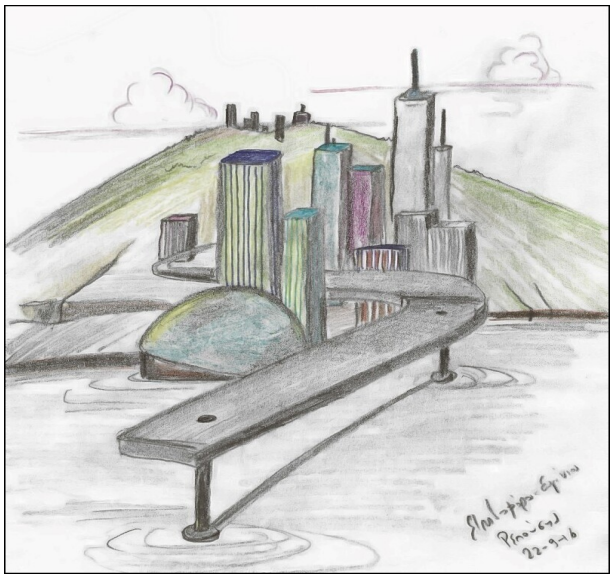
NATIONAL TECHNICAL UNIVERSITY OF ATHENS
SCHOOL OF CIVIL ENGINEERING
LABORATORY FOR EARTHQUAKE ENGINEERING
MSc in ANALYSIS AND DESIGN OF EARTHQUAKE RESISTANT STRUCTURES

POSTGRADUATE THESIS

Advanced Non Linear Analysis of Frame Structures under Seismic Loading



Nikiforos D. Repousis

SUPERVISOR :
Michail Fragiadakis
Associate Professor NTUA

ATHENS

March 2022

ACKNOWLEDGEMENTS

Completing my postgraduate studies, I would like to thank everybody who supported me through this journey.

Specifically, I would like to acknowledge the following who have helped me undertake this thesis:

- My supervisor, Associate Professor Michalis Fragiadakis, for the excellent cooperation and for providing guidance and feedback throughout this project.
- My family -Dimitris, Nikolena, Elpida, Sissy- that always supports my efforts.
- The foundation «Σπυρίδωνος Φ. Αντύπα Υπέρ της Κεφαλληνίας» for the financial support of my postgraduate studies.
- Mr. Nikos Constandopoulos for his support during my postgraduate studies.
- My cousin, Elpidoforos, for his valuable advices and for the illustration of the cover of the present thesis.
- My friends, my girlfriend and my fellow students for making this journey enjoyable.

Nikiforos D. Repousis

Athens, March 2022

ΠΕΡΙΛΗΨΗ

ΕΘΝΙΚΟ ΜΕΤΣΟΒΙΟ ΠΟΛΥΤΕΧΝΕΙΟ
ΣΧΟΛΗ ΠΟΛΙΤΙΚΩΝ ΜΗΧΑΝΙΚΩΝ
ΕΡΓΑΣΤΗΡΙΟ ΑΝΤΙΣΕΙΣΜΙΚΗΣ ΤΕΧΝΟΛΟΓΙΑΣ
ΔΠΜΣ ”ΔΟΜΟΣΤΑΤΙΚΟΣ ΣΧΕΔΙΑΣΜΟΣ ΚΑΙ ΑΝΑΛΥΣΗΣ ΤΩΝ ΚΑΤΑΣΚΕΥΩΝ”

Μεταπτυχιακή Διπλωματική Εργασία:
Προχωρημένες Διαδικασίες Μη Γραμμικής Ανάλυσης Πλαισιωτών Φορέων σε Σεισμική Φόρτιση
Νικηφόρος Ρεπούσης
Επιβλέπων: Αναπληρωτής Καθηγητής Μιχαήλ Φραγκιαδάκης
Αθήνα, Μάρτιος 2022

Ο υπολογισμός της ακριβούς απόκρισης μιας κατασκευής υπό σεισμική φόρτιση αποτελεί ένα θεμελιώδες πρόβλημα για τον τομέα της Αντισεισμικής Τεχνολογίας. Ωστόσο, παρουσιάζει αρκετές δυσκολίες καθώς τα κύρια δομικά υλικά -όπως το οπλισμένο σκυρόδεμα και ο χάλυβας- παρουσιάζουν πλαστική (μη γραμμική) συμπεριφορά όταν τους επιβληθούν μεγάλα φορτία. Τα τελευταία χρόνια εντείνεται η έρευνα για την ανάπτυξη ακριβέστερων αριθμητικών μοντέλων, ικανών να περιγράψουν αυτή τη συμπεριφορά των υλικών και κατα συνέπεια των κατασκευών.

Η προσομοίωση ενός κτιρίου γίνεται συνήθως με χρήση ραβδόμορφων στοιχείων, τα οποία πρέπει να λαμβάνουν υπόψη τους την πλαστική συμπεριφορά που αναπτύσσεται , ώστε να μπορούν να περιγράψουν σωστά τη συμπεριφορά του φορέα και να οδηγήσουν σε ασφαλή και οικονομικό σχεδιασμό. Για τα ραβδόμορφα προσομοιώματα έχουν αναπτυχθεί δύο κύριες θεωρήσεις για την ενσωμάτωση της πλαστικής συμπεριφοράς κατά την ανάλυση ενός φορέα: η θεωρία της συγκεντρωμένης πλαστικότητας και η θεωρία της κατανεμημένης πλαστικότητας. Στην παρούσα μεταπτυχιακή διπλωματική εργασία εξετάστηκε η θεωρία της συγκεντρωμένης πλαστικότητας για τους ραβδόμορφους φορείς λόγω της απλότητας της, των χαμηλών υπολογιστικών της απαιτήσεων και της σχετικά υψηλής ακρίβειας όπως φαίνεται και από τη βιβλιογραφία.

Επομένως, το αντικείμενο της παρούσας μεταπτυχιακής διπλωματικής εργασίας ήταν η ανάπτυξη αλγορίθμων στη γλώσσα προγραμματισμού MATLAB για την προσομοίωση της συμπεριφοράς ραβδόμορφων φορέων υπό μονοτονική -οριζόντια και κατακόρυφη-

φόρτιση με χρήση της θεωρίας της συγκεντρωμένης πλαστικότητας. Τέλος, έγινε η προσομοίωση δύο πλαισιωτών φορέων οπλισμένου σκυροδέματος υπό οριζόντια φόρτιση με χρήση ελαστικών στοιχειών και στοιχείων συγκεντρωμένης πλαστικότητας, και συγκρίθηκαν τα αποτελέσματα που προέκυψαν με τα αντίστοιχα πειραματικά. Προέκυψε ότι τα αποτελέσματα των προσομοιωμάτων που λάμβαναν υπόψη την ύπαρξη της πλαστικότητας ήταν πιο ακριβή σε σχέση με το φυσικό πρόβλημα.

ABSTRACT

NATIONAL TECHNICAL UNIVERSITY OF ATHENS
SCHOOL OF CIVIL ENGINEERING
LABORATORY OF EARTHQUAKE ENGINEERING
MSc in ANALYSIS AND DESIGN OF EARTHQUAKE RESISTANT STRUCTURES

Postgraduate Thesis:
Advanced Non Linear Analysis of Frame Structures under Seismic Loading
Nikiforos Repousis
Supervisor: Associate Professor Michail Fragiadakis
Athens, March 2022

Determining the exact behaviour of a structure under seismic loading is considered to be a fundamental problem in the field of Structural and Earthquake Engineering. However, it is not an easy task, due to the fact that the main structural materials -such as reinforced concrete and steel- exhibit plastic behaviour (non-linear) when loaded heavily. That is the reason why a large amount of research has been performed over the years in order to develop mathematical models able to depict this behaviour of materials and consequently of structures.

A typical way to simulate a building is through beam elements, but the plastic behaviour of its components need to be taken into account in order to create a method that leads to the safe and efficient design of structures. There are two main theories developed in order to incorporate this plastic behaviour into structures composed of beam elements: the lumped (concentrated) plasticity and the distributed plasticity theory. In this postgraduate thesis, the lumped plasticity theory is examined due to its simplicity, its low computational needs and its high accuracy as shown in the literature.

Therefore, the lumped plasticity beam element was programmed in MATLAB programming language in order to create a tool for the analysis of structures under monotonic horizontal and vertical loading. After comparing the results of the elastic beam element model -that do not account for the plastic behaviour- and the lumped plasticity beam element model with the experimental results of two reinforced concrete frame structures that were subjected to lateral loading, it is evident that the beam elements incorporating the plastic phenomena were more efficient in depicting the real behaviour of the structure.

TABLE OF CONTENTS

TABLE OF FIGURES

CHAPTER 1. INTRODUCTION

1.1 General

The analysis of structures is categorized into the linear and the non-linear approach. Depending on the assumption used for implementation of the loads to a structure, the static or the dynamic method is then selected (See Figure 1.1). Moreover, experimental observations have demonstrated that the behaviour of structural materials such as steel and concrete is associated with significant non-linearity, making the non-linear approach relevant when designing a safe and economical structure. Furthermore, one of the most crucial loading is the seismic one, which can be easily modelled in a simplified manner with the non-linear static analysis (pushover) method. As a result, the goal of the present postgraduate thesis is the development of an efficient numerical tool, which will allow the structural engineers to predict the structural performance of a structure in a simplified manner with acceptable precision and reduced computational cost.

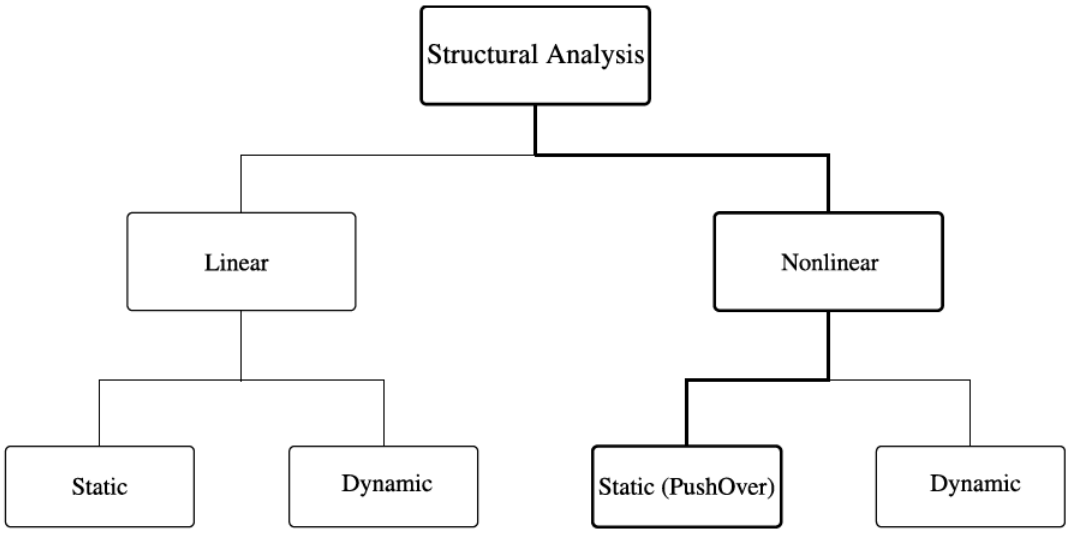


Figure 1.1: Different Approaches for the Structural Analysis of Frame Structures (Mohsen R. H., 2018)

1.2 Plastic Behaviour of Materials

Elasticity is the tendency of solid objects and materials to return to their original shape after the external forces causing a deformation are removed. An object is elastic when it comes back to its original size and shape when the load is no longer present. The elastic limit is the stress value beyond which the material no longer behaves elastically but becomes permanently deformed. For stresses beyond the elastic limit, a material exhibits

plastic behaviour. This means that the material deforms irreversibly and does not return to its original shape and size, even when the load is removed. When stress is gradually increased beyond the elastic limit, the material undergoes plastic deformation. Plasticity enables a solid under the action of external forces to undergo permanent deformation without rupture. This phenomenon is of high important for the design of economical and safe structures.

There are two wide used methods to model the plastic behaviour of the members of a structure. More specifically, the concentrated plasticity (lumped plasticity) models presume that the in-elasticity is limited to the end regions of the structural elements. This presumption is invoked by the fact that under lateral excitation (e.g. earthquake or wind), the structural elements experience larger values of bending moments at end regions. The concentrated plasticity models are constructed as a combination of either parallel or series subelements. On the other hand, the distributed plasticity assumes a parabolic distribution of the flexural and shear stiffnesses along the element length. The distributed plasticity approach involves the evaluation of the behaviour at fixed points along the element span. These points are coincident with the quadrature points and depend on the adopted integration rule. In order to obtain accurate results in the framework of distributed plasticity, the use of finer meshes, higher order shape functions, or more quadrature points along the element is required, which inevitably leads to high computational complexity and time requirements.The formulation of the distributed plasticity models are usually either based on the displacement-based element or force-based element. For this thesis, the lumped plasticity model is used for the modelling of the plastic behaviour of the members of a structure.

1.3 Scope of Postgraduate Thesis

The human civilization was always focused on the construction of structures in order to be protected from the natural phenomena. With the development of modern technology, the methods of designing these structures is constantly evolving with the goal of minimizing their cost and increasing their durability to be desirable. For this reason, the development of mathematical models able of simulating the behaviour of reinforced concrete and steel structure -that nowadays are the main structural materials- is of high interest the last decades. And since it is experimentally observed that both of these material behave plastically after their capacity is reached, there are lot of models developed trying to simulate this behaviour. Thus the objective of this postgraduate thesis was the development of a tool -using MATLAB programming language- which will analyse structures composed of beam elements using the one component lumped plasticity theory as proposed by Giberson (1967) in order to simulate the plastic behaviour of their members under monotonic loading.

1.4 Postgraduate Thesis Layout

This postgraduate thesis has the following layout:

- Chapter 2 : The theory of the beam element, the Newton-Raphson method and the material laws, that were used in this thesis, are presented.
- Chapter 3 : The lumped plasticity beam element model and its formulation is discussed.
- Chapter 4 : A variety of examples for the lumped plasticity beam element model is presented.
- Chapter 5 : The main remarks and suggestions are highlighted.
- Bibliography : The bibliography is listed in alphabetical order.
- Appendix : Parts of the scripts developed in the programming language MATLAB for the beam element with lumped plasticity are displayed. Also, the input files used in Opensees and ASDAP (Femlab) are shown.

CHAPTER 2. BEAM ELEMENT, NUMERICAL METHODS AND MATERIAL LAWS

2.1 Beam Element

2.1.1 General

The analysis of structures using computer soft-wares requires the existence of ways to simulate them. There is a variety of methods to simulate the behaviour of a structure such as the single degree of system method. However, the beam element is the most common modelling method for buildings due to its high accuracy and its ability to simulate all of components of the structure at hand. In this section the two-dimensional beam element is analysed due to its simplicity, but its extension to the three-dimensional beam element is rather straightforward.

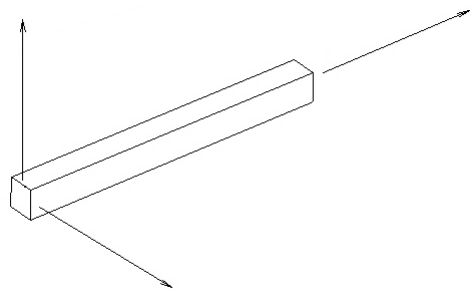


Figure 2.1: The Beam Element

2.1.2 Coordinate Systems and Transformation

It is important to define the three coordinates system of the beam element which are: the global, the local and the basic coordinate system. The local and global coordinate system are most commonly known as the Cartesian systems. The basic system -also known as corotational or natural systems- is always attached to the beam.

2.1.2.1 The Local Coordinate System

The local coordinate system follows the orientation of the element. Most specifically, the x-axis is always parallel to the beam axis and the y-axis is perpendicular to the x axis (See Figure 2.2). The direction of the x-axis is typically defined from the start node to the end node. The force and the displacement vectors in the local Cartesian system are defined as:

$$P_e = \begin{bmatrix} F_{e1} \\ F_{e2} \\ F_{e3} \\ F_{e4} \\ F_{e5} \\ F_{e6} \end{bmatrix} \quad u_e = \begin{bmatrix} u_{e1} \\ u_{e2} \\ u_{e3} \\ u_{e4} \\ u_{e5} \\ u_{e6} \end{bmatrix} \tag{2.1.1}$$

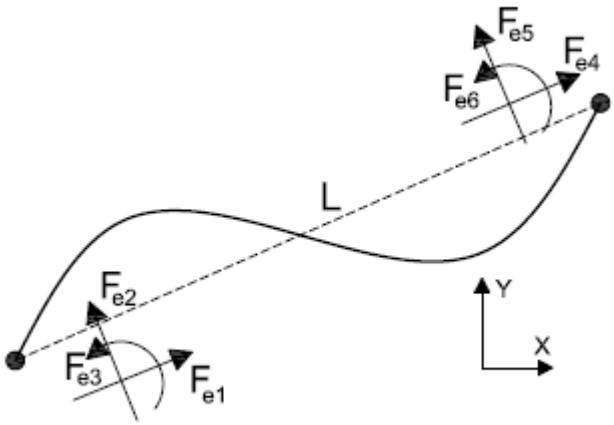


Figure 2.2: The Local Coordinate System (Fragiadakis M., 2020)

2.1.2.2 The Global Coordinate System

All stiffness matrices and force vectors of all the beam elements are rotated to the global system in order to assemble the global stiffness matrix and solve the equation of equilibrium of the structure. The global system is common for all the elements of the structure and the global force P_g and displacement u_g vectors are parallel to the global x and y axes (See Figure 2.3). The force and displacement vectors are defined as:

$$P_g = \begin{bmatrix} F_{g1} \\ F_{g2} \\ F_{g3} \\ F_{g4} \\ F_{g5} \\ F_{g6} \end{bmatrix} \quad u_g = \begin{bmatrix} u_{g1} \\ u_{g2} \\ u_{g3} \\ u_{g4} \\ u_{g5} \\ u_{g6} \end{bmatrix} \tag{2.1.2}$$

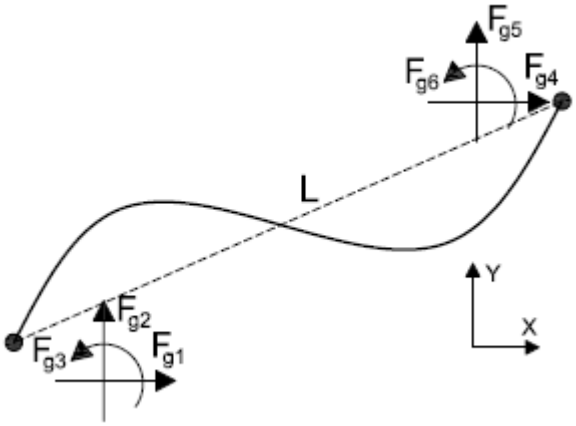


Figure 2.3: The Global Coordinate System (Fragiadakis M., 2020)

2.1.2.3 The Basic Coordinate System

As described by Fragiadakis M. (2020), the entire structure deforms from its original configuration when it is loaded. As a result the displacement vectors u_e and u_g - which contain information on how a beam element rotates, translates and deforms- include rigid body rotation and translations, which should be removed from the motion of the beam because they do not have any straining effect on each individual element. That is the reason why the basic system was introduced, which is a coordinate system attached to each element monitoring its motion as it deforms. Forces and displacements in the basic coordinate system are obtained from corresponding quantities of the local coordinate system using simple algebraic operations that allow removing the effect of rigid body motion from the displacement or the force vectors. The basic system has three degrees-of-freedom and the displacement vector v consists of: the axial elongation e and the two bending rotations θ_1 , θ_2 at each end. Moreover, the basic force vector S, consists of the axial force P and two bending moments M_1, M_2, respectively (See Figure 2.4). Thus S and v can describe any stress condition of a deformed beam element, which is required for geometric and material non-linearity problems.The basic force and displacement vectors are defined as:

$$S = \begin{bmatrix} P \\ M_1 \\ M_2 \end{bmatrix} \; v = \begin{bmatrix} e \\ \theta_1 \\ \theta_2 \end{bmatrix} \tag{2.1.3}$$

2.1.2.4 Transformations Between Coordinate Systems

The transformation between the local and the global Cartesian system is obtained using the transformation matrix T -where q is the element orientation angle and $c = cos(q)$,

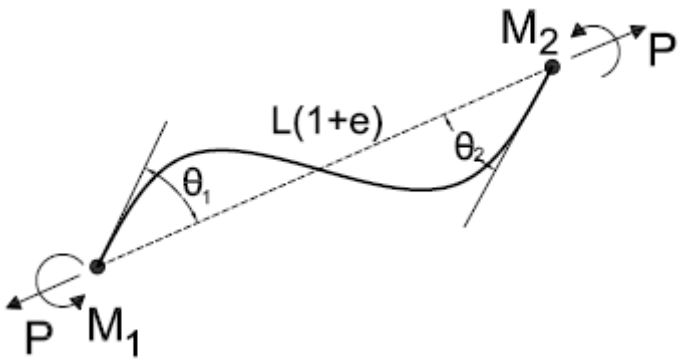


Figure 2.4: The Basic Coordinate System (Fragiadakis M., 2020)

$s = sin(q)$- as describe below:

$$T = \begin{bmatrix} c & s & 0 & 0 & 0 & 0 \\ -s & c & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & c & s & 0 \\ 0 & 0 & 0 & -s & c & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{2.1.4}$$

$$F_g = T^T F_e \tag{2.1.5}$$

$$u_g = T^T u_e \tag{2.1.6}$$

For small displacements -that are of interest for this thesis-, the relationship between the forces in the basic and the local Cartesian coordinate system are obtained as follows (See Figure 2.5):

- $F_1 = -P$
- $F_2 = \frac{M_1+M_2}{L}$
- $F_3 = M_1$
- $F_4 = P$
- $F_5 = -\frac{M_1+M_2}{L}$
- $F_6 = M_2$

Thus the above equations in a matrix form:

$$\begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{bmatrix} = \begin{bmatrix} -1 & 0 & 0 \\ 0 & \frac{1}{L} & \frac{1}{L} \\ 0 & -1 & 0 \\ 1 & 0 & 0 \\ 0 & -\frac{1}{L} & -\frac{1}{L} \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} P \\ M_1 \\ M_2 \end{bmatrix} \tag{2.1.7}$$

Therefore, the combined equation derived for the forces is the following:

$$P_e = b_e S \tag{2.1.8}$$

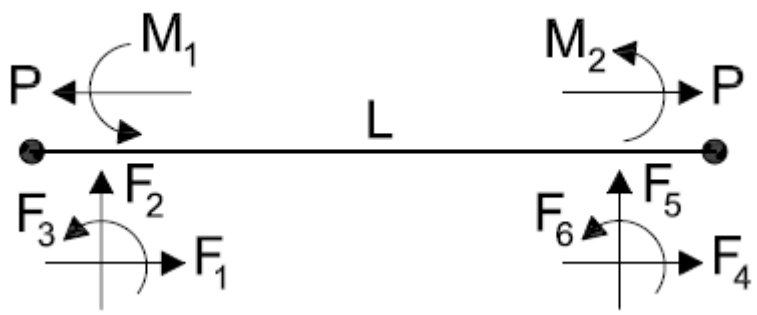


Figure 2.5: The Basic and Local Forces (Fragiadakis M., 2020)

Under the assumption of small displacements, according to Fragiadakis M.(2020) it is shown that the rigid body rotation β is equal to:

- $\beta = \frac{u_5 - u_2}{L} = \frac{u_5 - u_2}{L_0}$

Therefore, the relationship between the displacements in the basic and the local Cartesian coordinate system are obtained as follows (See Figure 2.6):

- $e = u_4 - u_1$
- $\theta_1 = u_3 - b = \frac{u_5 - u_2}{L}$
- $\theta_2 = u_6 - b = \frac{u_5 - u_2}{L}$

Grouping the above equations in a matrix form:

$$\begin{bmatrix} e \\ \theta 1 \\ \theta 2 \end{bmatrix} = \begin{bmatrix} -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & \frac{1}{L} & 1 & 0 & -\frac{1}{L} & 0 \\ 0 & \frac{1}{L} & 0 & 0 & -\frac{1}{L} & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{bmatrix} \tag{2.1.9}$$

Therefore, the combined equation derived for the displacements is the following:

$$v = b_e^T u_e \tag{2.1.10}$$

It is highlighted that the opposite transformations from local to basic displacements is not possible. Also, the transformation equations are obtained in a similar fashion for large displacements -see Fragiadakis M. (2020) for more details.

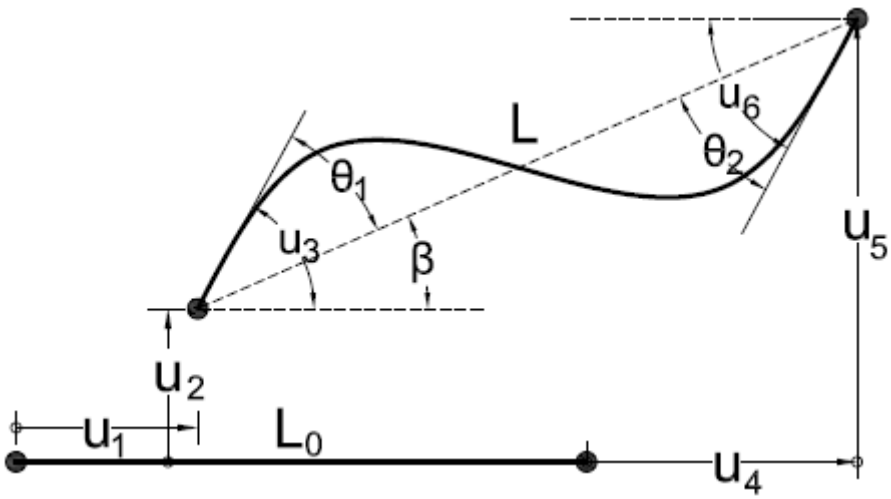


Figure 2.6: The Basic and Local Displacements (Fragiadakis M., 2020)

2.1.3 Stiffness Matrix and Internal Forces

2.1.3.1 Stiffness Matrix

The tangent stiffness matrix consists of two matrices, the first term represents the geometric contribution and the second is the material contribution. More specifically, the tangent stiffness matrix of a beam element at the local system will be:

$$K_e = K_T = K_E + K_G \tag{2.1.11}$$

Therefore the tangent stiffness matrix of a beam element at the global system will be:

$$K_g = T^T K_E T + T^T K_G T \tag{2.1.12}$$

However, for this thesis only the material contribution is considered. As a result, the stiffness matrix used is:

$$K_e = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & \frac{-EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & \frac{-12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & \frac{-6EI}{L^2} & \frac{2EI}{L} \\ \frac{-EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & \frac{-12EI}{L^3} & \frac{-6EI}{L^2} & 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & \frac{-6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \tag{2.1.13}$$

For the basic system -when the deformations are known-, we derive the forces as follows:

- $N = \frac{EAe}{L}$
- $M_1 = \frac{2EI}{L}(2\theta_1 + \theta 2)$
- $M_2 = \frac{2EI}{L}(\theta_1 + 2\theta 2)$

Gathering the above relationships in a matrix form, we have the following equation:

$$\begin{bmatrix} P \\ M_1 \\ M_2 \end{bmatrix} = \begin{bmatrix} \frac{EA}{L} & 0 & 0 \\ 0 & \frac{4EA}{L} & \frac{2EA}{L} \\ 0 & \frac{2EA}{L} & \frac{4EA}{L} \end{bmatrix} \begin{bmatrix} e \\ \theta 1 \\ \theta 2 \end{bmatrix} \tag{2.1.14}$$

As a result, we obtain the stiffness matrix K_N in the basic system as:

$$K_N = \begin{bmatrix} \frac{EA}{L} & 0 & 0 \\ 0 & \frac{4EA}{L} & \frac{2EA}{L} \\ 0 & \frac{2EA}{L} & \frac{4EA}{L} \end{bmatrix} \tag{2.1.15}$$

Therefore the relationship between the basic and local stiffness matrix is the following:

$$K_e = b_e K_N b_e^T \tag{2.1.16}$$

2.1.3.2 Internal Forces

The calculation of the internal forces of the beam element is necessary in order to check if the system is in equilibrium. The internal basic forces S are obtained as shown below:

$$S = K_N v \tag{2.1.17}$$

In the local Cartesian system the vector of internal forces will be:

$$F_e = b_e S \tag{2.1.18}$$

The internal forces F should be finally expressed in the global system in order to check if the system is in equilibrium:

$$F_g = T^T F_e \tag{2.1.19}$$

Also, the following equation holds for the transformation of forces between the basic and the local coordinate system:

$$S = (b_e^T b_e)^{-1} b_e^T F_e \tag{2.1.20}$$

2.2 Numerical Methods

2.2.1 General

When dealing with non-linear problems, the following equation for internal forces cannot be estimated explicitly:

$$f_s = K_s u_s \tag{2.2.1}$$

Therefore iterative solution algorithms are required to solve this problem. The most commonly used algorithm is Newton-Raphson which is an incremental step method for finding successively better approximations to the roots of a non-linear set of equations. Alternative versions of this method were proposed in the literature for handling the incremental steps, such as the Force-Control, the Displacement-Control and the Arc-Length. For this thesis, the Force-Control Newton-Raphson Method is used.

2.2.2 The Newton-Raphson Method (Force-Control)

According to Markou G. (2011), if the structural problem at hand contains non-linearities (material and/or geometrical) in order to find the equilibrium state between the internal forces of the structure and the external loads, it is assumed that the applied loads can be expressed as a function of pseudo-time t, for static problems, and the equilibrium at each step can be expressed as:

$$F_{s,t} - R_{s,t} = 0 \tag{2.2.2}$$

where $F_{s,t}$ are the externally applied nodal forces of the structure at time t and $R_{s,t}$ are the nodal forces that correspond to the internal stresses of the structure. The relation of the above equation expresses the equilibrium of the system in the current deformed geometry accounting for all non-linearities. It is important to note that, this relation is general and applies also for dynamic problems where the forces due to inertia and damping are included in the array. Therefore, by dividing the external forces into n load steps and by using a specific load increment ($Dt = \frac{1}{n}$), the external loads are applied incrementally and at each time step a new load increment is added to the structure external loading. This requires the satisfaction of the above equation through the whole loading time history. For the case of static loads, the definition of time is only a convenient variable which specifies different load levels and, correspondingly, different structural configurations. Assuming that i is the current load step of the analysis, then the accepted solution can be stated as:

$$u_{s,t}, t = iDt \tag{2.2.3}$$

therefore, the solution of the next load increment at time $t + Dt$ will be:

$$u_{s,t+Dt} = u_{s,t}, Du_s = u_{s,Dt}, t_{i+1} = (i+1)Dt \tag{2.2.4}$$

For the computation of the unknown displacements at load increment t_{i+1} a prediction of the solution is obtained by using the stiffness matrix of the previous load increment.

$$K_{s,t} Du_{s,t+Dt} = Df_{s,t+Dt} \tag{2.2.5}$$

The next stage of the non-linear algorithm is to compute the resisting forces at each node of the structure and assemble the array in order to verify if the equation of equilibrium is satisfied. In non-linear solution algorithms, this equation is never equal to zero thus a convergence criterion is applied which specifies if convergence is achieved. The result of the equation of equilibrium is the vector of the residual forces. This vector is used in order

to compute the error of the iterative procedure according to the adopted convergence criterion.

$$e_{er} = \frac{||Du_{s,j}||}{||u_{s,t+Dt}||} <= e_D \tag{2.2.6}$$

$$e_{er} = \frac{||F_{s,t+Dt} - R_{s,t+Dt}||}{||F_{s,t+Dt} - R_{s,t}||} <= e_F \tag{2.2.7}$$

$$e_{er} = \frac{Du_{s,j}||F_{s,t+Dt} - R_{s,t+Dt}||}{Du_{s,1}||F_{s,t+Dt} - R_{s,t}||} <= e_F \tag{2.2.8}$$

$$r_{s,j} = F_{s,t+Dt} - R_{s,t+Dt} \tag{2.2.9}$$

where j is the corresponding internal iteration, e_D is the displacement, e_F is the force and e_G is the energy convergence tolerance criterion, respectively. For each internal iteration j, the stiffness matrix of the structure is updated by using the new material properties which are implemented through the material constitutive matrix **C** of the finite element formulation. If the numerical problem at hand accounts for geometrical non-linearities, then the stiffness matrix of the FE model is also affected by the current configuration. The updated global stiffness matrix is known as the tangent stiffness matrix. In the event that the convergence criterion is not satisfied, the residual forces are applied as external forces and the non-linear solution algorithm proceeds with the $j+1$ internal iteration.

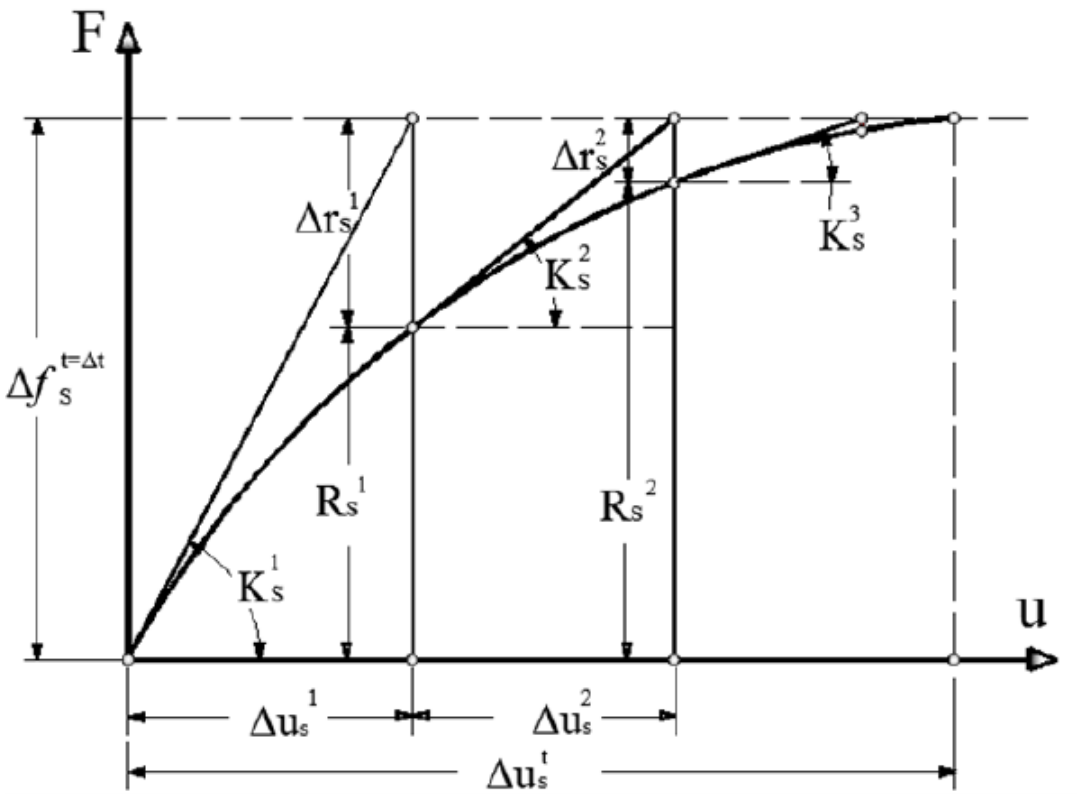


Figure 2.7: The Full Newton-Raphson Iterative Scheme (Markou G., 2011)

When the global stiffness matrix of the structure is updated for each internal iteration, then we have the full NR scheme (See Figure 2.7). This is computationally demanding with respect to the computational effort required for the factorization and back substitution procedures of the stiffness matrix at each iteration, but at the same time this effort is counter balanced by the increased convergence properties of the method. A reduction of the computational cost per iteration may be achieved with alternative NR algorithms like the “Modified” scheme or quasi-Newton scheme, where the stiffness matrix is updated after a specific number of internal iterations or implicitly after each iteration, respectively, during the solution procedure. The disadvantage of these methods is the slow convergence rate for cases with strong non-linearities requiring larger number of iterations until convergence. This is illustrated in Figures 2.8 and 2.9 where two NR schemes are presented. It is worth mentioning that the force-control NR schemes appear to be numerically less stable than the corresponding displacement-control.

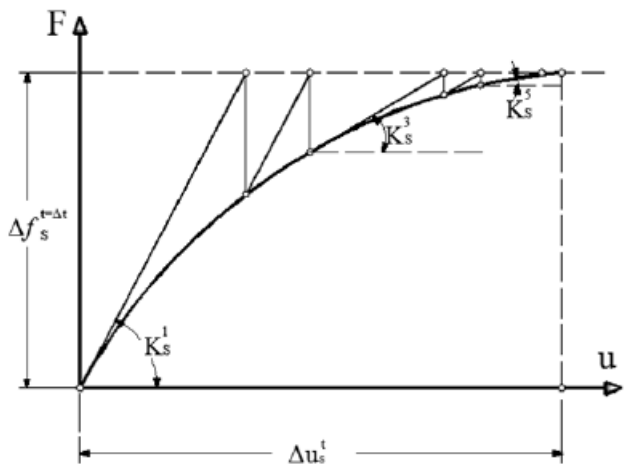


Figure 2.8: The Modified Newton-Raphson Iterative Scheme (Markou G., 2011)

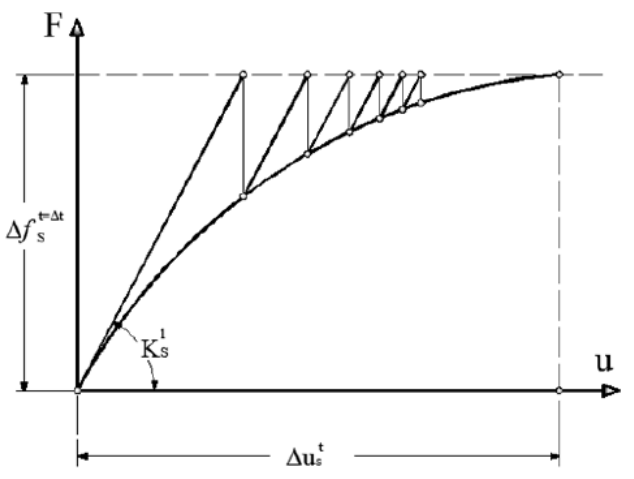


Figure 2.9: The Quasi Newton-Raphson Iterative Scheme (Markou G., 2011)

2.3 Material Laws

2.3.1 General

The selection of the right material law for the members of structures -when analysing them- is of high importance, in order to simulate correctly their properties and behaviour. The constitutive relationships selected will govern the performance of the structure analysed, especially when the analysis is non-linear. Therefore, the choice of a material model should be made carefully, taking on mind the structure's type and the way it is simulated. In this section, the elastic and bilinear model will be presented briefly.

2.3.2 Elastic Model

Elastic design is carried out by assuming that at design loads structures behave in a linearly elastic manner. Since the element forces are determined based on elastic behaviour, the design is governed by elastic stiffness distribution (ratios) among the system elements. It is commonly understood that most structures designed by elastic method possess considerable reserve strength beyond elastic limit until they reach their ultimate strength. The reserve strength is derived from factors, such as structural redundancy, ability of structural members to deform inelastically without major loss of strength (i.e., ductility). However, one drawback of using elastic method for designing such structures with ductile members is that the reserve strength beyond elastic limit is neither quantified nor utilized explicitly. But more importantly, the yield state of the structure at ultimate strength level is also not known. The yield mechanism may involve structural members that could lead to undesirable system performance under accidental overloading or extreme events, such as strong earthquake ground motion, blast, impact, etc.

The Force-Displacement relationship for the **Linear Elastic Model** is described from the following equation:

$$F = K_{elastic} u \tag{2.3.1}$$

,where $K_{elastic}$ is the elastic stiffness, u the displacement and F is the imposed force.

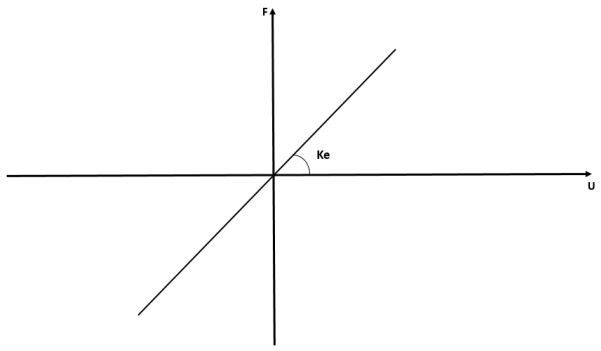


Figure 2.10: The Linear Elastic Model (Repousis N., 2019)

2.3.3 Non Linear Models

2.3.3.1 General

In the present postgraduate thesis, only monotonic loading case are examined, therefore the bilinear model can be used for the simulation of both reinforced concrete and steel structures with relatively high accuracy. However, more sophisticated hysteretic material laws might be needed when cyclic loading is examined as shown by Repousis N. (2019).

2.3.3.2 Bilinear Model

The **Bilinear Model** is described by two curves:

- the elastic curve, when the material behaves elastically
- the inelastic curve, after the material has yielded i.e. the yield criterion is met

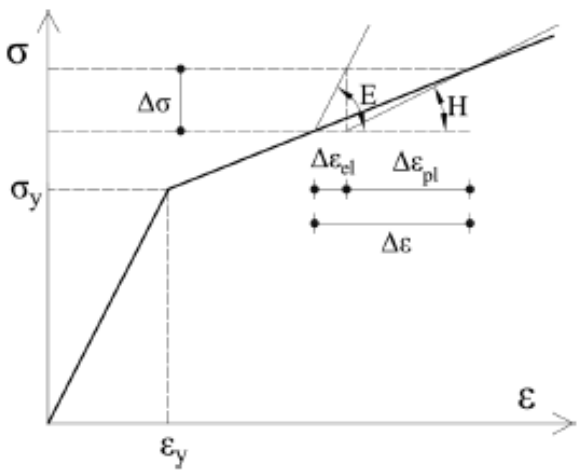


Figure 2.11: The Bilinear Model (Fragiadakis M., 2020)

The estimation of the stress for each step can be computed as follows (the same procedure can be used for force-displacement or moment-curvature relationships):

- $a_i = max[\sigma_{i-1} - \sigma_{y0}, \frac{b}{b-1}(\sigma_{i-1} - E\epsilon_{i-1})]$
- $\sigma_{elastic} = \sigma_{i-1} + E\epsilon_{increment}$
- $\eta = \sigma_{elastic} - a_i$
- $q = |\eta| - \sigma_{y0}$
- if $q \leq 0$, then $\sigma_i = \sigma_{elastic}$

- if $q > 0$, then $\sigma_i = \sigma_{elastic} - sign(\eta)E_0\epsilon_{increment,plastic}$,where $\epsilon_{increment,plastic} = \frac{q(1-b)}{E_0}$

Where:

- E is the elastic modulus of the material
- b is the hardening ratio
- σ_{y0} is the yield stress
- σ_{i-1} is the stress at step $i-1$
- ϵ_{i-1} is the strain at step $i-1$
- $\epsilon_{increment}$ is the strain increment from step $i-1$ to step i
- σ_i is the stress computed for the step i

At the initial development stage of non-linear dynamic analysis, the bilinear hysteretic model was used by many investigators. As described above the response point moves:

- on the elastic stiffness line before the yield stress is reached
- after yielding the response point moves on the perfectly plastic line until unloading takes place
- upon unloading, the response point moves on the line parallel to the initial elastic line i.e unloading stiffness after yielding is equal to the initial elastic stiffness

This model does not consider degradation of stiffness under cyclic loading. This is the reason why more sophisticated models -such as Clough and Johnston - were developed in order to describe the hysteretic behaviour of materials such as steel and reinforced concrete. However, this hysteretic behaviour exceeds the scope of this thesis.

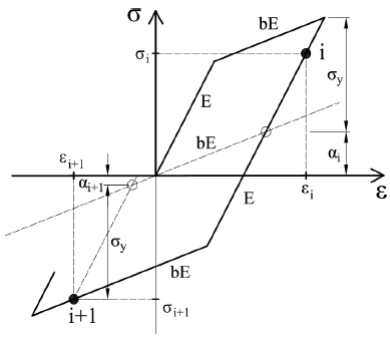


Figure 2.12: The Bilinear Hysteretic Model with Kinematic Hardening(Fragiadakis M., 2020)

CHAPTER 3. BEAM ELEMENT WITH LUMPED PLASTICITY

3.1 General

The prediction of the distribution of forces and deformations in structures under earthquake excitations requires arithmetically accurate models, that are able to describe the non-linear behaviour that is expected in the critical regions after plastic hinges have appeared. It should be noted here that the term plastic hinge is used to describe a section of structural element in which a plastic bending occurs. The lumped plasticity models are one of the main categories of such models developed, that are able to describe this behaviour. In this section, previous work presented in the literature regarding the different lumped plasticity models will be discussed briefly with main focus on the series model presented by Giberson(1967).

3.2 Different Models of Lumped Plasticity

It is observed by experimental investigations that under seismic excitations the inelastic behaviour of frames is mainly concentrated at the ends of girders and columns where maximum moments occur -opposed to element dead and live loads that produce moments in the middle span of beam members-. Therefore, it is assumed by lumped plasticity formulation that the inelasticity is “lumped” at the ends of the members and the rest behave linear elastic. Comprehensive reviews of the lumped plasticity models are presented in Fragiadakis(2020), Markou G. (2011) and Reshotkina (2015), among others.

The last decades, several concentrated plasticity models known as “lumped plasticity models” have been proposed in order to simulate the inelastic deformations of the member under seismic loading. These models are constructed as a combination of either parallel or series subelements. The first point-hinge model was introduced by Clough, Benuska and Wilson in 1965, and is named the two-component model. It consists of two structural beam components in a parallel series. One component is elastic-perfectly plastic, while the other is elastic without any ultimate limit (See Figure 3.1). The elastic member accounts for the strain hardening characteristics of the reinforcing steel, while the elastic perfectly plastic member accounts for yielding of the reinforcement. The interaction between these two components enables the model to represent bilinear response. As the formulation is based on a parallel model, the total beam stiffness is determined by directly summing up the stiffnesses of both components. The factor γ represents the ratio between the elastic stiffness EI and the post-yield stiffness $(1-\gamma)EI$. Thus the elastic rotational stiffness will be the sum of both components:

$$K_{elastic} = \gamma EI + (1-\gamma)EI = EI \tag{3.2.1}$$

while the post-yield stiffness will be the rotational stiffness of only the second component, as the first component has reached perfect plasticity, and therefore has zero stiffness. The moment-curvature relation of the two-component model is shown in Figure 3.2. Despite

the two-component model's strength of simulating an exact bilinear response, it does not represent the cyclic loading of concrete members with sufficient accuracy. The model overestimates the energy dissipation when members are subjected to inelastic load cycles. Therefore the model is only applicable for steel members with stable hysteresis loops, or non-cyclic inelastic deformations concrete members.

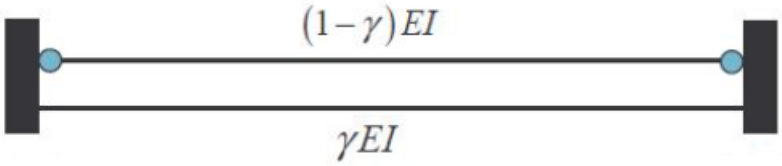


Figure 3.1: The Two Component Model (Gharakhanlo A.,2014)

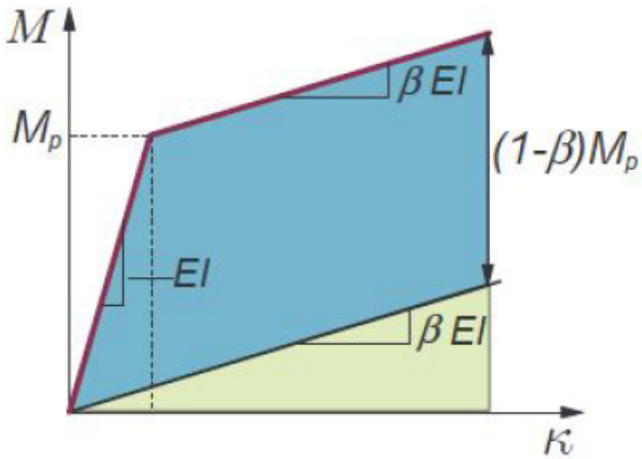


Figure 3.2: The Moment-Curvature Relation of the Two-Component Model (Gharakhanlo A.,2014)

The multicomponent extension of the model was developed by Aoyama and Sugano. The model consists of three elastic elements and two unique elasto-plastic rotational springs with trilinear hysteresis behaviour. The unique characteristic of the two rotational springs allows different level of concrete cracking and reinforcement steel yielding at the two ends of the element. However, due to the lack of versatility in the hysteresis loop of the multicomponent beam model, Giberson (1967) proposed a one-component beam element with rotational springs attached to its ends in series. It consists of a linear elastic beam with non-linear rotational springs at its member ends (See Figure 3.3). These springs only contribute to the rotational stiffness when the plastic capacity of the beam is reached at a particular end. The advantage is that the rotation depends solely on the moment acting at the end and thus any moment rotation hysteresis model can be assigned to the spring. Therefore, the one component beam element is able to describe curvilinear hysteresis loop, hence more appropriate for the hysteretic behaviour of RC members. The performance of the "one component model" is expected to be reasonably good for

relatively low-rise frame structures, in which the inflection point of the column is close to the mid-height. The one-component or series model will be further examined in the following section.

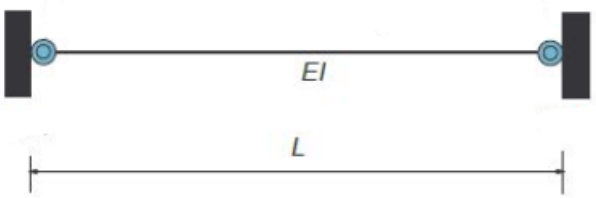


Figure 3.3: The One-Component or Series Model (Gharakhanlo A.,2014)

One major limitation of the concentrated plasticity models that needs to be attended is the effect of zero inelastic length zone that may lead to an overestimation of the ultimate strength of structures. Therefore, a more refined model of the nonlinear behavior of RC girders was first proposed by Soleimani et. al. (1979). More specifically, according to Filippou and Issa (1988) since the deformations of the girder before yielding of the reinforcement are accounted for in the elastic beam subelement, the spread rigid-plastic subelement only accounts for the inelastic girder deformations which take place when the end moments exceed the yield moment. The spread rigid-plastic beam subelement consists of two regions of finite length where the plastic deformations of the girder take place. These regions are connected by an infinitely rigid bar (See Figure 3.4). The length of each plastic zone varies during the response history as a function of the moment distribution in the girder. The model thus accounts for the gradual spread of inelastic deformations into the girder and the shift of the inflection point during the response time history.

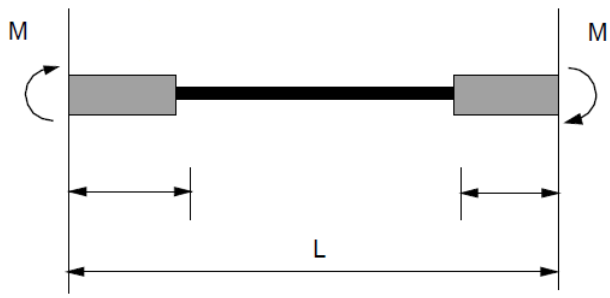


Figure 3.4: The Soleimni et al.(1979) Model (Filippou and Issa,1988)

Al-Haddad and Wight (1986) modified this model by varying the location of the plastic

hinges at the ends of the member. This model accounts for rigid end zones in conjunction with an elastic line element. The inelastic action is concentrated at the two plastic hinge locations (See Figure 3.5).

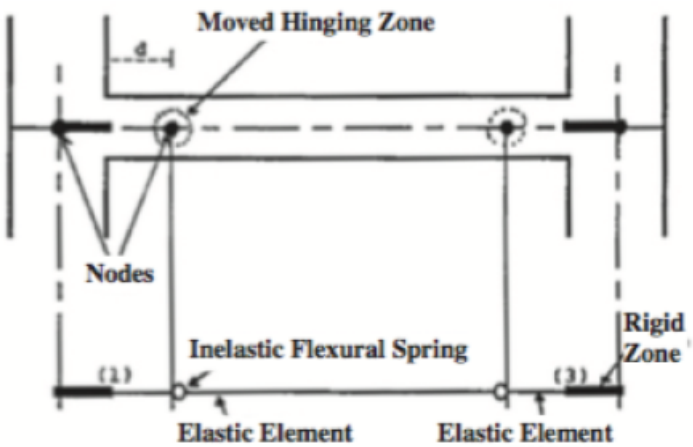


Figure 3.5: The Al-Haddad and Wight (1986) Model (Fragiadakis M.,2020)

The non-linear frame models based on the concentrated plastic hinge concepts are a simplification of the actual behaviour of reinforced concrete members which is characterized by the gradual spread of inelastic deformations along the member as a function of the loading history (See Figure 3.7). The element formulations of the concentrated plasticity models are based on plasticity relationships between the member end forces and the member end deformations, and therefore these relationships require calibrations based on expected axial load and moment gradient along the member. The main advantage of concentrated plasticity models is their relative simplicity and computational efficiency.

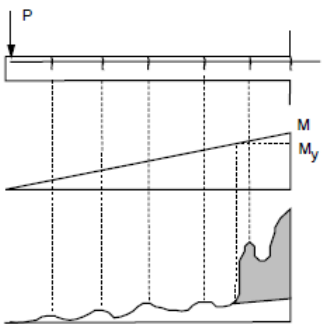


Figure 3.6: Development of Plastic Hinge in the case of a Cantilever Beam Loaded by a Concentrated Load at the Tip (Filippou and Issa,1988)

In subsequent models, the interaction between axial force and bending moment was included to account for the effect of the pullout-reinforcing steel and the effect of the concrete-filled tubes as mentioned by Fragiadakis M. (2020). To overcome some of the

limitations in the single-component hinge models, recent studies have employed yield-surface and evolution models approach to account for the force interaction in the case of multiaxial loading [Ricles et al. (1998), ElMandooh Galal (2003), Kaul (2004)] as suggested by Reshotkina (2015).

3.3 The Giberson (1967) Model

3.3.1 General

Giberson (1967) proposed the one-component beam element model for modelling elasto-plastic frame structures in his PhD thesis. This model is based on series spring analogy. It consists of two components. The first one is the perfectly elastic beam-column member, which has a length equal to the real member length. The second component are the two zero-length rotational springs, that are placed at the ends of the elastic member. The non-linear behaviour of the beam or column is simulated by the rotational springs. One of the major advantages of this model is that any moment-rotation hysteretic rule can be assigned into the springs. On the other hand, only member-end rotational springs cannot accurately estimate rotations along the member because different non-linear curvature distribution occurs in case of different zero moment points. Moreover, the stiffnessess of the rotational springs are defined by using the assumption that the contraflexure point is at midspan, due to asymmetric moment distribution. However, once yielding occurs at the member-end, the curvature distribution and contraflexure point changes. Hence, midspan assumption is not valid, but it can be used in practical modelling approach, for the sake of simplicity, particularly in the case of low-rise buildings that the contraflexure point of columns or structural walls locates relatively close to midheight (Otani 1980). As mentioned above, inelastic shear strain effect on both the member-end rotation and response of the member cannot be taken into account using one-component beam column element modelling approach.

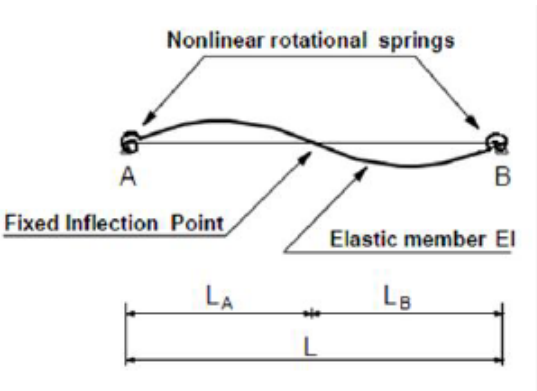


Figure 3.7: The One-Component Model (Fragiadakis M.,2020)

3.3.2 Stiffness Matrix and Internal Forces

As described by Fragiadakis M. (2020), the Giberson (1967) model consists of an elastic beam member and two springs, that their flexibilities are positioned at the beam ends (See Figure 3.8). Specifically, the values of the springs' flexibilities are:

- $f_{s1} = \frac{1}{k_{s1}}$
- $f_{s2} = \frac{1}{k_{s2}}$

Their stiffness is infinite when the beam end is elastic and equal to the inverse of the spring stiffness otherwise.

Figure 3.8: The Series Model's Components (Fragiadakis M.,2020)

Also, the flexibility matrix F_E of the elastic member at the basic coordinate system is:

$$F_E = \begin{bmatrix} \frac{L}{EA} & 0 & 0 \\ 0 & \frac{L}{3EI} & \frac{-L}{6EI} \\ 0 & \frac{-L}{6EI} & \frac{L}{3EI} \end{bmatrix} \tag{3.3.1}$$

Furthermore, the two springs are considered as zero-length elements thus in the basic coordinate system their contribution -for each one of them- can be written:

$$F_{s1} = \begin{bmatrix} 0 & 0 & 0 \\ 00 & f_{s1} & 0 \\ 00 & 0 & 0 \end{bmatrix} \tag{3.3.2}$$

$$F_{s2} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & f_{s2} \end{bmatrix} \tag{3.3.3}$$

And since the three elements are in series, their flexibilities can be added in order to calculate the total flexibility of the element:

$$F_N = F_E + F_{s1} + F_{s2} = \begin{bmatrix} \frac{L}{EA} & 0 & 0 \\ 0 & \frac{L}{3EI} + \frac{1}{k_{s1}} & \frac{-L}{6EI} \\ 0 & \frac{-L}{6EI} & \frac{L}{3EI} + \frac{1}{k_{s2}} \end{bmatrix} \tag{3.3.4}$$

Therefore, the total stiffness of the element is equal to:

$$K_N = (F_N)^{-1} \tag{3.3.5}$$

The properties of the two rotational springs should be determined with cautious in order to describe correctly the behaviour of the real problem. Specifically, when the behaviour is linear elastic, the spring should not affect the stiffness of the member. But when the yielding occurs, the member should have the correct stiffness.

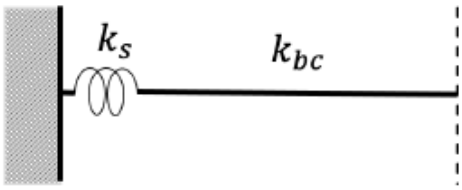


Figure 3.9: A Cantilever Beam with a Rotational Spring (Fragiadakis M.,2020)

Consider the example of simple cantilever beam (See Figure 3.9) for ease of understanding. When the element is linear elastic, the stiffness should be equal to that of the elastic cantilever beam. Therefore, if k_s and k_{bc} is the rotational stiffness of the spring and the beam, respectively, the total stiffness k_m of the structure -the system of beam combined with the spring- will be k_m. And since the spring and the member are connected in series, we obtain:

$$\frac{1}{k_m} = \frac{1}{k_s} + \frac{1}{k_{bc}} \tag{3.3.6}$$

Therefore:

$$k_m = \frac{k_s k_{bc}}{k_s + k_{bc}} \tag{3.3.7}$$

The expression above implies that, when the behaviour is linear elastic, the stiffness of the spring should be infinite so that the rotational stiffness of the member is equal to that of the elastic beam. However, instead of infinite stiffness -for arithmetical and computational

reasons-, we set the stiffness of the spring to a value proportional to the stiffness of the member. For example:

$$k_s = nk_{bc} \tag{3.3.8}$$

where n is an arbitrary positive integer number. Typically, sufficient results can be obtained by setting n = 10 avoiding any possible problems that may arise if very stiff elements are introduced in the solution. Therefore, combining the above equations we obtain:

$$k_{bc} = \frac{n+1}{n} k_m \tag{3.3.9}$$

and

$$k_m = n + 1k_m \tag{3.3.10}$$

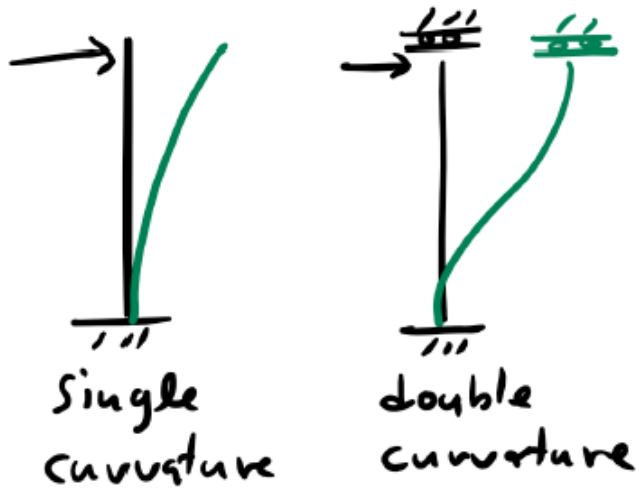


Figure 3.10: Single versus double curvature distribution along an element(Fragiadakis M.,2020)

Another issue that remains is how to define the value of the stiffness of the member k_m. The two extremes is that the member may have a “single” or a “double” curvature (See Figure 3.10). For the two extreme cases the member stiffness value is $k_m = \frac{EI}{L}$ and $k_m = \frac{6EI}{L}$, for single and double curvature, respectively. Therefore, the rotational stiffness of the spring and the member for the cantilever example will be:

$$k_s = \frac{(n+1)3EI}{L}, \text{ for single curvature} \tag{3.3.11}$$

and

$$k_s = \frac{(n+1)6EI}{L}, \text{ for double curvature} \tag{3.3.12}$$

The above equations are not restricted to the cantilever beam example though. They can be used when the member has a spring at each of its members. As a result, we obtain:

$$k_s = \frac{(n+2)3EI}{L}, \text{ for single curvature} \tag{3.3.13}$$

and

$$k_s = \frac{(n+2)6EI}{L}, \text{ for double curvature} \tag{3.3.14}$$

However, since n is arbitrary, there is not practical difference between the two equations derived. Hence, for simplicity we can use the first equation for all the cases.

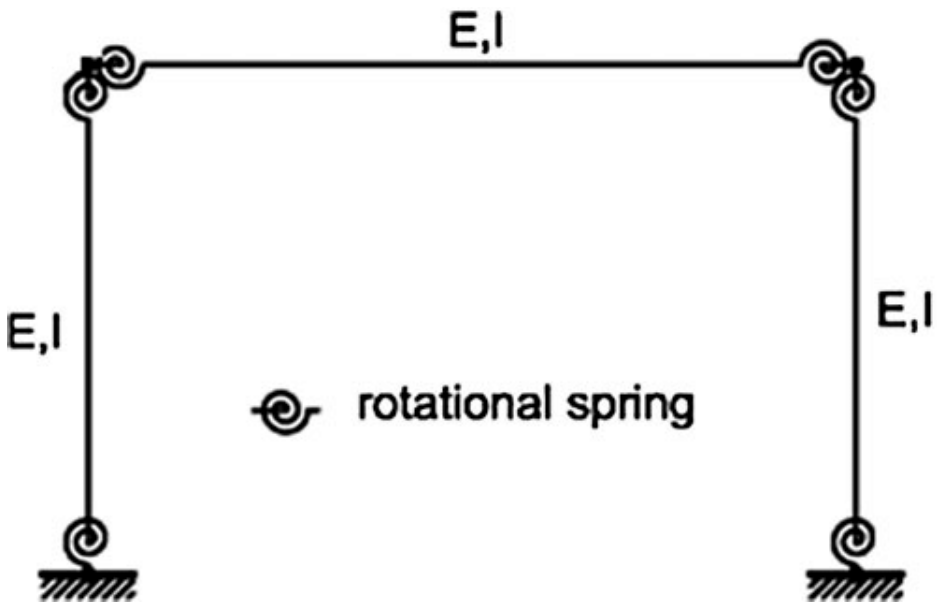


Figure 3.11: Lumped Plasticity Frame

As a result, due to the simple rules that governs the behaviour of the one-component model, its high accuracy and its adapt-fullness to various moment-curvature relationships, the goal of this thesis is the development of a tool -using MATLAB programming language- which will analyse structures composed of beam elements -such as the one shown in Figure 3.11- using the one component lumped plasticity theory as proposed by Giberson (1967) under monotonic loading.

CHAPTER 4. EXAMPLES USING THE LUMPED PLASTICITY BEAM ELEMENT

4.1 Comparison with OpenSees

The open source software OpenSees of the University of Berkeley was selected for the verification that the code developed for the beam element with lumped plasticity -and that was integrated in the software ASDAP (Femlab) of the National Technical university of Athens- is correct.

4.1.1 Cantilever Beam

The analysis for the cantilever beam with lumped plasticity was done using the following inputs for both softwares:

- Length of Beam: $L = 3m$
- Modulus of Elasticity: $E = 30GP_a$
- Area of Cross Section: $A = 0.0929m^2$
- Moment of Inertia: $I = 0.0012786m^4$
- Yield Moment for the Springs -that were placed at Node 1 and Node 2-: $M_y = 160.1kNm$
- Hardening for the Springs Used: $b = 0.001$
- Applied Vertical Load at Node 2: $P = 70kN$
- For Newton-Raphson: $Tolerance = 0.001$, $LoadIncrements = 100$ and $InternalIterations = 1000$

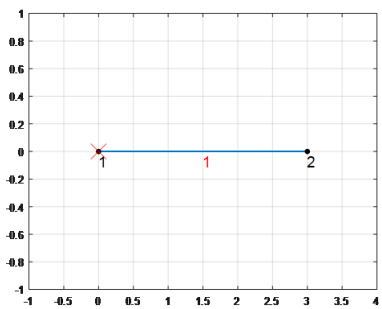


Figure 4.1: The Cantilever Beam from ASDAP (Femlab)

In Figures 4.2 and 4.3, the comparison of the imposed load at Node 2 and its equivalent displacement are presented i.e the P-U curve. It is apparent that the code developed and Opensees produce the same results.

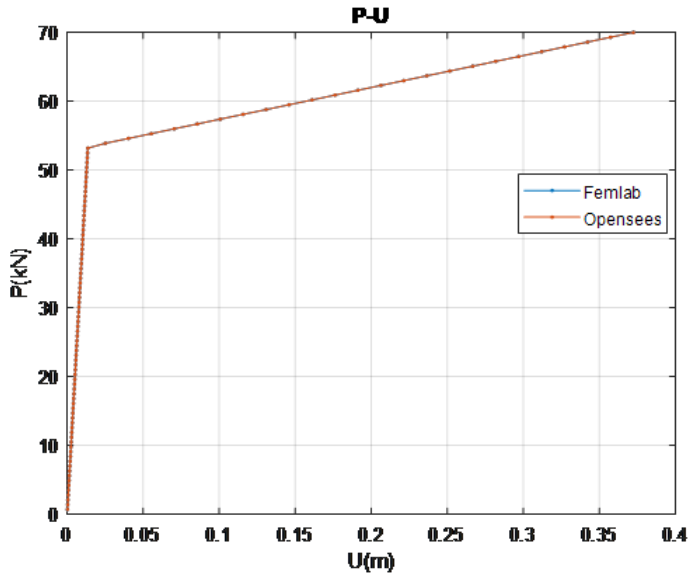


Figure 4.2: The Comparison of the P-U Curve between ASDAP (Femlab) and Opensees for the Cantilever Beam

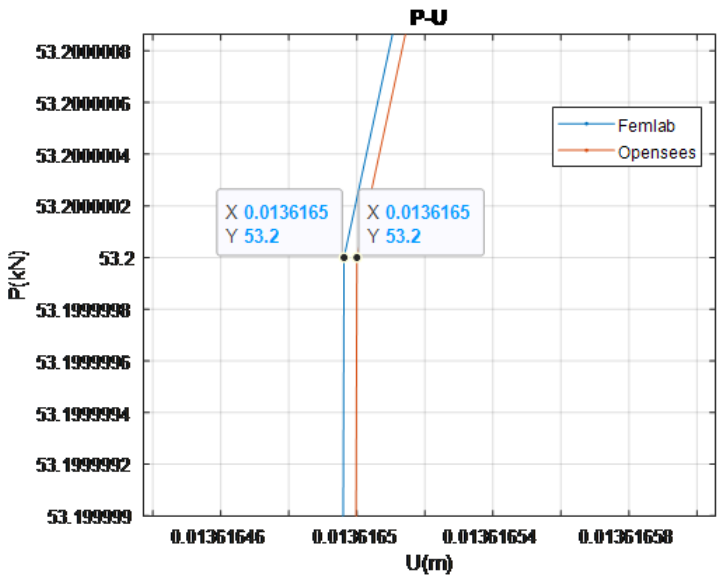


Figure 4.3: The Comparison of the P-U Curve between ASDAP (Femlab) and Opensees for the Cantilever Beam (Zoomed In)

4.1.2 Symmetric Clamped Beam

The analysis for the symmetric clamped beam with lumped plasticity was done using the following inputs for both softwares:

- Length of Beams: $L = 3m$
- Modulus of Elasticity: $E = 30GP_a$
- Area of Cross Sections: $A = 0.0929m^2$
- Moment of Inertia: $I = 0.0012786m^4$
- Yield Moment for the Springs -that were placed at Node 1, Node 2 and Node 3-: $M_y = 160.1kNm$
- Hardening for the Springs Used: $b = 0.00000001$
- Applied Vertical Load at Node 2: $P = 105kN$
- For Newton-Raphson: $Tolerance = 0.001$, $LoadIncrements = 100$ and $InternalIterations = 1000$

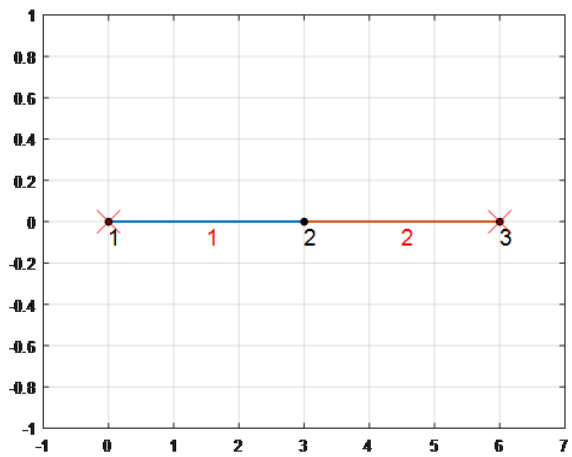


Figure 4.4: The Symmetric Clamped Beam from ASDAP (Femlab)

In Figures 4.5 and 4.6, the comparison of the imposed load at Node 2 and its equivalent displacement are presented i.e the P-U curve. It is apparent that the code developed and Opensees produce the same results.

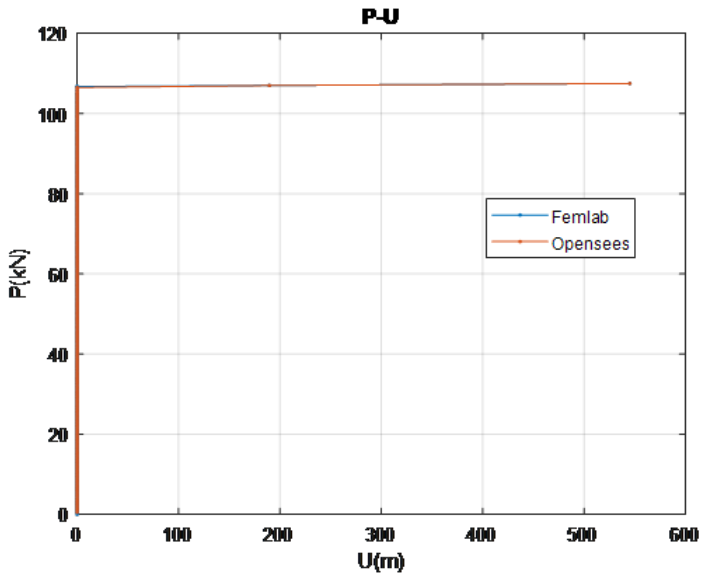


Figure 4.5: The Comparison of the P-U Curve between ASDAP (Femlab) and Opensees for the Symmetric Clamped Beam

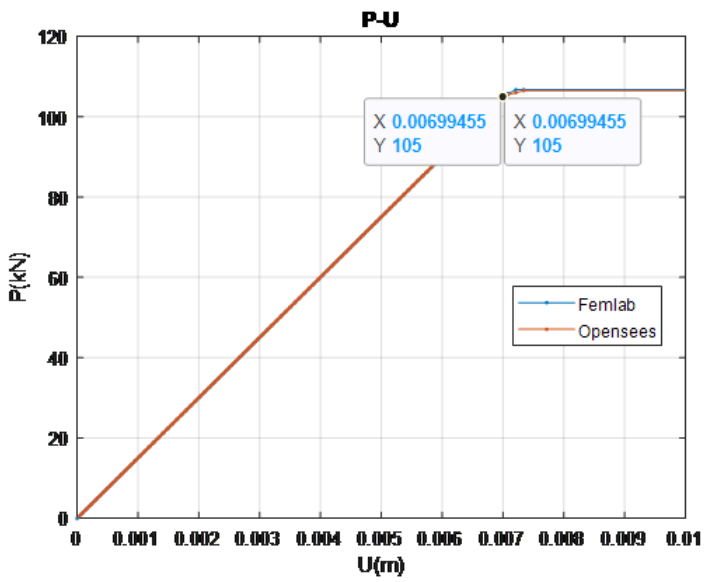


Figure 4.6: The Comparison of the P-U Curve between ASDAP (Femlab) and Opensees for the Symmetric Clamped Beam (Zoomed In)

4.1.3 Not-Symmetric Clamped Beam

The analysis for the not-symmetric clamped beam with lumped plasticity was done using the following inputs for both softwares:

- Length of Beams: $L_1 = 1.5m$ and $L_2 = 4.5m$
- Modulus of Elasticity: $E = 30GP_a$
- Area of Cross Sections: $A = 0.0929m^2$
- Moment of Inertia: $I = 0.0012786m^4$
- Yield Moment for the Springs that were placed at Node 1 and Node 3: $M_y = 160.1kNm$
- Yield Moment for the Spring that was placed at Node 2: $M_y = 85.8kNm$
- Hardening for the Springs Used: $b = 0.01$
- Applied Vertical Load at Node 2: $P = 230kN$
- For Newton-Raphson: $Tolerance = 0.001$, $LoadIncrements = 100$ and $InternalIterations = 1000$

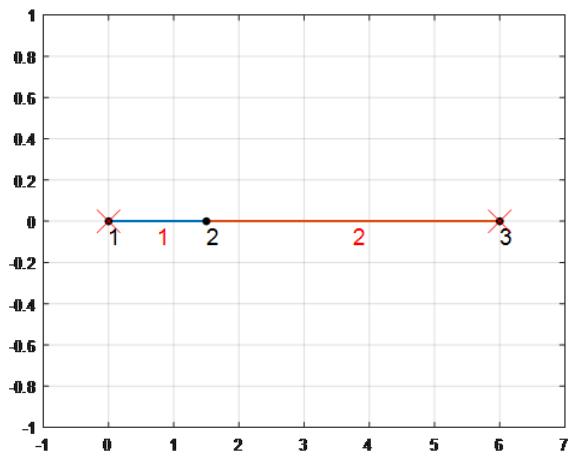


Figure 4.7: The Not-Symmetric Clamped Beam from ASDAP (Femlab)

In Figures 4.8 and 4.9, the comparison of the imposed load at Node 2 and its equivalent displacement are presented i.e the P-U curve. It is apparent that the code developed and Opensees produce the same results.

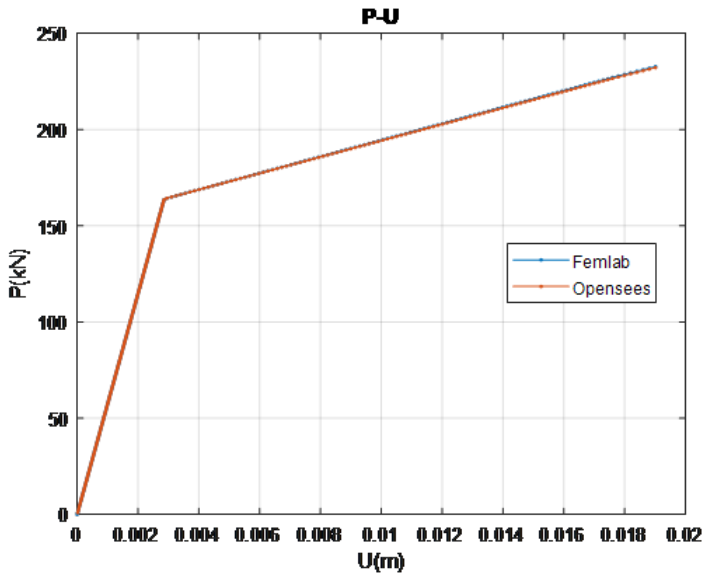


Figure 4.8: The Comparison of the P-U Curve between ASDAP (Femlab) and Opensees for the Not-Symmetric Clamped Beam

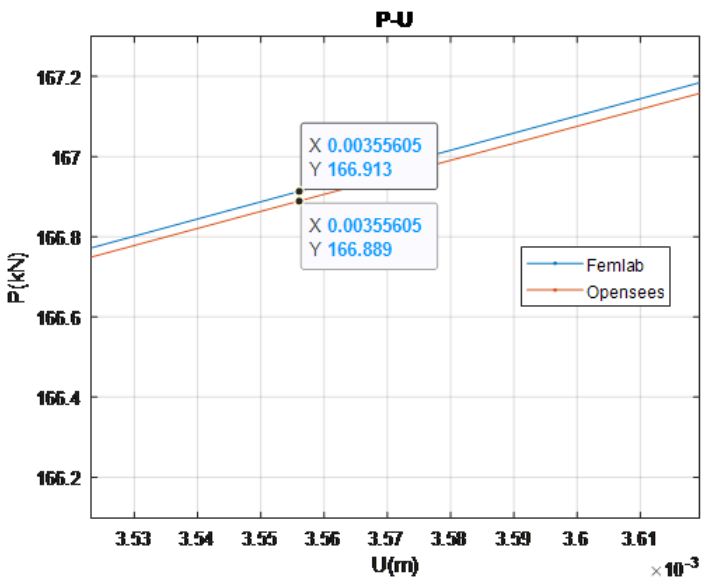


Figure 4.9: The Comparison of the P-U Curve between ASDAP (Femlab) and Opensees for the Not-Symmetric Clamped Beam (Zoomed In)

4.2 Comparison with Experimental Results

In this section, two frame models -one with elastic beam elements and one with beam elements with lumped plasticity- are compared to the experimental behaviour of two frame structures in order to make evident that taking account of the non-linear behaviour of structures is vital for their safe and cost efficient design.

4.2.1 Experiment of Arslan M.E.(2013)

The bare frame specimen PRCF-1 -that was experimentally examined by Arslan M.E. (2013) under lateral loading- was selected to be analysed in the present thesis.

Figure 4.10: The Test Specimen PRCF-1 (Arslan M.E., 2013)

As described by Timurağaoğlu et al. (2016), the specimen PRCF-1 is a one-storey one-bay RC frame that has no infill wall. More specifically, the frame was designed in accordance with the provisions of Turkish Earthquake Code. The test specimen was chosen to be 1/1 scale. The design details for the frame specimen are shown in Figure 4.11. The columns and beam were selected to be $0.2x0.25$ m and $0.25x0.2$ m, respectively. For the base of the frame, $4x0.6x0.4$ m dimensions were selected. The base is fixed to the ground with shear connectors. The loads are applied to the system by increasing the amplitude in each cycle as shown in Figure 4.11. A minimum reinforcement of $6\phi14$ is used for columns while a reinforcement of $3\phi12$ for bottom and $2\phi12$ for top of beam is selected as shown in Figure 4.11. Confinement reinforcement, which is required by Turkish Earthquake Code, is used both along the columns and beam. The details of confinement in columns and beam is shown in Figure 4.11. The compressive strength and elasticity modulus of concrete are defined as $25MP_a$ and $28000MP_a$.

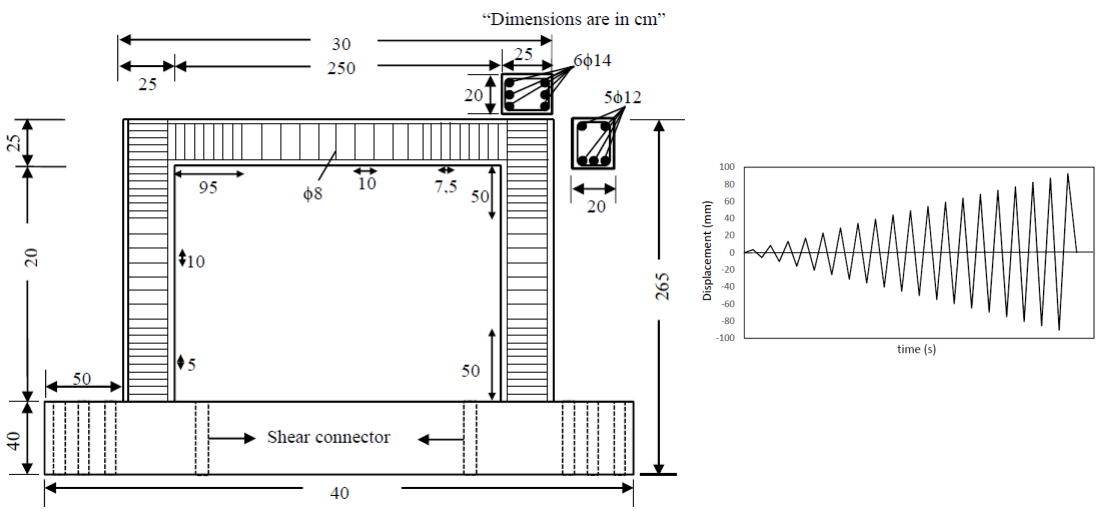


Figure 4.11: Dimensions-reinforcement Details and Displacement Amplitude Used in Laboratory Test for Specimen PRCF-1(Timurağaoğlu et al., 2016)

The hysteretic response of the frame for the imposed amplitude shown in Figure 4.11 and its equivalent backbone curve is shown in Figure 4.12:

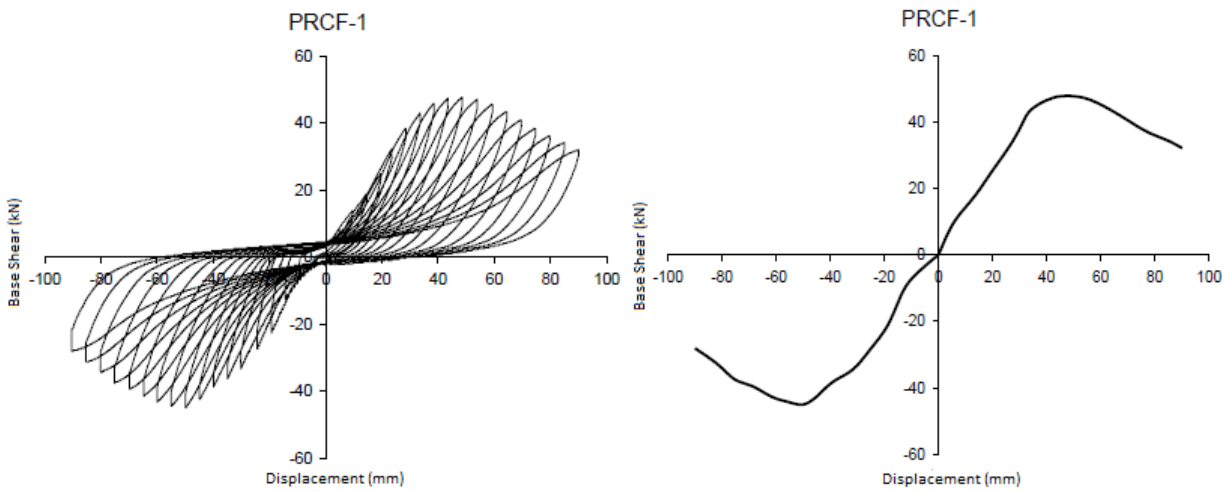


Figure 4.12: The Hysteretic Response and its Equivalent Backbone Curve for Specimen PRCF-1(Arslan M.E., 2013)

The failure types occurred in test specimen PRCF-1 -after imposing the lateral load- are shown in Figure 4.13. Failures in the frame generally occurred as plastic hinges in the columns base and column-beam joints. As a result, a first conclusion is that the beam element with lumped plasticity that consider springs in the members' end, will probably be able to simulate the behaviour of this frame.

Figure 4.13: The Experimental Failure Types (Arslan M.E., 2013)

The analysis for the model with elastic beam elements was done using the following inputs:

- Beam's Length, $L = 2.5m$
- Columns' Height,$H = 2.25m$
- Modulus of Elasticity, $E = 28GP_a$
- Area of Beam's Cross Section, $A = 0.05m^2$
- Area of Columns' Cross Section, $A = 0.05m^2$
- Moment of Inertia for the Beam, $I = 0.000167m^4$
- Moment of Inertia for Columns, $I = 0.00026m^4$
- Applied Horizontal Load at Node 2 according to the Experiment: $P = 98kN$

Whereas the analysis for the model with lumped plasticity beam elements was done using the following inputs:

- Beam's Length, $L = 2.5m$
- Columns' Height,$H = 2.25m$
- Modulus of Elasticity, $E = 28GP_a$

- Area of Beam's Cross Section, $A = 0.05m^2$
- Area of Columns' Cross Section, $A = 0.05m^2$
- Moment of Inertia for the Beam, $I = 0.000167m^4$
- Moment of Inertia for Columns, $I = 0.00026m^4$
- Yield Moment for the Beam and the Top of the Columns, $M_y = 42kNm$
- Yield Moment for the Bottom of the Columns, $M_y = 65kNm$
- Hardening for the Beam and Columns, $b = 0.000000000001$
- Applied Horizontal Load at Node 2 according to the Experiment: $P = 98kN$

Therefore, the comparison between the two models and the experiment is shown in the Figure 4.14, where it is obvious that the model with lumped plasticity beam elements can describe the physical problem with higher accuracy.

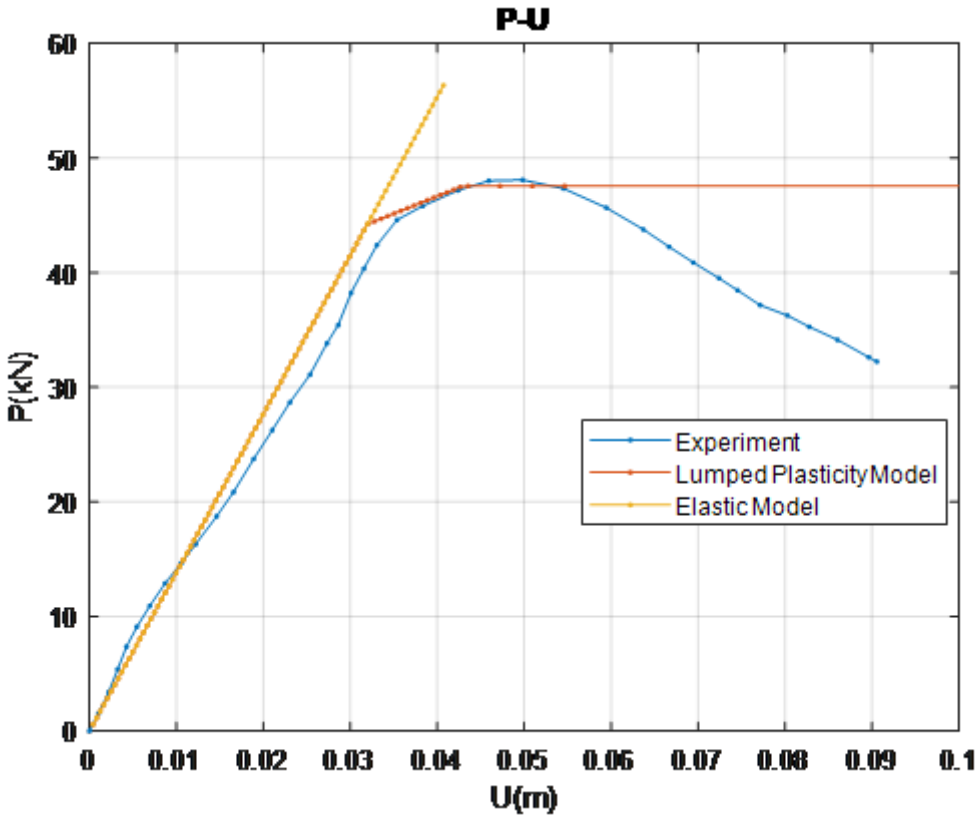


Figure 4.14: The Comparison between the Model with Elastic Beam Elements, the Model with Lumped Plasticity Beam Elements and the Experimental Backbone Curve

4.2.2 Experiment of Akgüzel U. (2003)

The bare frame specimen U1 -that was experimentally examined by Akgüzel U. (2013) under lateral loading- was selected to be analysed in the present thesis.

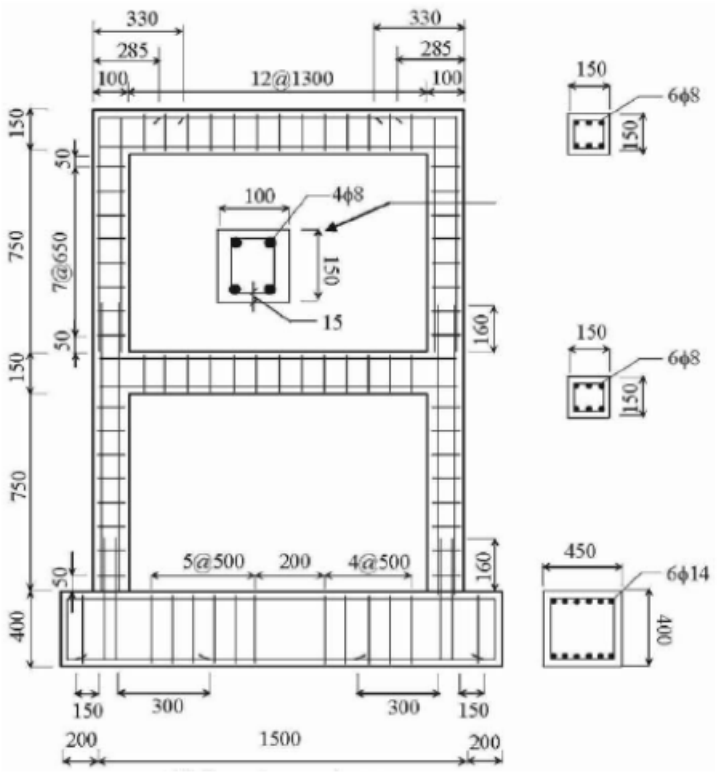


Figure 4.15: Dimensions-reinforcement Details Used in Laboratory Test for Specimen U1(Spyrakos et al., 2012)

As described by Spyrakos et al. (2012), the specimen U1 is a two-storey, one-bay plane frame with a $0.90m$ typical storey height and $1.50m$ typical bay length as shown in Fig. 4.15. Orthogonal sections of $100mmx150mm$ and $150mmx150mm$ were used for the columns and the beams, respectively with the small dimension of the columns' section being in-plane with the frame. The compressive strength of the concrete had an average value of $f_c = 15.4M_{Pa}$. Typical longitudinal reinforcement of $4\phi8$ and $6\phi8$, with yield strength $f_y = 380M_{Pa}$, was used for the columns and the beams, respectively. Transverse reinforcement of $\phi4/100mm$, with yield strength $f_{wy} = 241M_{Pa}$, was used for the columns and the beams. An insufficient lap splice length of $160mm$ was provided at each storey base. The concrete cover was $15mm$. Cyclic loading was applied to the specimen in two phases; firstly, the frame was pushed under lateral cyclic forces with a triangular pattern along its height, until the first yield occurred; then, a cyclic loading phase was following till failure.

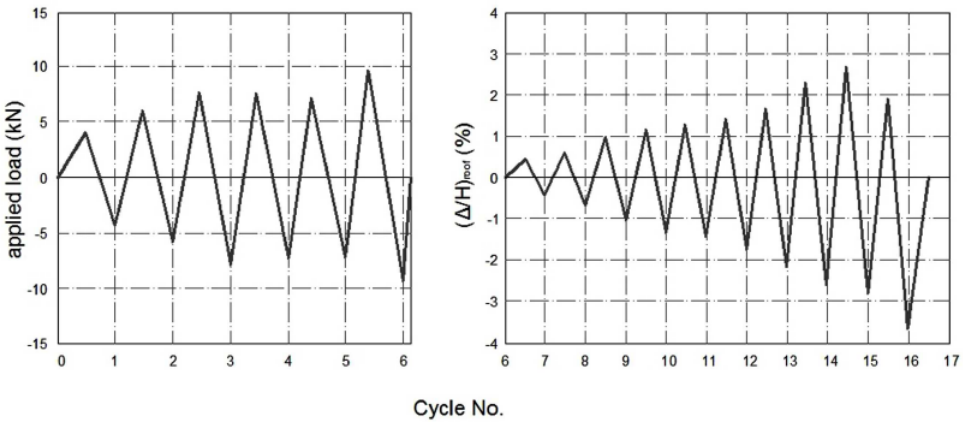


Figure 4.16: The Loading History for Cyclic Analysis of the Bare Frame U1 (Spyrakos et al., 2012)

The hysteretic response of the frame for the imposed amplitude shown in Figure 4.16 and its equivalent backbone curve is shown in Figure 4.17:

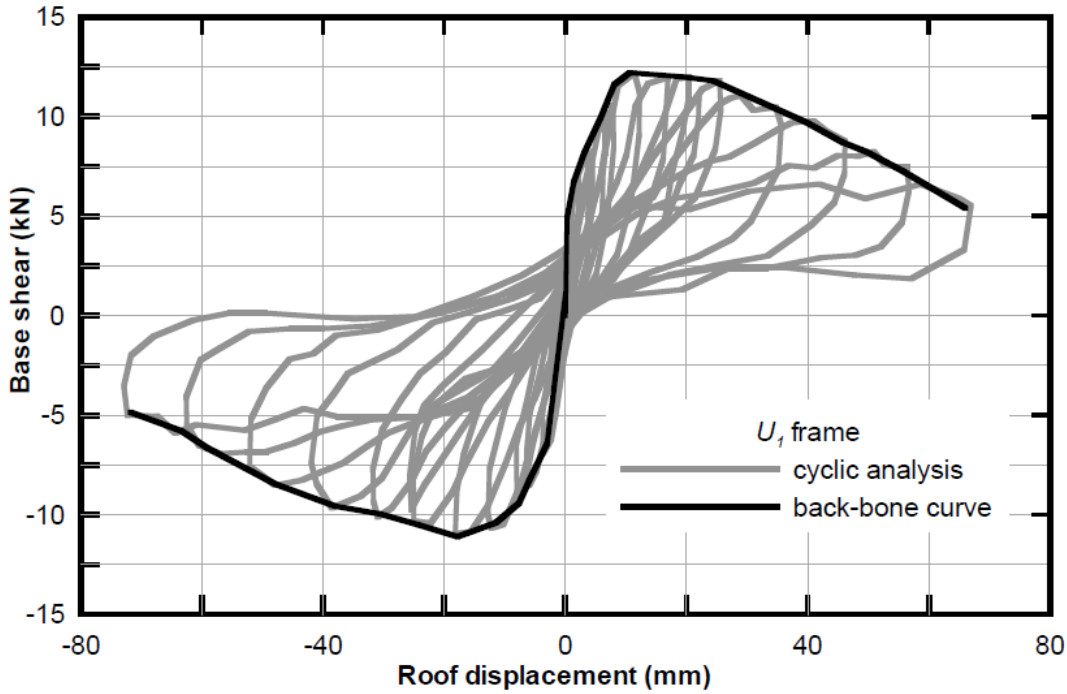


Figure 4.17: The Hysteretic Response and its Equivalent Backbone Curve for Specimen U1 (Spyrakos et al., 2012)

The failure types occurred in test specimen U1 -after imposing the lateral load- are shown in Figure 4.18. Failures in the frame generally occurred as plastic hinges in the columns base and column-beam joints.

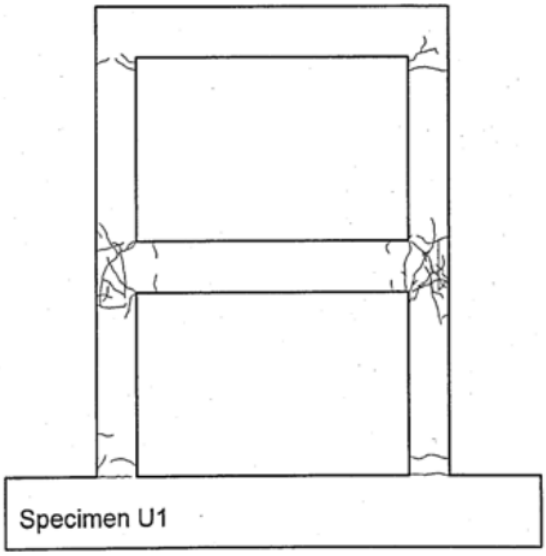


Figure 4.18: The Experimental Failure Types (Spyrakos et al., 2012)

The analysis for the model with elastic beam elements was done using the following inputs:

- Beam's Length, $L = 1.5m$
- Columns' Height,$H = 0.9m$
- Modulus of Elasticity, $E = 30GP_a$
- Area of Beam's Cross Section, $A = 0.0225m^2$
- Area of Columns' Cross Section, $A = 0.015m^2$
- Moment of Inertia for the Beam, $I = 0.0000421m^4$
- Moment of Inertia for Columns, $I = 0.0000125m^4$
- Applied Horizontal Load at Node 5 according to the Experiment: $P = 22kN$

Whereas the analysis for the model with lumped plasticity beam elements was done using the following inputs:

- Beam's Length, $L = 1.5m$
- Columns' Height,$H = 0.9m$
- Modulus of Elasticity, $E = 30GP_a$
- Area of Beam's Cross Section, $A = 0.0225m^2$
- Area of Columns' Cross Section, $A = 0.015m^2$

- Moment of Inertia for the Beam, $I = 0.0000421m^4$
- Moment of Inertia for Columns, $I = 0.0000125m^4$
- Yield Moment for the Beam and the Top of the Columns, $M_y = 7kNm$
- Yield Moment for the Bottom of the Columns, $M_y = 7.5kNm$
- Hardening for the Beam and Columns, $b = 0.0000001$
- Applied Horizontal Load at Node 5 according to the Experiment: $P = 22kN$

Therefore, the comparison between the two models and the experiment is shown in the Figure 4.19, where it is obvious that the model with lumped plasticity beam elements can describe the physical problem with higher accuracy.

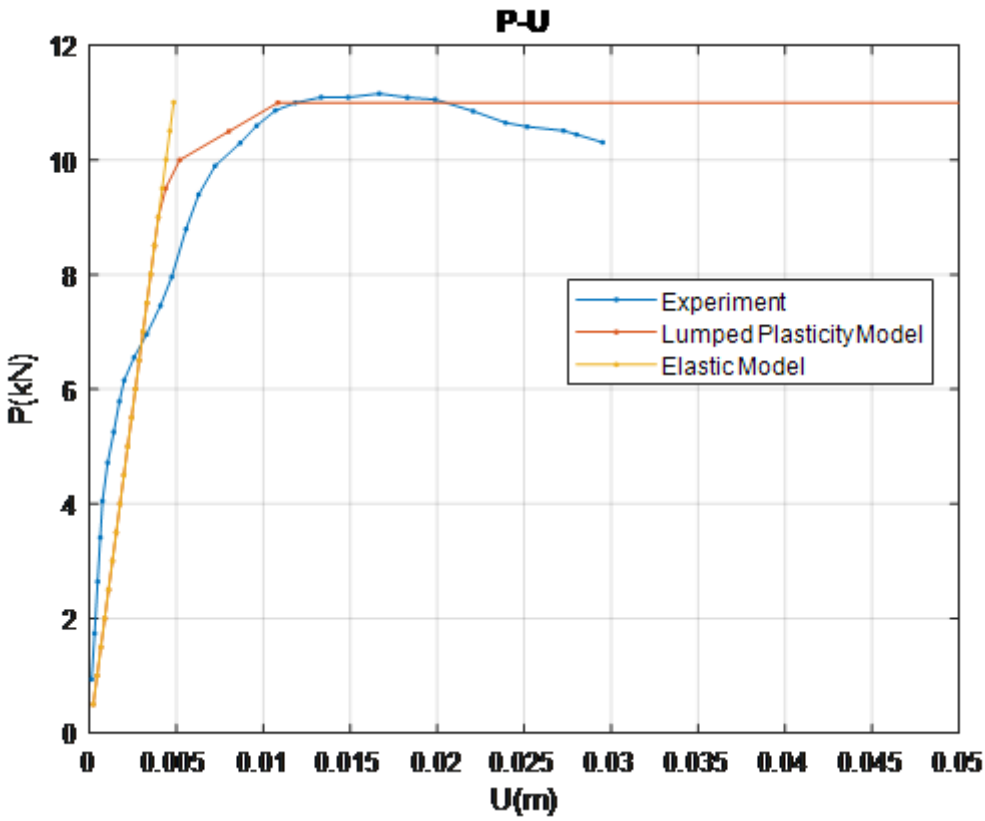


Figure 4.19: The Comparison between the Model with Elastic Beam Elements, the Model with Lumped Plasticity Beam Elements and the Experimental Backbone Curve

4.3 Simulations of Frame Structures

4.3.1 Simulation of a Single Storey Frame

The analysis for the single storey frame with lumped plasticity was done using the following inputs:

- Length of Beams: $L = H = 3m$
- Modulus of Elasticity: $E = 30GP_a$
- Area of Cross Section: $A = 0.0929m^2$
- Moment of Inertia: $I = 0.0012786m^4$
- Yield Moment for the Springs: $M_y = 160.1kNm$
- Hardening for the Springs Used: $b = 0.001$
- Applied Horizontal (Seismic) Load at Node 2: $P = 240kN$
- For Newton-Raphson: $Tolerance = 0.001$, $LoadIncrements = 100$ and $InternalIterations = 1000$

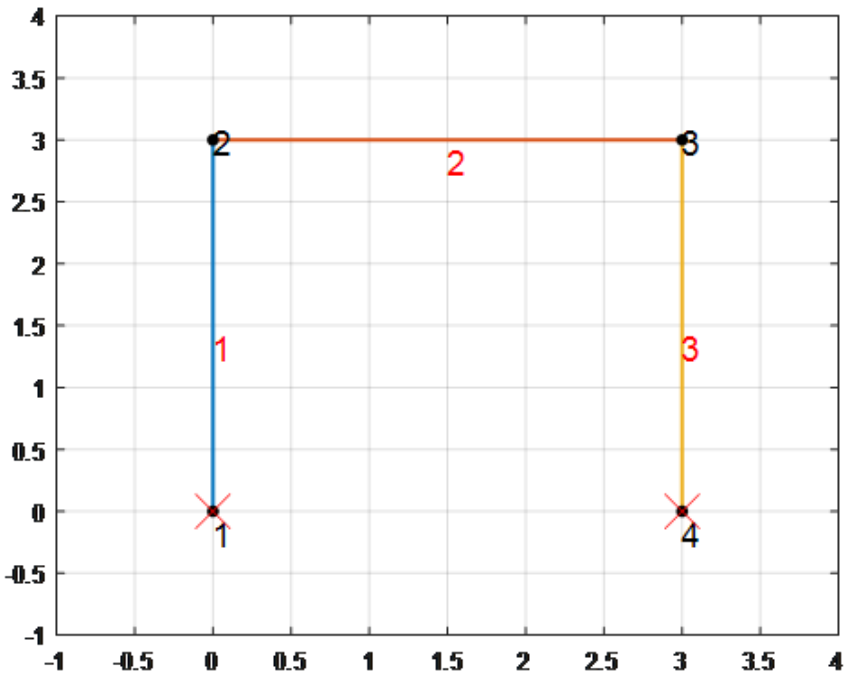


Figure 4.20: The Single Storey Frame from ASDAP (Femlab)

In Figure 4.21, the imposed load at Node 2 and its equivalent displacement are presented i.e the P-U curve.

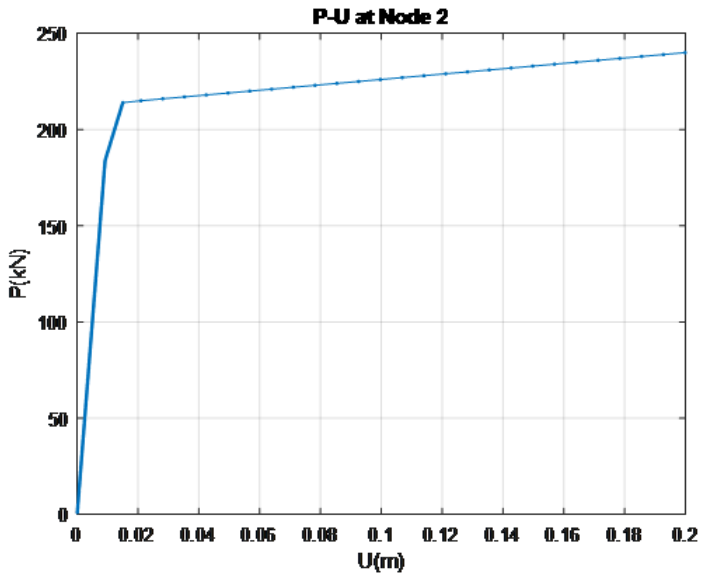


Figure 4.21: The P-U Curve for the Single Storey Frame

4.3.2 Simulation of a Two Storey Frame

The analysis of a two storey frame with lumped plasticity was done using the following inputs:

- Length of Beams: $L = H = 3m$
- Modulus of Elasticity: $E = 30GP_a$
- Area of Cross Sections: $A = 0.0929m^2$
- Moment of Inertia: $I = 0.0012786m^4$
- Yield Moment for the Springs: $M_y = 160.1kN$
- Hardening for the Springs Used: $b = 0.001$
- Applied Horizontal (Seismic) Load at Node 2 and 6 respectively: $P_{node2} = 0.333x240 = 79.92kN$ and $P_{node6} = 0.667x240 = 160.08kN$
- For Newton-Raphson: $Tolerance = 0.001$, $LoadIncrements = 100$ and $InternalIterations = 1000$

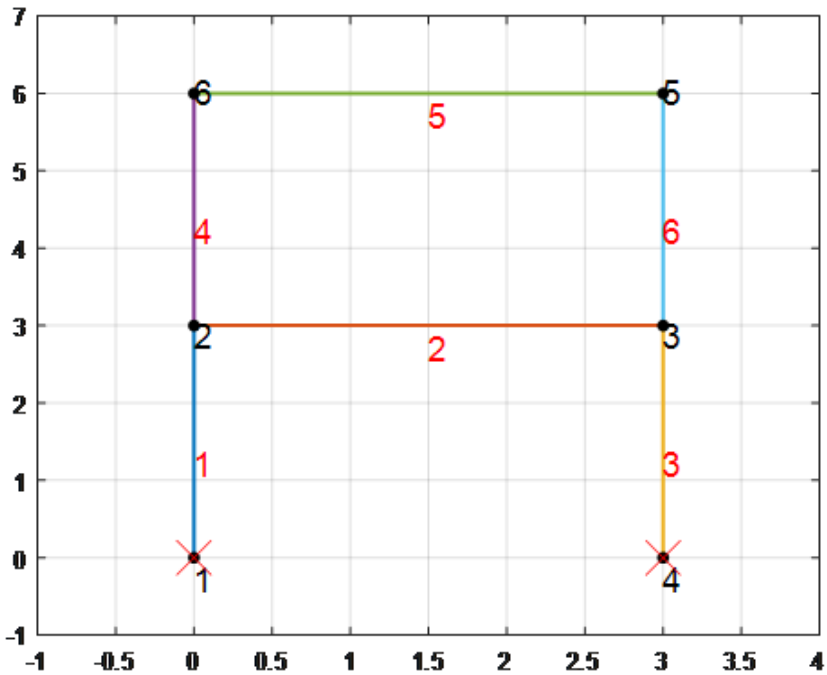


Figure 4.22: The Two Storey Frame from ASDAP (Femlab)

In Figures 4.23 and 4.24, the imposed load at Node 2 and Node 6 and their equivalent displacements are presented i.e the P-U curves.

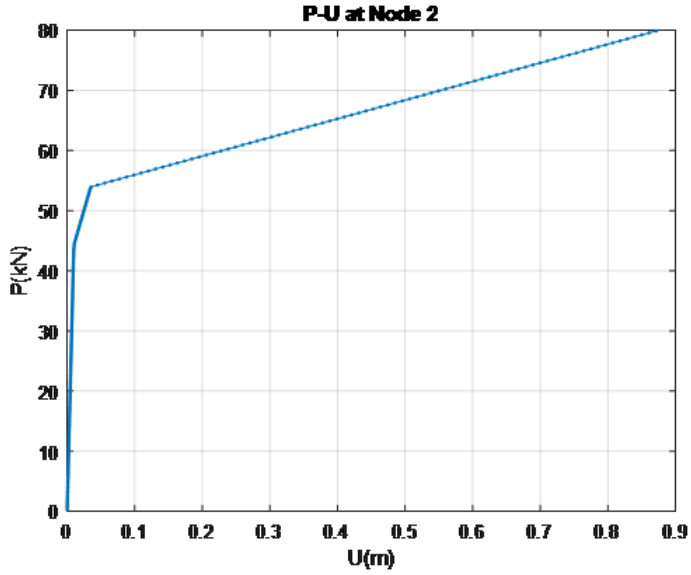


Figure 4.23: The P-U Curve at Node 2 for the Two Storey Frame

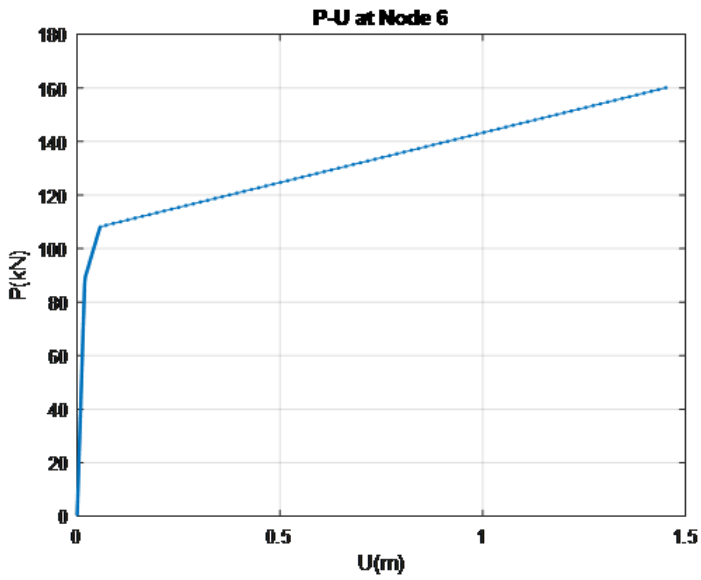


Figure 4.24: The P-U Curve at Node 6 for the Two Storey Frame

4.3.3 Simulation of a Three Storey Frame

The analysis of a three storey frame with lumped plasticity was done using the following inputs:

- Length of Beams: $L = H = 3m$
- Modulus of Elasticity: $E = 30GP_a$
- Area of Cross Sections: $A = 0.0929m^2$
- Moment of Inertia: $I = 0.0012786m^4$
- Yield Moment for the Springs: $M_y = 160.1kN$
- Hardening for the Springs Used: $b = 0.01$
- Applied Horizontal (Seismic) Load at Node 2, 5 and 7 respectively: $P_{node2} = 0.4771x240 = 114.5kN$, $P_{node5} = 0.8396x240 = 201.5kN$ and $P_{node7} = 1.0x240 = 240kN$
- For Newton-Raphson: $Tolerance = 0.001$, $LoadIncrements = 100$ and $InternalIterations = 1000$

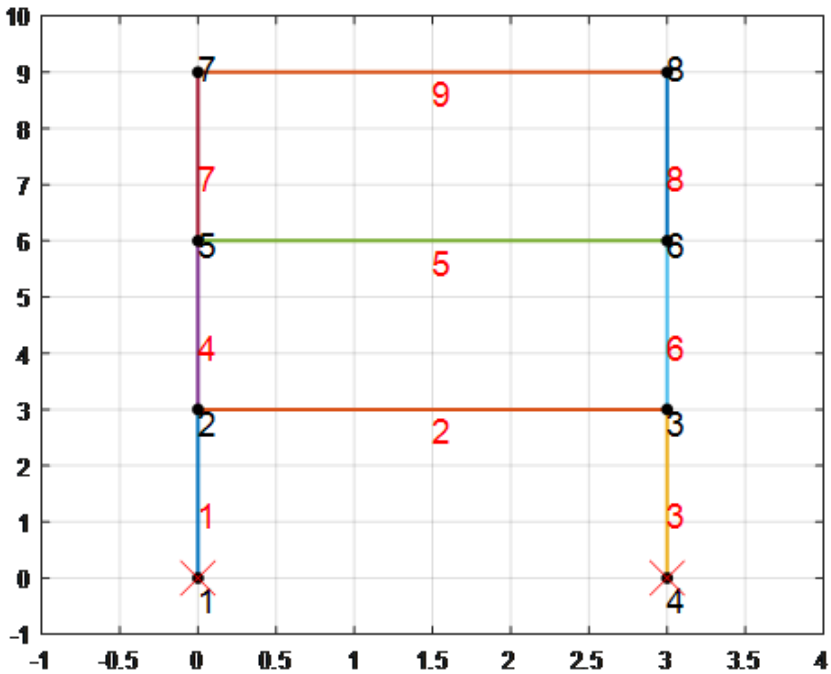


Figure 4.25: The Three Storey Frame from ASDAP (Femlab)

In Figures 4.26, 4.27 and 4.28, the imposed load at Node 2, 5 and Node 7 and their equivalent displacements are presented i.e the P-U curves.

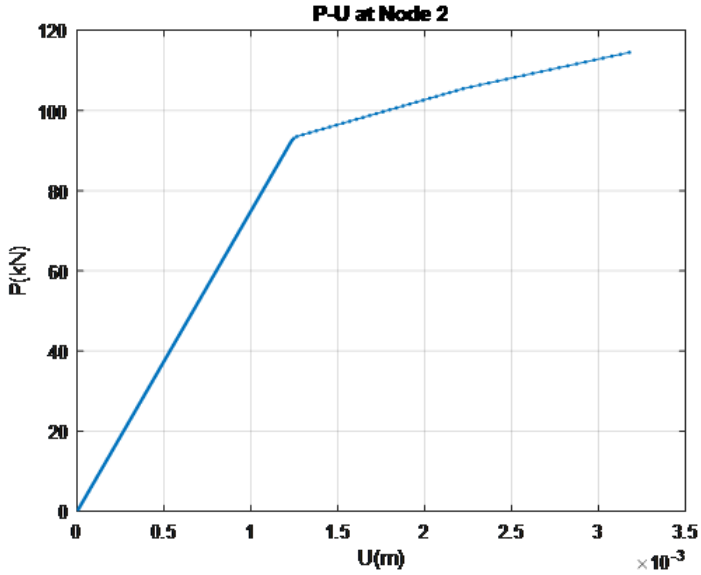


Figure 4.26: The P-U Curve at Node 2 for the Three Storey Frame

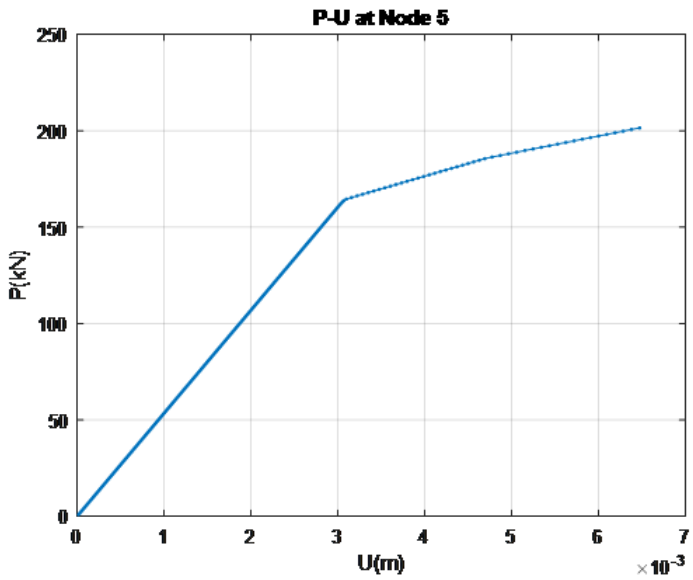


Figure 4.27: The P-U Curve at Node 5 for the Three Storey Frame

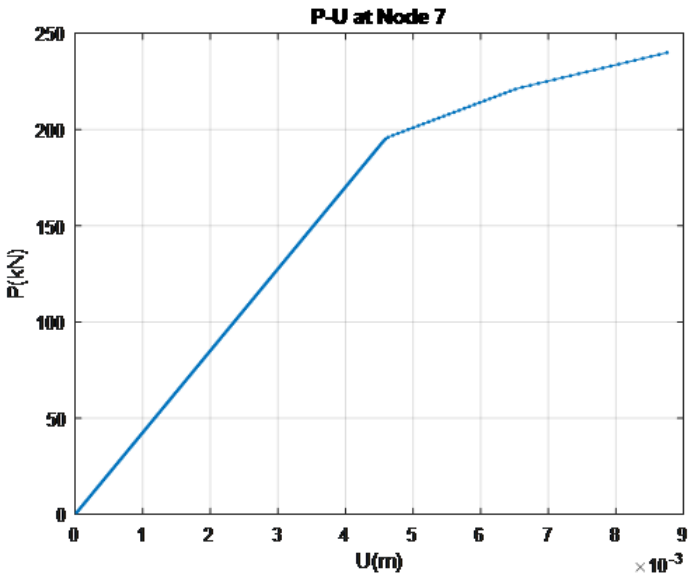


Figure 4.28: The P-U Curve at Node 7 for the Three Storey Frame

Therefore, it is evident that the simulation of a frame structure using the lumped plasticity element is quite simple and has high accuracy. However, it is noted that the definition of its parameters -such as modulus of elasticity, hardening etc.- should be done carefully in order to produce reliable results and to lead to efficient design.

CHAPTER 5. CONCLUSIONS AND SUGGESTIONS

5.1 Conclusions

After the literature review and the analyses undertaken for the present postgraduate thesis the main conclusions are presented. Firstly, it is highlighted that when designing a structure, the most important task is the understanding of its static behaviour and the correct definition of its properties in order to select the right model and type of analysis to simulate it. Secondly, it is crucial to take into account the elasto - plastic behaviour of the main structural materials -i.e. reinforced concrete and steel- when analysing a structure in order to design it safe and cost efficiently. More specifically, as the recent research work suggests the linear-elastic analysis is not always the safer case, due to the uncertainties of the post-yield behaviour of its members and subsequently of the entire structure. That is the reason why a non-linear analysis should be preferred or at least investigated when designing a structure. Finally, it is evident that the lumped plasticity theory is a simple, precise and computationally efficient way to take into account the plastic behaviour of structures as concluded from the comparison of the models composed of lumped plasticity elements with the equivalent experimental results. Therefore, the usage of lumped plasticity elements is highly suggested for structures that can be simulated with the beam theory and that are subjected to lateral loads -in order to experience the largest moments at end regions of members.

5.2 Suggestions

The main suggestions for future research work are briefly summarized in this section. Firstly, it is suggested a more in depth investigation of the Newton-Raphson method and its alternatives, due to the importance of the iterative algorithm for finding the solution of non-linear problems. More specifically, it shown from the literature review that the Force-Control NR version -that was used for the present thesis- does not converge for certain problems, thus the use of Displacement Control or Arc-Length versions are essential in order to find the correct solution. Secondly, it is suggested to further investigate the behaviour of structures under seismic loading using dynamic analysis methods, because the non-linear static analysis (pushover) method is a simplified approach and might not predict all the phenomena that appear during a cycling loading. Thirdly, it is suggested to investigate the usage of different moment-curvature relationship for the springs used to simulate the plastic behaviour of the lumped plasticity beam element, in order to depict more precisely the behaviour of each material and consequently of the structure. Finally, it is suggested to examine the distributed plasticity theory and compare it with the lumped plasticity theory, in order to determine which is better suited for each type of structure and loading case.

CHAPTER 6. BIBLIOGRAPHY

Akgüzel U. (2003). *Seismic Retrofit of brick infilled R/C frames with lap splice problem in columns*, M.Sc. thesis, Department of Civil Engineering, Bogazici University, Istanbul, Turkey.

Arslan M.E. (2013). *Deprem kumaşıyla güçlendirilen çevrimsel yük etkisindeki dolgu duvarlı betonarme çerçevelerin davranışlarının deneysel ve teorik olarak incelenmesi*, Ph.D. Dissertation, Karadeniz Teknik Üniversitesi, Trabzon (In Turkish).

Clough R.W., Johnston S.B. (1966). *Effect of stiffness degradation on earthquake ductility requirements*, In Transactions of Japan earthquake engineering symposium, Tokyo, pp. 195-1988.

Dides M.A., de la Liera J.C., (2005). *A comparative study of concentrated plasticity models in dynamic analysis of building structures*, Earthquake Eng. Struct. Dyn., 34(8), pp. 1005-1026.

Filippou F.C., Issa A. (1998). *Nonlinear analysis of reinforced concrete frames under cyclic load reversals*, Report No. UBC/EERC-88/12, Earthquake Engineering Research Center, University of California, Berkeley, CA.

Fragiadakis M. (2020). *Nonlinear Analysis of Frame Structures under Seismic Loading*, Lecture Notes, National Technical University of Athens.

Gharakhanloo A. (2014). *Distributed and Concentrated Inelasticity Beam-Column Elements used in Earthquake Engineering*, Dissertation, Norwegian University of Science and Technology, Department of Structural Engineering.

Giberson M.F. (1967). *The response of nonlinear multi-story structures subjected to earthquake excitation*, Ph.D. Dissertation, California Institute of Technology, Pasadena, CA.

Giberson M.F. (1969). *Two nonlinear beams with definitions of ductility*, J Struct. Div., 95(ST2), pp. 137-157.

Al-Haddad M.S.,Wight J.K. (1986) *Feasibility and consequences of moving beam plastic hinging zones for earthquake resistant design of R/C buildings*, Report No. UMCE 861. The University of Michigan, Ann Arbor, Michigan.

Hajjar J.F., Gourely B.C. (1997). *A cyclic nonlinear model for concrete-filled tubes. 1. Formulation*, J Struct. Eng., 123(6), pp. 736–744.

Heng P. (2017). *Simplified mechanical models for the nonlinear dynamic analysis of elasto-plastic steel structures impacted by a rigid body.*, Ph.D. Dissertation, Department of Civil and Architectural Engineering, at KTH Royal Institute of Technology (Sweden).

Ibarra L.F., Krawinkler H. (2005). *Global Collapse of Frame Structures under Seismic Excitations*, John A. Blume Earthquake Engineering Center Technical Report 152. Stanford Digital Repository.

Kappos A. (1986). *Διερεύνηση της ανελαστικής σεισμικής συμπεριφοράς πολυόροφων*

κτιρίων από οπλισμένο σκυρόδεμα, Ph.D. Dissertation, Aristotle University of Thessaloniki (in Greek).

Kunnath S.K., Reinhorn A.M. (1989). *Inelastic three-dimensional response analysis of reinforced concrete structures subjected to seismic loads*, Technical Report No. NCEER-88-0041, University at Buffalo, The State University of New York.

Markou G. (2011). *Detailed Three-Dimensional Nonlinear Hybrid Simulation for the Analysis of Large-Scale Reinforced Concrete Structures*, Ph.D. Dissertation, National Technical University of Athens.

Mohsen R.H. (2018). *Nonlinear Analysis of Reinforced Concrete Frames: Safety Evaluation and Retrofitting Techniques*, Ph.D. Dissertation, Università degli Studi di Palermo.

Otani S. (1980). *Nonlinear dynamic analysis of reinforced concrete building structures*, Can J Civil Eng., 7(2), pp. 333-344.

Park Y.J., Reinhorn A.M., Kunnath S.K. (1987) *IDARC: Inelastic damage analysis of reinforced concrete frame-shear wall structures*, Technical Report No. NCEER- 87-0008, University at Buffalo, The State University of New York.

Repousis N. (2019). *Μη-Γραμμικά Μοντέλα για τη Συμπεριφορά Κατασκευών από Οπλισμένο Σκυρόδεμα*, Diploma Thesis, National Technical University of Athens (in Greek).

Reshotkina S.S. (2015). *A thesis submitted to the Faculty of Graduate and Postdoctoral Affairs in partial fulfillment of the requirements for the degree of*, Ph.D. Dissertation, Carleton University Ottawa, Ontario.

Roh H., Reinhorn A.M., Lee J.S. (2012). *Power spread plasticity model for inelastic analysis of reinforced concrete structures*, Engineering Structures, 39, pp. 148–161.

Scott M.H., Filippou F.C. (2007). *Response Gradients for Nonlinear Beam-Column Elements under Large Displacements*, J Struct. Eng., 133(2), pp. 155–165.

Spyrakos C.C., Maniatakis C.A., Smyrou E., Psycharis I.N. (2012). *FRP Strengthened Brick-Infilled RC Frames: An Approach for their Proper Consideration in Design*, The Open Construction and Building Technology Journal (6), pp. 306–324.

Timurağaoğlu M. Ö., Doğangün A., Livaoğlu R. (2016). *Efficiency of the Numerical Models on Determination of Behavior of Brick Infilled RC Frames*, In Transactions of 12th International Congress on Advances in Civil Engineering, Turkey.

Tsiotsias K. (2016). *Σύγκριση ελαστοπλαστικών μεθόδων προσομοίωσης πλαισιακών φορέων*, Diploma Thesis, National Technical University of Athens (in Greek).

CHAPTER 7. APPENDIX

For the present postgraduate thesis, a code was developed in MATLAB -and integrated into the program ASDAP (Femlab)- in order to model the behaviour of a beam element with lumped plasticity. Below a part of this code is shown as example.

The MATLAB script for a Lumped Plasticity 2D Beam Element with Bilinear Material Law:

```
function [elements] = InitializeLumpedBeam(varargin)

elements=varargin{1};
elementsinput = elements;
nel = length(elements);
for i=1:nel
    eledef = elements{i};
    x1 = eledef{2};
    x2 = eledef{3};
    x3 = eledef{4};
    x4 = eledef{5};
    x5 = eledef{6};
    x7 = eledef{7};
    x10 = eledef{10};
    elements{i} = {'beam2DLumpedInitiation' x1 x2 x3 x4 x5 x7 x10};
end
nodes = varargin{2};
Analysis=varargin{3};

bcon = Analysis.bcon;
NodalLoads = Analysis.NodalLoads;
Analysis.Increments=1;

% Initialise the program, clear variables, set path of other .m scripts
ElemLoads={};
fixnode=[];

format long; %Show results with many decimals
rootdir=cd;
%cd('femlab')
a=dir(rootdir);
ic=0;
for i=1:length(a)
    if isdir(a(i).name)
        ic=ic+1;
        ffname{ic}=fullfile(rootdir,a(i).name);
```

```
        addpath(genpath(ffname{ic}));
    end
end

RunAnalysis

BeamForceMultpliers = fel;


max = 0;
nel = size(NodalLoads,1);
for i=1:nel
    for j=2:4
        if abs(NodalLoads(i,j))>max
            max = abs(NodalLoads(i,j));
        end
    end
end
if max~=0
    BeamForceMultpliers = (1/max)*fel;
end
BeamForceMultpliers = BeamForceMultpliers(:,1:6);
BeamForceMultpliers = BeamForceMultpliers';

elements = elementsinput;
nel = length(elements);

for i=1:nel
    eleused = elements{i};
    elementscase = eleused{1};
    if strcmp(elementscase,'beam2DLumped')

        % ELASTIC 2D BEAM with Lumped Plasticity
        Node_i = eleused{2};  %Start node
        Node_j = eleused{3};  %End  node

        Xi= nodes(Node_i,1);     %Start Node - X Coordinate
        Xj= nodes(Node_j,1);     %End Node - X Coordinate
        Yi= nodes(Node_i,2);     %Start Node - Y Coordinate
        Yj= nodes(Node_j,2);     %End Node - Y Coordinate

        L = sqrt((Xj-Xi)*(Xj-Xi) + (Yj-Yi)*(Yj-Yi));

        [T] = ElementTransformation(Xi,Yi,Xj,Yj);

        BeamForceMultpliers(:,i) = T*(BeamForceMultpliers(:,i));

```

```
        E = eleused{4};
        A = eleused{5};
        I = eleused{6};
        kspring1 = eleused{7};
        bs1 = eleused{8};
        My1 = eleused{9};
        kspring2 = eleused{10};
        bs2 = eleused{11};
        My2 = eleused{12};

        be  = [ -1     0        0;
            0   (1/L)    (1/L);
            0     1        0;
            1     0        0;
            0   (-1/L)    (-1/L);
            0     0        1];

        if bs1~=1
            if BeamForceMultpliers(3,i)~=0
                x1 = (My1/BeamForceMultpliers(3,i));
            else
                x1 = 0;
            end
            Fyy1 = BeamForceMultpliers(:,i)*x1;
            Sy1 = ((inv(transpose(be)*be))*(transpose(be)))*Fyy1;
            Felc1 = [L/(E*A)          0                    0;
                0        ((L/(3*E*I))+(1/kspring1))   L/(-6*E*I);
                0          L/(-6*E*I)            ((L/(3*E*I))+(1/
                   kspring2))];
            Kelc1 = inv(Felc1);
            vy1 = Kelc1\Sy1;
            vyy1 = vy1(2,1);
            kinit1 = (My1/vyy1);
            Fpl1 = [L/(E*A)          0                    0;
                0        ((L/(3*E*I))+(1/(bs1*kspring1)))   L/(-6*E*I);
                0          L/(-6*E*I)            ((L/(3*E*I))+(1/
                   kspring2))];
            Kplc1 = inv(Fpl1);
            vy11 = Kplc1\Sy1;
            vyy11 = vy11(2,1);
            kinit11 = (My1/vyy11);
            bm1 = (kinit11/kinit1);
            if BeamForceMultpliers(3,i)==0
                bm1=0;
                kinit1=0;
            end
```

```
        end
        %
        if bs2~=1
            if BeamForceMultpliers(6,i)~=0
                x2 = (My2/BeamForceMultpliers(6,i));
            else
                x2 = 0;
            end
            Fyy2 = BeamForceMultpliers(:,i)*x2;
            Sy2 = ((inv(transpose(be)*be))*(transpose(be)))*Fyy2;
            Felc2 = [L/(E*A)          0                    0;
                0              ((L/(3*E*I))+(1/kspring1))            L/(-6*E*I);
                0          L/(-6*E*I)            ((L/(3*E*I))+(1/kspring2))];
            Kelc2 = inv(Felc2);
            vy2 = Kelc2\Sy2;
            vyy2 = vy2(3,1);
            kinit2 =(My2/vyy2);
            %
            Fpl2 = [L/(E*A)          0                    0;
                0        ((L/(3*E*I))+(1/kspring1))   L/(-6*E*I);
                0          L/(-6*E*I)            ((L/(3*E*I))+(1/(bs2*kspring2)))];
            Kplc2 = inv(Fpl2);
            vy22 = Kplc2\Sy2;
            vyy22 = vy22(3,1);
            kinit22 =(My2/vyy22);
            bm2 =(kinit22/kinit2);
            if BeamForceMultpliers(6,i)==0
                bm2=0;
                kinit2=0;
            end
        end

        elements{i} = {'beam2DLumped' Node_i  Node_j  E  A I kspring1 bs1 kinit1 bm1 My1 kspring2 bs2 kinit2 bm2 My2};
    end
end


end
```

```
function [K_loc,K_gl_el,history] = beam2DLumped (icase,n,elements,m,history,nodes,udisp)

% ELASTIC 2D BEAM with Lumped Plasticity
```

```
Node_i = elements{2};  %Start node
Node_j = elements{3};  %End  node

Xi= nodes(Node_i,1);     %Start Node - X Coordinate
Xj= nodes(Node_j,1);     %End Node - X Coordinate
Yi= nodes(Node_i,2);     %Start Node - Y Coordinate
Yj= nodes(Node_j,2);     %End Node - Y Coordinate

L = sqrt((Xj-Xi)*(Xj-Xi) + (Yj-Yi)*(Yj-Yi));

E = elements{4};
A = elements{5};
I = elements{6};
kspring1 = elements{7};
bs1 = elements{8};
kinit1 = elements{9};
bm1 = elements{10};
My1 = elements{11};
kspring2 = elements{12};
bs2 = elements{13};
kinit2 = elements{14};
bm2 = elements{15};
My2 = elements{16};

%
udispprev = history{1};
Fprev = history{2};

%calculations
be  = [ -1      0        0;
    0   (1/L)    (1/L);
    0     1        0;
    1     0        0;
    0   (-1/L)    (-1/L);
    0     0        1];

[T] = ElementTransformation(Xi,Yi,Xj,Yj);
udisp = (T)*udisp;

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
Sprev = ((inv(transpose(be)*be))*(transpose(be)))*Fprev;
Mc2prev = Sprev(3,1);
Mc1prev = Sprev(2,1);
% Mc2prev = Fprev(6,1);
% Mc1prev = Fprev(3,1);

format long
```

```
if bs1~=1 && bs2~=1
        %previous step
        vprev = transpose(be)*udispprev;
        %current step
        v = transpose(be)*udisp;
        %nomos ulikou - 1
        vp1 = vprev(2,1);
        msp1 = Mc1prev;
        vc1 = v(2,1);
        [Mc1,ksc1] = bilinear(kinit1,bm1,My1,vp1,msp1,vc1);

        if ksc1==kinit1
            ks1=kspring1;
        elseif ksc1==kinit1*bm1
            ks1=kspring1*bs1;
        end

        %nomos ulikou - 2
        vp2 = vprev(3,1);
        msp2 = Mc2prev;
        vc2 = v(3,1);
        [Mc2,ksc2] = bilinear(kinit2,bm2,My2,vp2,msp2,vc2);

        if ksc2==kinit2
            ks2=kspring2;
        elseif ksc2==kinit2*bm2
            ks2=kspring2*bs2;
        end

        %stiffness matrix - beam
        Fnc3 = [L/(E*A)          0                    0;
            0                ((L/(3*E*I))+(1/ks1))           L/(-6*E*I)
                ;
            0          L/(-6*E*I)            ((L/(3*E*I))+(1/ks2))];
        Knc3 = inv(Fnc3);

        %topiko
        Kc3 = be * Knc3 * transpose(be);

        S = [((E*A)/L*v(1,1));  Mc1; Mc2];
        Fe=be*S;
end


if strcmp(icase,'stiffness')

    K_loc = Kc3;
```

```
    K_gl_el = T'* K_loc * T;

elseif strcmp(icase,'forces')

    Floc = Fe;
    Fgl=transpose(T)*Floc;

    K_loc = Floc;
    K_gl_el=Fgl; % auta einai dunameis, oxi K

elseif strcmp(icase,'Committed')
    K_loc = Kc3;
    K_gl_el = T'* K_loc * T;
    % sto telos kathe bimatos swzoume edw tis metakiniseis/dunameis
    history = {udisp Fe};


end
```

The following scripts -among others- were used in ASDAP (Femlab) and Opensees as executable files for the present postgraduate thesis:

- the Lumped Plasticity 2D Cantilever Beam with Bilinear Material Law
- the Lumped Plasticity 2D Clamped Beam with Bilinear Material Law
- the Lumped Plasticity 2D One Storey Frame with Bilinear Material Law
- the Lumped Plasticity 2D Two Storey Frame with Bilinear Material Law

The ASDAP (Femlab) script for a Lumped Plasticity 2D Cantilever Beam with Bilinear Material Law:

```
ClearAll

%INPUTS
L =  3;
E = 30000000;
A = 0.09290304;
I = 0.0012786629394432;
bs1 = 0.01;
My1 =  160.1;
bs2 = 0.01;
My2 =  160.1;
%
n = 10;
%single curvature: cur = 3 , double curvature: cur = 6
cur = 3;
km = (cur*E*I)/(L);
Kspring1 = km*(n+1);
Kspring2 = Kspring1;

Analysis.Type='StaticLoadControl';
Analysis.ndim=3;
Analysis.Increments=400;
Analysis.Iterations=1000;
Analysis.Tolerance=0.001;
Analysis.crv = [2,2];  % store node 2, dof 2 for the curve

% NODAL COORDINATES  X,  Y,
%%%%for cantilever
nodes= [0   0
         L   0];

%%%%for cantilever
bcon=[1  1  1  1];
```

```
%%%%for cantiliver
NodalLoads=[2 0 360 0];

Analysis.bcon = bcon;
Analysis.NodalLoads = NodalLoads;

% ELEMENT PROPETIES
%%%%for cantilever
elements{1} =  {'beam2DLumped' 1  2  E  A I Kspring1 bs1 My1 Kspring2
    bs2 My2};

PlotFrame(nodes,elements,bcon,'labels')

RunAnalysis

%%% FOR PLOTS
%%% DO SOME PLOTS
figure(20)
title('Q-U from femlab')
xlabel('U')
ylabel('Q')
hold on; grid on; box on;

plot(crv(:,1),crv(:,2),'.-');
```

The Opensees script for a Lumped Plasticity 2D Cantilever Beam with Bilinear Material Law:

```
wipe

%Inputs

model BasicBuilder -ndm 2 -ndf 3
source units_constants_metric.tcl

%gia suntetagmenes

set L1 [expr 3.];

%Create nodes
%    tag        X        Y
node  1       0.0      0.0
node  2       $L1      0.0

% Fix supports at base of columns
```

```
%    tag   DX   DY   DM
fix   1     1    1   1

set E [expr 30000000];
set A1 [expr 0.09290304];
set Iz [expr 0.0012786629394432];
set Mp1 [expr 160.1];
set Mp2 [expr 160.1];
% modification of the K values
set n 10;%10; % an arbitrary value
set Kmem [expr 3.0*$E*$Iz/$L1]
set Kspr [expr ($n+1)*$Kmem]

%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 1 $Mp1 $Kspr 0.000000001
%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 2 $Mp2 $Kspr 0.000000001


% auxiliary nodes in order to define the springs
node  11       0.0     0.0
node  22       $L1     0.0

%%morfwsh elements
geomTransf Linear 1;

%element elasticBeamColumn $eleTag $iNode $jNode $A $E $Iz $transfTag <-mass $massDens> <-cMass>
element elasticBeamColumn 1 11 22 $A1 $E $Iz 1
% zero length springs, rotational degree of freedom
% element zeroLength elem id, master node, slave node, M-f relationship,
%restrained dofs
element zeroLength 3 1 11 -mat 1 -dir 3
% equal dof for ux and uy of the spring nodes
% equalDOF master node, slave node, restrained dofs
equalDOF 1 11 1 2
%
element zeroLength 4 22 2 -mat 1 -dir 3
equalDOF 22 2 1 2

% Define loads
set P [expr (1)];

% Create a Plain load pattern with a Linear TimeSeries
pattern Plain 1 "Linear" {
      % Create nodal loads at node 2
```

```
        %    nd    FX   Y   M
        load 2    [expr 2*$P]  [expr 2*$P]    [expr 2*$P]
}

% Create a recorder to monitor nodal displacements
recorder Node -file plotU1Node2.txt -time -node 2 -dof 1 disp
recorder Node -file plotU2Node2.txt -time -node 2 -dof 2 disp
recorder Node -file plotU3Node2.txt -time -node 2 -dof 3 disp
recorder Node -file plotU1Node1.txt -time -node 1 -dof 1 disp
recorder Node -file plotU2Node1.txt -time -node 1 -dof 2 disp
recorder Node -file plotU3Node1.txt -time -node 1 -dof 3 disp


recorder Element -file ele1global1.txt -time -ele 1 globalForce
recorder Element -file ele1local1.txt -time -ele 1 localForce

%DIADIKASIA

initialize
system UmfPack
constraints Plain
numberer RCM

% Create the convergence test
%test NormUnbalance $tol    $iter <$pFlag> <$nType>
test NormUnbalance  0.001 1000

algorithm Newton

% Create the integration scheme
integrator LoadControl [expr 1]

% Create the analysis object
analysis Static

file delete filename.txt
print  filename.txt

% perform analysis
analyze 200
```

The ASDAP (Femlab) script for a Lumped Plasticity 2D Clamped Beam with Bilinear Material Law:

```
ClearAll

%INPUTS
```

```
L =  3;
E = 30000000;
A = 0.09290304;
I = 0.0012786629394432;
bs1 = 0.01;
My1 =  160.1;
bs2 = 0.01;
My2 =  160.1;
%
n = 10;
%single curvature: cur = 3 , double curvature: cur = 6
cur = 3;
km = (cur*E*I)/(L);
Kspring1 = km*(n+1);
Kspring2 = Kspring1;

Analysis.Type='StaticLoadControl';
Analysis.ndim=3;
Analysis.Increments=400;
Analysis.Iterations=1000;
Analysis.Tolerance=0.001;
Analysis.crv = [2,2];  % store node 2, dof 2 for the curve

% % NODAL COORDINATES  X,  Y,
%%%%for fixed ended beam / clamped beam
nodes= [0   0
        0.5*L   0
        2*L   0];

%%%%for fixed ended beam
bcon=[1  1  1  1
      3  1  1  1];


%%%%for fixed ended beam
NodalLoads=[2 0 360 0];

Analysis.bcon = bcon;
Analysis.NodalLoads = NodalLoads;

% ELEMENT PROPETIES
%%%%for fixed ended beam
elements{1} =  {'beam2DLumped' 1  2  E  A I Kspring1 bs1 My1 Kspring2
    bs2 My2};
elements{2} =  {'beam2DLumped' 2  3  E  A I Kspring2 bs2 My2 Kspring1
    bs1 My1};

```

```
PlotFrame(nodes,elements,bcon,'labels')

RunAnalysis

%% FOR PLOTS
%% DO SOME PLOTS
figure(20)
title('Q-U from femlab')
xlabel('U')
ylabel('Q')
hold on; grid on; box on;

plot(crv(:,1),crv(:,2),'.-');
```

The Opensees script for a Lumped Plasticity 2D Clamped Beam with Bilinear Material Law:

```
wipe

%Inputs

model BasicBuilder -ndm 2 -ndf 3
source units_constants_metric.tcl

%gia suntetagmenes

set L1 [expr 3.];

%Create nodes
%    tag        X        Y
node  1       0.0      0.0
node  2       $L1      0.0
node  3    [expr 2*$L1]    0.0


% Fix supports at base of columns
%    tag   DX   DY   DM
fix   1     1    1   1
fix   3     1    1   1

set E [expr 30000000];
set A1 [expr 0.09290304];
set Iz [expr 0.0012786629394432];
set Mp1 [expr 160.1];
set Mp2 [expr 160.1];
% modification of the K values
```

```
set n 10;%10; % an arbitrary value
set Kmem [expr 3.0*$E*$Iz/$L1]
set Kspr [expr ($n+1)*$Kmem]


%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 1 $Mp1 $Kspr 0.000000001
%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 2 $Mp2 $Kspr 0.000000001


% auxiliary nodes in order to define the springs
node  11       0.0     0.0
node  22       $L1     0.0
node  222       $L1     0.0
node  33    [expr 2*$L1]    0.0
%%morfwsh elements
geomTransf Linear 1;

%element elasticBeamColumn $eleTag $iNode $jNode $A $E $Iz $transfTag <-mass $massDens> <-cMass>
element elasticBeamColumn 1 11 22 $A1 $E $Iz 1
element elasticBeamColumn 5 2 33 $A1 $E $Iz 1
% zero length springs, rotational degree of freedom
% element zeroLength elem id, master node, slave node, M-f relationship,
%restrained dofs
element zeroLength 3 1 11 -mat 1 -dir 3
% equal dof for ux and uy of the spring nodes
% equalDOF master node, slave node, restrained dofs
equalDOF 1 11 1 2
%
element zeroLength 4 22 2 -mat 1 -dir 3
equalDOF 22 2 1 2
%
element zeroLength 6 3 33 -mat 1 -dir 3
equalDOF 3 33 1 2
%
element zeroLength 2 2 222 -mat 1 -dir 3
equalDOF 2 222 1 2

% Define loads
set P [expr (1)];

% Create a Plain load pattern with a Linear TimeSeries
pattern Plain 1 "Linear" {
      % Create nodal loads at node 2
```

```
        %    nd    FX  Y  M
        load 2    [expr 2*$P]  [expr 2*$P]    [expr 2*$P]
}

% Create a recorder to monitor nodal displacements
recorder Node -file plotU1Node2.txt -time -node 2 -dof 1 disp
recorder Node -file plotU2Node2.txt -time -node 2 -dof 2 disp
recorder Node -file plotU3Node2.txt -time -node 2 -dof 3 disp
recorder Node -file plotU1Node1.txt -time -node 1 -dof 1 disp
recorder Node -file plotU2Node1.txt -time -node 1 -dof 2 disp
recorder Node -file plotU3Node1.txt -time -node 1 -dof 3 disp


recorder Element -file ele1global1.txt -time -ele 1 globalForce
recorder Element -file ele1local1.txt -time -ele 1 localForce

%DIADIKASIA

initialize
system UmfPack
constraints Plain
numberer RCM

% Create the convergence test
%test NormUnbalance $tol    $iter <$pFlag> <$nType>
test NormUnbalance  0.001 1000

algorithm Newton

% Create the integration scheme
integrator LoadControl [expr 1]

% Create the analysis object
analysis Static

file delete filename.txt
print  filename.txt

% perform analysis
analyze 200

%OUTPUTS
print -node 2
```

The ASDAP (Femlab) script for a Lumped Plasticity 2D One Storey Frame with Bilinear Material Law:

```
ClearAll

%INPUTS
L =  3;
E = 30000000;
A = 0.09290304;
I = 0.0012786629394432;
bs1 = 0.01;
My1 =  160.1;
bs2 = 0.01;
My2 =  160.1;
%
n = 10;
%single curvature: cur = 3 , double curvature: cur = 6
cur = 3;
km = (cur*E*I)/(L);
Kspring1 = km*(n+1);
Kspring2 = Kspring1;

Analysis.Type='StaticLoadControl';
Analysis.ndim=3;
Analysis.Increments=400;
Analysis.Iterations=1000;
Analysis.Tolerance=0.001;
Analysis.crv = [2,1];  % store node 2, dof 2 for the curve

% % NODAL COORDINATES  X,  Y,
%%%for frame
nodes= [0   0
         0   L
         L   L
         L   0];


%%%%for frame & double frame
bcon=[1  1  1  1
       4  1  1  1];

%%%%for frame
NodalLoads=[2 240 0 0];


Analysis.bcon = bcon;
Analysis.NodalLoads = NodalLoads;

% ELEMENT PROPETIES
%%%%for frame
```

```
elements{1} =  {'beam2DLumped' 1  2  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{2} =  {'beam2DLumped' 2  3  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{3} =  {'beam2DLumped' 3  4  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};


PlotFrame(nodes,elements,bcon,'labels')

RunAnalysis

%% FOR PLOTS
%% DO SOME PLOTS
figure(20)
title('Q-U from femlab')
xlabel('U')
ylabel('Q')
hold on; grid on; box on;

plot(crv(:,1),crv(:,2),'.-');
```

The Opensees script for a Lumped Plasticity 2D One Storey Frame with Bilinear Material Law:

```
wipe

%Inputs

model BasicBuilder -ndm 2 -ndf 3
source units_constants_metric.tcl

%gia suntetagmenes

set L1 [expr 3.];

%Create nodes
%    tag        X        Y
node  1       0.0      0.0
node  2       0.0      $L1
node  3       [expr $L1]      $L1
node  4       $L1      0.0


% Fix supports at base of columns
%    tag   DX   DY   DM
fix   1     1    1   1
```

```
fix   4     1    1   1

set E [expr 30000000];
set A1 [expr 0.09290304];
set Iz [expr 0.0012786629394432];
set Mp1 [expr 160.1];
set Mp2 [expr 160.1];
% modification of the K values
set n 10;%10; % an arbitrary value
set Kmem [expr 3.0*$E*$Iz/$L1]
set Kspr [expr ($n+1)*$Kmem]


%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 1 $Mp1 $Kspr 0.01
%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 2 $Mp2 $Kspr 0.01

% auxiliary nodes in order to define the springs
node 11 0.0 0.0
node 22 0.0 $L1
node 222 0.0 $L1
node 333 [expr $L1] $L1
node 33 [expr $L1] $L1
node 44 [expr $L1] 0.0

%%morfwsh elements
geomTransf Linear 1;

%element elasticBeamColumn $eleTag $iNode $jNode $A $E $Iz $transfTag
    <-mass $massDens> <-cMass>
element elasticBeamColumn 1 11 22 $A1 $E $Iz 1
element elasticBeamColumn 5 222 333 $A1 $E $Iz 1
element elasticBeamColumn 7 33 44 $A1 $E $Iz 1
% zero length springs, rotational degree of freedom
% element zeroLength elem id, master node, slave node, M-f relationship
    ,
%restrained dofs
element zeroLength 3 1 11 -mat 1 -dir 3
% equal dof for ux and uy of the spring nodes
% equalDOF master node, slave node, restrained dofs
equalDOF 1 11 1 2
%
element zeroLength 4 22 2 -mat 2 -dir 3
equalDOF 22 2 1 2
%
element zeroLength 6 33 3 -mat 1 -dir 3
```

```
equalDOF 33 3 1 2
%
element zeroLength 8 4 44 -mat 1 -dir 3
equalDOF 4 44 1 2
%
element zeroLength 9 2 222 -mat 1 -dir 3
equalDOF 2 222 1 2
%
element zeroLength 10 3 333 -mat 1 -dir 3
equalDOF 3 333 1 2

% Define loads
set P [expr 4.8];

% Create a Plain load pattern with a Linear TimeSeries
pattern Plain 1 "Linear" {
      % Create nodal loads at node 2
        %    nd    FX  Y  M
        load 2    [expr 0.333*$P]  [expr 0*$P]    [expr 0.*$P]
}

% Create a recorder to monitor nodal displacements
recorder Node -file plotU1Node1.txt -time -node 11 -dof 1 disp
recorder Node -file plotU2Node1.txt -time -node 11 -dof 2 disp
recorder Node -file plotU3Node1.txt -time -node 11 -dof 3 disp

recorder Node -file plotU1Node2.txt -time -node 22 -dof 1 disp
recorder Node -file plotU2Node2.txt -time -node 22 -dof 2 disp
recorder Node -file plotU3Node2.txt -time -node 22 -dof 3 disp

recorder Element -file ele1global1.txt -time -ele 1 globalForce
recorder Element -file ele1local1.txt -time -ele 1 localForce

%DIADIKASIA

initialize
system UmfPack
constraints Plain
numberer RCM

% Create the convergence test
%test NormUnbalance $tol    $iter <$pFlag> <$nType>
test NormUnbalance  1.0e-3  100

algorithm Newton

% Create the integration scheme
```

```
integrator LoadControl [expr (1)]

%%integrator DisplacementControl $node $dof $incr
%integrator DisplacementControl   2     2  [expr 0.0002]

% Create the analysis object
analysis Static

file delete filename.txt
print  filename.txt

% perform analysis
analyze 150
```

The ASDAP (Femlab) script for a Lumped Plasticity 2D Two Storey Frame with Bilinear Material Law:

```
ClearAll

%INPUTS
L =  3;
E = 30000000;
A = 0.09290304;
I = 0.0012786629394432;
bs1 = 0.01;
My1 =  160.1;
bs2 = 0.01;
My2 =  160.1;
%
n = 10;
%single curvature: cur = 3 , double curvature: cur = 6
cur = 3;
km = (cur*E*I)/(L);
Kspring1 = km*(n+1);
Kspring2 = Kspring1;

Analysis.Type='StaticLoadControl';
Analysis.ndim=3;
Analysis.Increments=400;
Analysis.Iterations=1000;
Analysis.Tolerance=1;
Analysis.crv = [2,1];  % store node 2, dof 1 for the curve

% NODAL COORDINATES  X,  Y,
%%%%for double frame
nodes= [0   0
        0   L
```

```
          L    L
          L    0
          0    2*L
          L    2*L];


%%%%for frame & double frame
bcon=[1   1   1   1
       4   1   1   1];


%%%%for double frame
NodalLoads=[5 0.667*190 0 0
            2 0.333*190 0 0];

Analysis.bcon = bcon;
Analysis.NodalLoads = NodalLoads;

% ELEMENT PROPETIES
%%%%for double frame
elements{1} =  {'beam2DLumped' 1  2  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{2} =  {'beam2DLumped' 2  3  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{3} =  {'beam2DLumped' 3  4  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{4} =  {'beam2DLumped' 2  5  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{5} =  {'beam2DLumped' 5  6  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};
elements{6} =  {'beam2DLumped' 6  3  E  A I Kspring1 bs1 My1 Kspring2 bs2 My2};

PlotFrame(nodes,elements,bcon,'labels')

RunAnalysis

%% FOR PLOTS
%% DO SOME PLOTS
figure(20)
title('Q-U from femlab')
xlabel('U')
ylabel('Q')
hold on; grid on; box on;

plot(crv(:,1),crv(:,2),'.-');
```

The Opensees script for a Lumped Plasticity 2D Two Storey Frame with Bilinear Material Law:

```
wipe

%Inputs

model BasicBuilder -ndm 2 -ndf 3
source units_constants_metric.tcl

%gia suntetagmenes

set L1 [expr 3.];

%Create nodes
%    tag        X       Y
node  1       0.0     0.0
node  2       0.0      $L1
node  3       [expr $L1]     $L1
node  4       $L1     0.0
node  5       0.0    [expr 2*$L1]
node  6       [expr $L1]     [expr 2*$L1]

% Fix supports at base of columns
%    tag   DX   DY   DM
fix   1     1    1   1
fix   4     1    1   1

set E [expr 30000000];
set A1 [expr 0.09290304];
set Iz [expr 0.0012786629394432];
set Mp1 [expr 160.1];
set Mp2 [expr 160.1];
% modification of the K values
set n 10;%10; % an arbitrary value
set Kmem [expr 3.0*$E*$Iz/$L1]
set Kspr [expr ($n+1)*$Kmem]

%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 1 $Mp1 $Kspr 0.01
%uniaxialMaterial Steel01 $matTag $Fy $E0 $b <$a1 $a2 $a3 $a4>
uniaxialMaterial Steel01 2 $Mp2 $Kspr 0.01

% auxiliary nodes in order to define the springs
node 11 0.0 0.0
node 22 0.0 $L1
node 222 0.0 $L1
```

```
node 2222 0.0 $L1
node 333 [expr $L1] $L1
node 33 [expr $L1] $L1
node 3333 [expr $L1] $L1
node 44 [expr $L1] 0.0
node  55       0.0    [expr 2*$L1]
node  555       0.0    [expr 2*$L1]
node  66       [expr $L1]     [expr 2*$L1]
node  666      [expr $L1]     [expr 2*$L1]
%%morfwsh elements
geomTransf Linear 1;

%element elasticBeamColumn $eleTag $iNode $jNode $A $E $Iz $transfTag
    <-mass $massDens> <-cMass>
element elasticBeamColumn 1 11 22 $A1 $E $Iz 1
element elasticBeamColumn 2 2222 55 $A1 $E $Iz 1
element elasticBeamColumn 5 222 333 $A1 $E $Iz 1
element elasticBeamColumn 7 33 44 $A1 $E $Iz 1
element elasticBeamColumn 12 3333 66 $A1 $E $Iz 1
element elasticBeamColumn 11 555 666 $A1 $E $Iz 1
% zero length springs, rotational degree of freedom
% element zeroLength elem id, master node, slave node, M-f relationship
    ,
%restrained dofs
element zeroLength 3 1 11 -mat 1 -dir 3
% equal dof for ux and uy of the spring nodes
% equalDOF master node, slave node, restrained dofs
equalDOF 1 11 1 2
%
element zeroLength 4 22 2 -mat 2 -dir 3
equalDOF 22 2 1 2
%
element zeroLength 6 33 3 -mat 1 -dir 3
equalDOF 33 3 1 2
%
element zeroLength 8 4 44 -mat 1 -dir 3
equalDOF 4 44 1 2
%
element zeroLength 9 2 222 -mat 1 -dir 3
equalDOF 2 222 1 2
%
element zeroLength 10 3 333 -mat 1 -dir 3
equalDOF 3 333 1 2
%
element zeroLength 13 2 2222 -mat 1 -dir 3
equalDOF 2 2222 1 2
%
```

```
element zeroLength 14 3 3333 -mat 1 -dir 3
equalDOF 3 3333 1 2
%
element zeroLength 15 55 5 -mat 1 -dir 3
equalDOF 55 5 1 2
%
element zeroLength 16 5 555 -mat 1 -dir 3
equalDOF 5 555 1 2
%
element zeroLength 17 66 6 -mat 1 -dir 3
equalDOF 66 6 1 2
%
element zeroLength 18 6 666 -mat 1 -dir 3
equalDOF 6 666 1 2


% Define loads
set P [expr 4.8];

% Create a Plain load pattern with a Linear TimeSeries
pattern Plain 1 "Linear" {
      % Create nodal loads at node 2
        %    nd    FX  Y  M
        load 2    [expr 0.333*$P]  [expr 0*$P]    [expr 0.*$P]
        load 5    [expr 0.667*$P]  [expr 0*$P]    [expr 0.*$P]
}

% Create a recorder to monitor nodal displacements
recorder Node -file plotU1Node1.txt -time -node 11 -dof 1 disp
recorder Node -file plotU2Node1.txt -time -node 11 -dof 2 disp
recorder Node -file plotU3Node1.txt -time -node 11 -dof 3 disp

recorder Node -file plotU1Node2.txt -time -node 22 -dof 1 disp
recorder Node -file plotU2Node2.txt -time -node 22 -dof 2 disp
recorder Node -file plotU3Node2.txt -time -node 22 -dof 3 disp

recorder Element -file ele1global1.txt -time -ele 1 globalForce
recorder Element -file ele1local1.txt -time -ele 1 localForce

%DIADIKASIA

initialize
system UmfPack
constraints Plain
numberer RCM

% Create the convergence test
```

```
%test NormUnbalance $tol    $iter <$pFlag> <$nType>
test NormUnbalance  1.0e-3  100

algorithm Newton

% Create the integration scheme
integrator LoadControl [expr (1)]

%%integrator DisplacementControl $node $dof $incr
%integrator DisplacementControl   2     2  [expr 0.0002]

% Create the analysis object
analysis Static

file delete filename.txt
print  filename.txt

% perform analysis
analyze 150
```