resisting system and all supporting analyses.

Along with the plans and calculations, the engineer of record is also required to submit a confirmatory statement by all members of the engineering team involved in the review process.

Towards the end, a discussion about two important issues related to nonlinear analysis (taken from ATC – 72-1) is included as follows.

Energy Dissipation and Viscous Damping: Traditionally, viscous damping has been used as a convenient way to idealize energy dissipation in elastic response history analyses. In nonlinear response history analyses, it is important to identify the sources of energy dissipation, and to determine how these effects are represented in the analytical model. For components that are modeled with nonlinear elements, most of the energy dissipation will be modeled explicitly through hysteretic response. However, energy dissipation that is modeled at low deformations may vary significantly with the type of model used. For example, continuum finite element models for reinforced concrete tend to capture damping effects due to concrete cracking that is not captured in concentrated hinge models. In tall buildings, the relative contribution of damping from certain components can be substantially different from values typically assumed in low-rise buildings. For example, measured data show that levels of damping tend to be lower in tall buildings, suggesting that there may be proportionally less damping. Possible reasons for this include soil-foundation-structure interaction or special “isolation” detailing of nonstructural partitions and other components.

Gravity Load Effects in Nonlinear Analysis: Unlike linear analyses, nonlinear analyses are load path dependent, in which the results depend on the combined gravity and lateral load effects. For seismic performance assessment using nonlinear analysis, the gravity load applied in the analysis should be equal to the expected gravity load, which is different from factored gravity loads assumed in standard design checks. In general, the expected gravity load is equal to the unfactored dead load and some fraction of the design live load. The dead load should include the structure self-weight, architectural finishes (partitions, exterior wall, floor and ceiling finishes), and mechanical and electrical services and equipment. The live load should be reduced from the nominal design live load to reflect:(1) the low probability of the nominal live load occurring throughout the building; and (2) the low probability of the nominal live load and earthquake occurring simultaneously. Generally, the first of these two effects can be considered by applying a live load reduction multiplier of 0.4 and the second by applying a load factor of 0.5 (such as is applied to evaluation of other extreme events). The net result is a load factor of 0.4 x 0.5 = 0.2, which should be applied to the nominal live load. For example, in a residential occupancy with a nominal live load of 40 psf, an expected live load of 0.2 x 40 psf = 8 psf should be considered in the nonlinear analysis. Accordingly, a general load factor equation for gravity loads applied in nonlinear analysis is:

$$1.0\,D + 0.2\,L$$

where D is the nominal dead load, and L is the nominal live load. In the case of storage loads, only the 0.5 factor would apply, and the net load factor on storage live loads should be taken as 0.5. Expected gravity loads should also be used as the basis for establishing the seismic mass to be applied in the nonlinear analysis. Vertical gravity loads acting on the entire structure, not just the seismic force-resisting elements, should be included in the analysis in order to capture destabilizing P-Delta effects. Nonlinear analysis should include leaning columns with applied gravity loads that rely on the seismic force-resisting system for lateral stability.

References

ACI Committee 318 (2019) Building code requirements for structural concrete (ACI 318-08) and commentary. American Concrete Institute.

FEMA 356 (2000) commentary for the seismic rehabilitation of buildings (FEMA356), Washington, DC: Federal Emergency Management Agency 7.

BCP (2007) Building Code of Pakistan, Seismic Provision- 2007. Ministry of housing and works, Islamabad, Pakistan.

UBC (1997) Uniform Building Code International Conference of Building Officials, Whittier, California.

CSI Analysis Reference Manual (SAP 2000, ETABS and CSI Bridge), Computers and Structures Inc., USA.

Seismic Evaluation and Retrofit of Existing Buildings, ASCE/SEI 41-17 (formerly FEMA 356), American Society of Civil Engineers.

Graham H. Powell (2010) Modeling for Structural Analysis, Computers and Structures Inc., ISBN-10: 0923907882.

Modeling and Acceptance Criteria for Seismic Design and Analysis of Tall Buildings, PEER/ATC 72-1, Applied Technology Council and Pacific Earthquake Engineering Research Center, 2010.

An Alternative Procedure for Seismic Analysis and Design of Tall Buildings Located in the Los Angeles Region, 2017 Edition with 2018 Supplements, Los Angeles Tall Buildings Structural Design Council, March 20, 2018.

Experimental Evaluation and Analytical Modeling of ACI 318-05/08 Reinforced Concrete Coupling Beams Subjected to Reversed Cyclic Loading, UCLA-SGEL Report 2009/06

Design Recommendations for Steel-reinforced Concrete (SRC) Coupling Beams, UCLA-SGEL Report 2013/06.

Prepared by

Hamza Mazhar
Fawad Ahmed Najam
Luqman Ahmed
Hafiz Zain Akram

NUST Institute of Civil Engineering (NICE)
School of Civil and Environmental Engineering (SCEE)
National University of Sciences and Technology (NUST)
H-12 Islamabad, Pakistan, Cell No: 92-333-9912091